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

Mechanical Help

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

Release 2025 R1
January 2025

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

Copyright and Trademark Information

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

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

Table of Contents **Contents-1**

1 - Welcome to Mechanical Help **1-1**

Introduction to Mechanical 1-3

Inserting a Mechanical Design 1-3

Choose Mechanical Solution Type 1-5

Example Projects 1-6

Mechanical Example Projects 1-7

Modal Example – Motor Housing 1-8

Steady-State Thermal Example – Transistor with Heat Sink 1-9

Transient Thermal Example – Current Regulator 1-12

Result Images 1-17

Structural Example – PCB Stress 1-22

Structural Example – Substrate Warpage 1-25

Finding Information in the Online Help 1-30

Using the Search Function in the Help 1-30

Help Conventions 1-35

Getting Help from Ansys Technical Support 1-37

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

Ansys Electronics Desktop Student 2-3

HFSS Student Limitations 2-4

Q3D Extractor and 2D Extractor Student Limitations 2-4

Circuit Limitations 2-4

Maxwell and RMxprt Student Limitations 2-4

Icepak Student Limitations 2-5

System Requirements 2-5

Launching Ansys Electronics Desktop 2-5

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Obtaining Information about the Software and Release 2-7
Ansys Product Improvement Program 2-8
Ansys Electronics Desktop Overview 2-11
Choosing a Color Scheme [Beta] 2-12
Working with Ribbons 2-14
IC Mode Layout Ribbon 2-17
Ansys Electronics Desktop Windows 2-17
Showing and Hiding Windows 2-18
Auto Hiding Windows 2-19
Moving and Resizing Windows 2-20
Project Manager Window 2-22
Working with the Project Tree 2-22
Setting the Project Tree to Expand Automatically 2-22
Viewing Material Definitions 2-23
Viewing Ansys Electronics Desktop Design Details 2-24
Message Manager Window 2-25
Setting the Message Manager to Open Automatically 2-26
Showing New Messages 2-26
Automatically Expanding the Message Manager Tree 2-27
Action Messages 2-27
Clearing Messages 2-28
Hiding the Message Manager Window until Messages Appear 2-28
Progress Window 2-28
Stopping or Aborting Simulation Progress 2-29
Properties Window 2-30
Opening the Properties Window 2-31
Setting the Properties Window to Open Automatically 2-31
Showing and Hiding the Properties Window 2-32

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Object Attributes Using the Properties Window 2-32
Modifying Object Command Properties Using the Properties Window 2-32
Ansys Electronics Desktop Menus 2-33
Top Menu Bar 2-34
File Menu 2-35
Edit Menu 2-36
Edit Menu for 3D Modeler 2-36
Edit Menu for Report Window 2-39
View Menu 2-40
Basic View Menu 2-40
3D Modeler View Menu 2-41
Project Menu 2-46
Editor and Design Specific Menus 2-47
Tools Menu 2-47
Adding External Tools to the Tools Menu 2-50
Window Menu 2-52
Help Menu 2-55
Context-Sensitive Help 2-56
Obtaining Information about the Software and Release 2-56
Shortcut Menus 2-56
Shortcut Menu in the 3D Modeler Window 2-57
Shortcut Menus in the Project Manager Window 2-58
Customizing Ansys Electronics Desktop Menus 2-60
Ansys Electronics Desktop Design Area 2-69
3D Modeler Window 2-70
Report Window 2-70
Ansys Electronics Desktop History Tree 2-71
Organizing the History Tree 2-74

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controlling the View of Objects in the History Tree ... 2-75
Group Commands for Modeler Objects ... 2-75
Selecting Objects in the History Tree ... 2-80
Viewing and Editing Commands on History Tree Objects ... 2-82
Upgrading Version in the History Tree ... 2-83
Ansys Electronics Desktop Status Bar ... 2-84
Keyboard Shortcuts ... 2-85
Desktop Shortcuts ... 2-85
3D Modeler Window Shortcuts ... 2-86
Report Shortcuts ... 2-87
Choosing the View Navigation Options ... 2-88
Custom Keyboard Shortcuts ... 2-89
Auto-Complete for Variables and Properties in Electronics Desktop ... 2-91
Using the Password Manager to Control Access to Resources ... 2-95
Running Ansys Electronics Desktop from a Command Line ... 2-96
Command-line Syntax ... 2-96
Run Commands ... 2-97
Job Management from the Command Line ... 2-103
Run Command Examples ... 2-103
Specifying Project Files ... 2-103
Options ... 2-104
-Batchoptions ... 2-105
-Batchoptions Examples ... 2-106
Export Options Files ... 2-107
Examples and Further Explanations of -batchoptions Use ... 2-107
For -batchoptions Use: Project Directory and Lib Paths ... 2-109
For -batchoptions Use: TempDirectory ... 2-110
For -batchoptions Use: Various Desktop Settings ... 2-110

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running Ansys Electronics Desktop from a Windows Remote Terminal 2-113
Using Windows HPC Commands 2-114
Customizing Ansys Electronics Desktop with Ansys ACT 2-114
Installing PyAEDT (Beta) 2-115
Debug Logging 2-116
3 - Working with Ansys Electronics Desktop Projects 3-1
Ansys Electronics Desktop Files 3-1
Setting Project Options 3-2
Setting General Options 3-3
General Options: Desktop Configuration 3-5
General Options: Project Options 3-7
General Options: Miscellaneous Options 3-8
General Options: User Interface Options 3-9
General Options: Directories Options 3-13
General Options: Desktop Performance 3-14
General Options: Default Units 3-17
General Options: Remote Analysis Options 3-18
General Options: Component Libraries Options 3-19
Setting Mechanical Options 3-21
Setting 3D Modeler Options 3-23
3D Modeler Options: Drawing Options 3-24
3D Modeler Options: Operation Options 3-25
3D Modeler Options: Snap Options 3-27
3D Modeler Options: Display Options 3-28
Display Options: General Options 3-29
Display Options: Rendering Options 3-31
Display Options: History Tree Options 3-33
3D Modeler Options: Group Options 3-34

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Options: Advanced Options 3-34
Setting Reporter Options 3-35
Reporter Options: Report Setup Options 3-36
Reporter Options: Fields Reporter Options 3-37
Reporter Options: Report2D Options 3-37
Report2D Options: Curve 3-38
Report2D Options: Axis 3-38
Report2D Options: Grid 3-39
Report2D Options: Header 3-39
Report2D Options: Note 3-39
Report2D Options: Legend 3-40
Report2D Options: Marker 3-40
Report2D Options: Marker Table 3-41
Report2D Options: X-Y Markers 3-41
Report2D Options: Stacked 3-42
Report2D Options: Digital 3-43
Report2D Options: General 3-43
Curve Tooltip 3-43
Clipboard Option 3-44
Format 3-44
Report2D Options: Table 3-44
Header Row 3-44
Format 3-45
Copy to Clipboard 3-45
Setting HPC and Analysis Options 3-45
Configurations Tab 3-46
Selecting an Available Configuration 3-47
Options Tab 3-48

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Licensing Settings Tool ... 3-51
Setting Options via Configuration Files ... 3-53
Behavior Examples ... 3-54
Rules for Modifying Option Settings ... 3-54
Configuration File Locations ... 3-54
Products with Multiple Desktop Versions ... 3-55
Table of Directories and Files ... 3-55
Setting or Removing Option Values in Configuration Files: UpdateRegistry Command ... 3-56
UpdateRegistry -Set Command ... 3-57
UpdateRegistry -Get Command ... 3-57
UpdateRegistry -GetKeys Command ... 3-58
UpdateRegistry -Delete Command ... 3-59
UpdateRegistry -FromFile Command ... 3-60
UpdateRegistry File Format ... 3-60
Example Uses for Export Options Features ... 3-61
Options that Apply to All Users ... 3-61
Example: Searching for a Registry Key Pathname ... 3-62
Example: Setting an Installation Default Value ... 3-62
Example: Setting a Host-Dependent Default Value ... 3-63
Example: Reverting from a User-Defined Option Value to the Administrator Default ... 3-63
User Options and the UpdateRegistry Tool ... 3-64
Example: Removing a Host-Dependent User Option Setting ... 3-65
Example: Getting a Value from a Specific Configuration File ... 3-65
Example: Getting a Value Using Precedence Rules ... 3-66
Example: Adding a Host-Independent User Option Setting ... 3-66
Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml ... 3-66

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example: Change a Newly Set Registry Key ... 3-66
Setting the Temporary Directory ... 3-67
Temporary Directory Configuration File Format ... 3-69
UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files ... 3-70
Batchoptions Command Line Examples ... 3-71
Batchoptions File Format ... 3-71
Example -BatchOptions with -Remote (Windows) ... 3-72
Example -Batchsolve with -Machinelist (Windows) ... 3-73
Example -Batchsolve with -Machinelist (Linux) ... 3-73
Example -Batchsolve for Local (Windows) ... 3-75
Batch Options and Analysis Configurations in the Registry ... 3-75
Managing Projects and Designs ... 3-79
Opening Projects ... 3-80
Opening Recent Projects ... 3-82
Opening Example Projects ... 3-83
Creating Projects ... 3-83
Creating a Target Design from 3D Designs ... 3-85
Saving Projects ... 3-86
Path Name Length Issues for Windows ... 3-87
Changing Auto-Save Settings ... 3-88
Save Before Solving Option ... 3-89
Recovering Project Data from an Auto-Save File ... 3-90
Importing and Exporting Projects and Data ... 3-90
Importing Files ... 3-90
Importing 2D Model (GDSII) Files ... 3-92
Creating GDSII Layer Mapping Files ... 3-95
Tech File Colors with RGB Values ... 3-96

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing DXF and DWG Format Files ... 3-110
Importing 3D Model Files ... 3-113
Exporting Files ... 3-115
Exporting 2D Model Files ... 3-115
Exporting 3D Model Files ... 3-116
GDSII Files ... 3-116
Parasolid Files ... 3-117
Exporting Graphics Files ... 3-118
Exporting Options Files ... 3-120
Exporting Options Files Using the Desktop UI ... 3-120
Exporting Options Files Using a Script ... 3-120
Validating Projects ... 3-121
Modeler Validation Settings ... 3-122
Copying and Pasting a Project or Design ... 3-123
Renaming Projects ... 3-124
Archiving Projects ... 3-124
Restoring an Archived Project ... 3-127
Deleting a Project or Design ... 3-131
Setting Read Only Designs ... 3-131
Updating Design Components ... 3-132
Undoing and Redoing Commands ... 3-133
Inserting a Documentation File ... 3-134
Working with Design Notes ... 3-134
Printing ... 3-135
Example Projects ... 3-136
Mechanical Example Projects ... 3-137
Working with Variables ... 3-138
Working with Project Variables ... 3-139

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Project Variable ... 3-140
Importing and Exporting Project Variables ... 3-143
Deleting a Project Variable ... 3-143
Editing a Project Variable ... 3-143
Adding a Project Variable Array ... 3-144
List of Intrinsic Variables ... 3-148
List of Constants ... 3-149
Working with Design Variables ... 3-150
Defining Local Variables at the Design Level ... 3-151
Adding a Local Variable ... 3-151
Deleting a Local Variable ... 3-154
Editing a Local Variable ... 3-155
Overriding a Local Variable ... 3-155
Defining Parameter Defaults at the Design Level ... 3-155
Adding a Parameter Default ... 3-155
Deleting a Parameter Default ... 3-157
Editing a Parameter Default ... 3-157
Overriding a Parameter Default ... 3-157
Adding a Design Variable Array ... 3-158
Converting Variables and Parameter Defaults ... 3-163
Fixed vs. Non-Fixed Variables ... 3-163
Non-Fixed Variable Sweeps ... 3-164
Orphaned Sweeps ... 3-165
Choosing a Variable to Optimize ... 3-165
Choosing a Variable to Tune ... 3-167
Including a Variable in a Sensitivity Analysis ... 3-168
Including a Variable in a Statistical Analysis ... 3-169
Copying and Pasting a Variables List ... 3-170

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Datasets ... 3-171
Adding Datasets ... 3-171
Importing Datasets ... 3-175
Editing Datasets ... 3-177
Cloning Datasets ... 3-179
Exporting Datasets ... 3-182
Removing Datasets ... 3-183
Changing Dataset Plot Properties ... 3-184
Using SheetScan ... 3-187
SheetScan Menus and Settings ... 3-188
SheetScan View Options ... 3-188
SheetScan Right-click Menu ... 3-189
SheetScan Settings ... 3-189
Loading a Datasheet Picture into SheetScan ... 3-190
Loading a Datasheet Picture Directly ... 3-190
Loading a Datasheet Picture Using the HTML Viewer ... 3-190
Defining a SheetScan Coordinate System ... 3-191
Defining a Characteristic Curve in SheetScan ... 3-192
Performing Operations on SheetScan Curves ... 3-193
Checking for Monotonicity in X ... 3-194
Importing Characteristic Data into SheetScan ... 3-195
Exporting SheetScan Data ... 3-197
Defining an Expression ... 3-198
Valid Operators for Expressions ... 3-198
Using Intrinsic Functions in Expressions ... 3-199
clp Formula ... 3-200
Using Piecewise Linear Functions in Expressions ... 3-201
Using Dataset Expressions ... 3-202

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Handling Delta Temperature Units in Expressions ... 3-204
Defining Mathematical Functions ... 3-206
Meshing Quick Links ... 3-206
Modeling Quick Links ... 3-207
Shortcut Keys ... 3-208
4 - Mechanical Licensing Requirements ... 4-1
Choose Structural/Modal License ... 4-1
5 - Supported Geometry ... 5-1
6 - Modal Solutions ... 6-1
Assigning Modal Materials ... 6-2
Material Selection ... 6-3
Boundaries (Modal) ... 6-9
General Usage Notes Common to all Boundaries ... 6-9
Solution Setup (Modal) ... 6-10
How to Set Up a Modal Solution: ... 6-11
Results (Modal) ... 6-12
Modal Report ... 6-16
Creating a Modal Report ... 6-19
Fields Report ... 6-26
Creating a Fields Report ... 6-30
Field Overlay and Animation ... 6-36
7 - Thermal Solutions ... 7-1
Transient Thermal Solutions ... 7-2
Video: Transient Thermal Analysis with Trace Mapping ... 7-3
Using Datasets to Define Transient Parameters ... 7-9
Piecewise Constant (pwc) Datasets ... 7-13
Thermal Monitor ... 7-15
To Assign Thermal Monitor Points ... 7-16

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Display Thermal Monitor Results 7-18
Other Available Thermal Monitor Commands 7-20
Assigning Initial Temperatures 7-21
Transient Thermal Restart Analyses 7-23
Restart Requirements and Limitations: 7-24
Restarting from a Steady-State Thermal Solution 7-25
Verifying Restart Data Linking 7-28
Restarting from a Separate Transient Thermal Solution 7-29
Verifying Restart Data Linking 7-31
Extending the Current Transient Thermal Solution 7-32
Setup Link (Transient Thermal Restart) 7-34
How to Set Up Transient Restart Linking 7-34
Cached Link Data and Solution Invalidation (Transient Thermal Restart) 7-40
Assigning Thermal Materials 7-41
Material Selection 7-43
Shell Objects 7-48
Boundaries (Thermal) 7-50
Excitations (Thermal) 7-51
Thermal Contact 7-51
Thermal Contact Video 7-53
Thermal Resistance Types 7-58
Contact Guidelines and Examples 7-59
Solid-to-Solid Contact Examples 7-61
Shell-to-Solid Contact Examples 7-63
How to Assign Thermal Contact 7-69
Solution Setup (Thermal) 7-72
How to Set Up a Thermal Solution: 7-77
Convergence Tab 7-78

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Conditions Tab 7-81
Save Fields Tab 7-82
Two-Way Coupling 7-83
Two-Way Coupling Requirements 7-84
Two-Way Coupling Workflow 7-85
Solution Invalidation 7-90
Results (Thermal) 7-90
Fields Report 7-92
Creating a Fields Report 7-98
Monitor Report 7-106
Creating a Monitor Report 7-109
Field Overlay and Animation 7-114
To Produce a Field Overlay Plot and Animate it: 7-114
Creating a Fields Summary 7-120
Fields Summary Procedure 7-121
8 - Structural Solutions 8-1
Assigning Structural Materials 8-2
Material Selection 8-3
Boundaries (Structural) 8-10
General Usage Notes Common to all Boundaries 8-10
Excitations (Structural) 8-11
Specifying the Reference Temperature 8-11
Solution Setup (Structural) 8-13
How to Set Up a Structural Solution: 8-17
Results (Structural) 8-18
Fields Report 8-22
Creating a Fields Report 8-27
Field Overlay and Animation 8-32

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Produce a Field Overlay Plot and Animate it: 8-32
Creating a Fields Summary 8-37
Fields Summary Procedure 8-38
9 - Mechanical Boundaries and Excitations 9-1
Boundaries 9-1
Convection 9-1
How to Assign a Convection Boundary: 9-3
Setup Link (Convection) 9-8
How to Set Up the Icepak Link 9-10
Ambient Temperature Guidelines 9-15
Cylindrical Support 9-17
How to Assign a Cylindrical Support Boundary: 9-19
Fixed Support 9-21
How to Assign a Fixed Support Boundary: 9-22
Frictionless Support 9-23
How to Assign a Frictionless Support Boundary: 9-23
Rotating Fluid 9-24
Background 9-24
Equivalent Thermal Conductivity (kequiv) 9-25
Prerequisites for Assigning a Rotating Fluid Boundary: 9-26
How to Assign a Rotating Fluid Boundary: 9-28
Mesh Considerations 9-31
Prerequisites for using the Clone Mesh Option 9-34
Prerequisites for using the Rotational Layer Based Mesh Operation 9-35
Temperature 9-35
How to Assign a Temperature Boundary: 9-35
Excitations 9-37
Displacement 9-37

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Assign a Displacement Excitation: 9-38
EM Loss 9-43
How to Assign an EM Loss Excitation: 9-47
Verifying EM Loss Linking 9-56
Best Practices: 9-57
Cached Link Data and Solution Invalidation 9-58
Force 9-58
How to Assign a Force Excitation: 9-59
Setup Link (Force) 9-61
How to Set Up the Force Link 9-62
Heat Flux 9-68
How to Assign a Heat Flux Excitation: 9-69
Heat Generation 9-73
How to Assign a Heat Generation Excitation: 9-73
Pressure 9-75
How to Assign a Pressure Excitation: 9-75
Thermal Condition 9-78
How to Assign a Thermal Condition Excitation: 9-78
Verifying Thermal Condition Linking 9-82
Best Practices: 9-83
Cached Link Data and Solution Invalidation 9-84
Setup Link (Thermal Condition) 9-85
How to Set Up Temperature Linking 9-86
Editing Boundaries, Excitations and Other Assignments 9-92
Assigning a Boundary or Excitation to a Portion of a Face 9-94
Imprinting on a Flat Face 9-95
Imprinting on a Curved Face 9-97
10 - Layout Components in Mechanical [Beta] 10-1

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Layout Component ... 10-8
Inserting a Layout Component Instance ... 10-23
Instance Variation ... 10-24
Replacing or Editing Layout Components ... 10-27
Edit/Replace Procedure ... 10-28
Editing a Layout Component Definition ... 10-29
Layout Component Properties ... 10-35
General Tab ... 10-35
Parameters Tab ... 10-36
Component Data Tab ... 10-37
Visualization Tab ... 10-38
Technical Notes: Metal Fraction Map ... 10-41
11 - Assigning Materials ... 11-1
Searching for Materials ... 11-2
Searching by Material Property ... 11-5
Adding New Materials ... 11-7
Assigning Material Property Types ... 11-11
Defining Anisotropic Tensors ... 11-12
Assigning Anisotropic Tensors ... 11-13
Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS ... 11-14
Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS ... 11-18
Defining Variable Material Properties ... 11-22
Specifying Thermal Modifiers ... 11-22
Defining Material Properties as Expressions ... 11-31
Defining Functional Material Properties ... 11-32
Assigning Materials from the Object Properties Window ... 11-32

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing and Modifying Material Attributes 11-33
Validating Materials 11-35
Copying Materials 11-35
Removing Materials 11-36
Exporting Materials to a Library 11-36
Sorting Materials 11-36
Filtering Materials 11-37
Working with Material Libraries 11-38
Editing Libraries 11-39
12 - Drawing a Model 12-1
Setting Units of Measurement for the Model 12-1
Drawing Objects 12-3
Drawing a Line Segment 12-5
Drawing a Three-Point Arc Line 12-10
Drawing a Center-Point Arc Line 12-12
Drawing a Spline 12-16
Drawing a Polyline 12-19
Assigning a Cross Section and Dimensions to a Polyline 12-21
Inserting Line Segments 12-23
Drawing an Equation-Based Curve 12-24
Functions, Operators and Quantities for the Edit Equation Dialog Box 12-28
Drawing a Circle 12-29
Drawing an Ellipse 12-32
Drawing a Rectangle 12-36
Drawing a Regular Polygon 12-39
Drawing an Equation-Based Surface 12-42
Drawing a Sphere 12-46
Drawing a Cylinder 12-50

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Box 12-54
Drawing a Regular Polyhedron 12-58
Drawing a Cone 12-61
Drawing a Torus 12-64
Drawing a Helix 12-67
Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive 12-71
Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive 12-72
Drawing a Spiral 12-73
Drawing a Spiral Using User Defined Primitives 12-75
Drawing a Bondwire 12-76
Drawing a Point 12-78
Drawing a Plane 12-78
Creating Segmented Geometry 12-80
Segmented Objects 12-80
Drawing Non-Model Objects 12-81
Selecting Non-Model Drawing Mode 12-82
Changing an Object to Non Model 12-82
Drawing a Region 12-82
Drawing a SubRegion 12-85
Group Commands for Modeler Objects 12-90
Model Analysis 12-95
Analysis Options Dialog Box 12-96
Analyzing the Surface Mesh 12-97
Model Analysis Dialog Box 12-98
Objects Tab 12-99
Object Misalignment Tab 12-100

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Mesh (Single/Pairs) Tab 12-101
Mesh Feedback Tab 12-101
Design Settings 12-107
Design Settings for Mechanical 12-108
Model Preparation Commands 12-115
Simplify Command 12-115
Heal 12-123
Validation and Healing Options for Import File 12-125
Healing Non-Manifold Objects 12-125
Setting the Healing Options 12-127
Stitch Sheets 12-128
Align Selected Faces 12-129
Remove Selected Faces 12-130
Remove Selected Edges 12-131
Perform Explicit Subtractions 12-131
Creating a User-Defined Primitive 12-132
User Customization through User Defined Primitives (UDPs) 12-134
User Defined Model (UDM) 12-135
UDM Compared to User Defined Primitives 12-136
Insert UDM Command on Draw Menu 12-137
UDM Properties 12-137
UDM Parameters 12-138
UDM Part Edits 12-138
Library of Models for CAD Integration 12-139
3D Modeler Discovery Link 12-141
Importing Discovery Geometry into Ansys Electronics Desktop 12-141
Connecting to an Active Discovery Session 12-143
Property Window of Discovery Link UDM 12-144

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Archive a Project Containing Discovery Link 12-147
Break Connection with Discovery Link 12-147
SpaceClaim Integration 12-147
3D Component Library 12-147
Creating a 3D Component from an Existing Model 12-149
Create 3D Component: Model Tab 12-152
Create 3D Component Boundaries Tab 12-153
Create 3D Component Excitations Tab 12-154
Create 3D Component Mesh Tab 12-155
Create 3D Component Coordinate Systems Tab 12-156
Create 3D Component Parameters Tab 12-158
Non-Geometry Variables in 3D Components 12-159
Dependent Variables in 3D Components 12-159
Create 3D Component Encryption Tab 12-160
Set Passwords 12-163
Hiding Component Content 12-164
Inserting Encrypted Components 12-165
Editing Encrypted Components 12-165
Opening a 3D Component File 12-166
Copying and Pasting Encrypted Components 12-166
Create 3D Component Licensing Tab 12-167
Create 3D Component Image Tab 12-167
Create 3D Component Info Tab 12-170
Save 3D Component File 12-172
Using the Component Libraries Window 12-173
Component file organization on the disk 12-177
Replace Selection by 3D Component 12-177
Inserting a 3D Component into a Design 12-181

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Making Component Instances Independent 12-187
Project Manager 12-188
History Tree 12-188
Viewing Properties of 3D Components 12-189
3D Components in the Project Manager Window 12-192
3D Components in the History Tree 12-194
Editing 3D Component Properties 12-195
Editing a 3D Component Definition 12-199
Verifying Updates to Components 12-202
Updating Definitions 12-202
Updating a 3D Component Definition 12-203
Layout Components [Beta] – Topic Index by Product 12-204
Modifying Objects 12-205
Object and History Editing 12-205
Modify Object Appearance 12-206
Modify Object Dimensions 12-206
Modify Object Location or Orientation 12-206
Surface or Edge Operations for Objects 12-206
Boolean Operations on Objects 12-207
Assigning Color to Objects 12-207
Setting the Default Color of Objects 12-208
Assigning Transparency to an Object 12-209
Setting the Default Transparency of Objects 12-209
Copying and Pasting Objects 12-209
Import a Model from the Clipboard 12-210
Copy Image 12-211
Deleting Objects 12-212
Deleting Start Points and Endpoints 12-212

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete Last Operation 12-213
Cutting Objects 12-214
Moving Objects 12-214
Rotating Objects 12-215
Changing the Orientation of an Object 12-217
Mirroring Objects 12-218
Offsetting Objects 12-220
Duplicating Objects 12-220
Duplicating Objects Along a Line 12-222
Duplicating Objects Around an Axis 12-223
Duplicating and Mirroring Objects 12-224
Scaling Objects 12-225
Sweeping Objects 12-227
Sweeping Around an Axis 12-227
Sweeping Along a Vector 12-229
Sweeping Along a Path 12-231
Best Practices 12-231
Sweeping Faces Along Normal 12-233
Draft Type Options 12-233
Thicken Sheet 12-236
Wrap Sheet Command 12-237
Covering Lines 12-242
Covering Faces 12-242
Uncovering Faces 12-243
Detaching Faces 12-243
Detaching Edges 12-243
Creating a Cross-Section 12-244
Connecting Objects 12-244

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project Sheet Object 12-245
Moving Faces or Edges 12-249
Moving Faces Along the Normal 12-249
Moving Faces Along a Vector 12-250
Moving Edges Along Normal 12-252
Uniting Objects 12-253
Subtracting Objects 12-253
Creating Objects from Intersections 12-255
Creating an Object from a Face 12-256
Creating an Object from an Edge 12-257
Splitting Objects 12-258
Separating Bodies 12-262
Converting Polyline Segments 12-262
Rounding the Edge of Objects (Fillet Command) 12-263
Flattening the Edge of Objects (Chamfer Command) 12-265
Imprinting an Object 12-268
Imprint Projection Commands 12-271
Purging Object History 12-274
Generate History to Reproduce Portions of Model 12-274
Selecting Items in the 3D Modeler Window 12-275
Selecting Objects 12-277
Selecting Several Objects 12-278
Selecting Objects by Name or Material 12-279
Selection by Object Name 12-280
Selection by Object Material 12-280
Select By Area 12-281
Setting the Default Color and Transparency of Selected Objects 12-283
Setting the Default Color of Highlighted Objects 12-283

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating an Object List 12-284
Reassigning Objects to Another Object List 12-285
Using or Viewing List Objects 12-285
Selecting Faces 12-286
Selecting All Faces of an Object 12-287
Selecting Faces by Name 12-288
Selecting Faces by Plane 12-289
Creating a Face List 12-290
Selecting Edges 12-291
Selecting All Edges of an Object or Face 12-292
Selecting Vertices 12-293
Selecting Multi 12-295
Controlling the Selection in Multi Mode 12-296
Selecting Groups and Submodels 12-297
Clearing a Selection 12-297
Selecting the Face, Edge, Vertex, or Object Behind 12-298
Selecting Objects and Surfaces That Lie Inside Other Objects 12-298
Selecting Objects by Variable 12-301
Assigning Coordinates to New Objects 12-301
Setting the Reference Point 12-302
Defining Cartesian Coordinates 12-302
Defining Cylindrical Coordinates 12-303
Defining Spherical Coordinates 12-303
Using Absolute Coordinates 12-304
Using Relative Coordinates 12-305
Choosing the Movement Mode 12-305
Moving the Cursor In Plane 12-306
Moving the Cursor Out of Plane 12-308

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving the Cursor in 3D Space ... 12-309
Moving the Cursor Along the X-Axis ... 12-310
Moving the Cursor Along the Y-Axis ... 12-311
Moving the Cursor Along the Z-Axis ... 12-312
Measure Modes ... 12-312
Setting Coordinate Systems ... 12-316
Setting the Working Coordinate System ... 12-318
Creating a Relative Coordinate System ... 12-318
Creating an Offset Relative CS ... 12-319
Creating a Rotated Relative CS ... 12-321
Creating an Offset and Rotated Relative CS ... 12-323
Creating a Face Coordinate System ... 12-325
Automatically Creating Face Coordinate Systems ... 12-328
Creating an Object Coordinate System ... 12-328
Creating an Offset Object CS ... 12-329
Creating a Rotated Object CS ... 12-330
Creating an Object CS that is Both Offset and Rotated ... 12-331
Move CS to End Command for History Tree ... 12-332
Modifying Coordinate Systems ... 12-333
Expressing Cartesian, Cylindrical, or Spherical Coordinates ... 12-334
Deleting Coordinate Systems ... 12-335
Setting the Drawing Plane ... 12-335
Cleaning Up Model History ... 12-336
13 - Modifying the 3D Model View ... 13-1
Change the View ... 13-1
Show or Hide Design Objects ... 13-2
Hiding Objects from View ... 13-2
Showing Objects ... 13-4

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Showing Only Selected Objects in All or Active Views 13-7
Showing and Hiding Boundaries and Excitations 13-9
Showing and Hiding Boundaries and Excitations in the Active View Window 13-9
Showing and Hiding Boundaries and Excitations in Every View Window 13-10
Change Object Visualization 13-11
Change the Background 13-11
View Options: 3D UI Options 13-11
Panning the View 13-16
Zooming in and Out 13-17
Zoom In and Zoom Out Commands 13-19
Zoom Previous Command 13-20
Zooming into a Rectangular Area 13-22
Zooming in or out Using a Mouse Wheel 13-23
Zoom to a Selected Boundary or Excitation 13-23
Modifying the View Orientation 13-24
Apply an Orientation to the Current View 13-26
Using the Orientation Gadget 13-28
Controlling Visibility of the Orientation Gadget 13-32
Rotating or Spinning the View 13-33
Changing the Model View with Alt+Double-Click Areas 13-40
Applying a Default View Orientation 13-41
Applying a New View Orientation 13-43
Removing an Orientation 13-44
Fitting Objects in the View Window 13-45
Fitting All Objects in a View Window 13-46
Fitting Selected Objects in a View Window 13-46
Active View Visibility Dialog Box 13-47
Rendering Objects as Wireframes or Solids 13-49

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Default View Rendering Mode ... 13-49
Setting the Enhanced Display of Material Color and Transparency ... 13-50
Setting the Curved Surface Visualization ... 13-52
Using Clip Planes ... 13-53
Modifying the Lighting ... 13-57
Setting the Projection View ... 13-59
Setting the Background Color ... 13-60
Modifying the Coordinate System Axes View ... 13-61
Showing or Hiding the Axes ... 13-62
Show the Axes for Selected Objects ... 13-62
Enlarging or Shrinking the Axes ... 13-63
Showing or Hiding the Triad Axes ... 13-63
Choosing Grid Settings ... 13-63
Setting the Grid Type ... 13-64
Setting the Grid Style ... 13-65
Setting the Grid Density and Spacing ... 13-66
Setting the Grid's Visibility ... 13-68
Setting the Grid Plane ... 13-68
14 - Specifying Solution Settings for Mechanical ... 14-1
Editing an Existing Solution Setup: ... 14-1
Setup Link Window ... 14-1
Clearing Linked Data ... 14-1
15 - Optimetrics ... 15-1
Parametric Overview ... 15-2
Setting Up a Parametric Analysis ... 15-2
Adding a Parametric Sweep from a File ... 15-4
Adding a Variable Sweep Definition ... 15-5
Specifying Variable Values for a Sweep Definition ... 15-5

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Synchronizing Variable Sweep Definitions 15-6
Creating Parametric Animations 15-7
Modifying a Variable Sweep Definition Manually 15-11
Overriding a Variable's Current Value in a Parametric Setup 15-11
Specifying a Solution Setup for a Parametric Setup 15-12
Specifying the Solution Quantity to Evaluate for Parametric Analysis 15-12
Setting up Calculations for Optimetrics 15-13
Specifying a Solution Quantity's Calculation Range 15-14
Using Distributed Analysis 15-15
Solving a Parametric Analysis on Another Machine 15-16
Viewing Results for Parametric Solution Quantities 15-17
Optimization Overview 15-17
Choosing an Optimizer 15-17
Legacy Optimizers 15-19
Adaptive Single Objective Optimization 15-20
ASO Workflow 15-20
AMO Steps 15-22
Setting Up Adaptive Single-Objective (Gradient) Optimizer 15-23
Adaptive Multiple Objective Optimization 15-29
AMO Workflow 15-29
AMO Steps 15-30
Setting Up Adaptive Multiple-Objective (Random Search) Optimizer 15-32
Genetic Algorithm (Random Search) 15-37
Advanced Genetic Algorithm Optimizer Options 15-38
Optimization Setup for the Genetic Algorithm (Random search) Optimizer 15-41
MATLAB Optimizer 15-43
Optimization Setup for the MATLAB Optimizer 15-48
Mixed-Integer Sequential Quadratic Programming 15-50

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP 15-51
Setting Up Merit-based Sequential Quadratic Programming(Gradient) Optimizer 15-53
Setting Up Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer 15-55
Setting Up Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer 15-60
Multi-Objective Genetic Algorithm (MOGA) 15-65
MOGA Workflow 15-65
MOGA Steps 15-66
MOGA Steps to Generate a New Population 15-67
Convergence Criteria in MOGA-Based Multi-Objective Optimization 15-70
Maximum Allowable Pareto Percentage 15-70
Convergence Stability Percentage 15-70
Setting Up Multi-Objective Genetic Algorithm(Random Search) Optimizer 15-71
Pattern Search (Search-Based) 15-77
Optimization Setup for the Pattern Search (Search-based) Optimizer 15-79
Quasi Newton (Gradient) 15-81
Optimization Setup for the Quasi Newton (Gradient) Optimizer 15-84
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) 15-86
Optimization Setup for the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer 15-86
Sequential Nonlinear Programming (Gradient) 15-88
Optimization Setup for the Sequential Nonlinear Programming (Gradient) Optimizer 15-89
Optimization Variables and the Design Space 15-91
Setting Up an Optimization Analysis 15-91
Setting the Maximum Iterations for an Optimization Analysis 15-93
Modifying the Starting Variable Value for Optimization 15-94
Cost Function 15-95

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acceptable Cost 15-96
Cost Function Noise 15-96
Adding a Cost Function 15-97
Specifying a Solution Quantity for a Cost Function Goal 15-99
Setting the Calculation Range of a Cost Function Goal 15-100
Setting a Goal Value 15-100
Specifying a Single Goal Value 15-101
Specifying an Expression as a Goal Value 15-101
Specifying a Variable-Dependent Goal Value 15-102
Goal Weight 15-102
Changing the Cost Function Norm 15-103
Explanation of L1, L2, and Max Norms in Optimization 15-104
Example of a More Complex Cost Function 15-106
Setting the Min. and Max. Variable Values for Optimization 15-107
Edit Calculation Range 15-107
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 15-109
Changing the Min. and Max. Variable Values for Every Optimization Setup 15-110
Step Size 15-110
Setting the Min. and Max. Step Sizes 15-111
Setting the Min and Max Focus 15-112
Equalizing the Influence of Different Optimization Variables 15-112
Setting the Min and Max Focus Values 15-113
Solving a Parametric Setup Before an Optimization 15-113
Solving a Parametric Setup During an Optimization 15-114
Automatically Updating a Variable's Value after Optimization 15-114
Sensitivity Analysis Overview 15-114
Selecting a Primary Output 15-115
Setting Up a Sensitivity Analysis 15-115

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Maximum Iterations Per Variable 15-117
Setting Up an Output Parameter 15-117
Specifying a Solution Quantity for an Output Parameter 15-118
Setting the Calculation Range of an Output Parameter 15-119
Modifying the Starting Variable Value for Sensitivity Analysis 15-120
Setting the Min. and Max. Variable Values 15-121
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 15-121
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 15-121
Setting the Initial Displacement 15-122
Solving a Parametric Setup before a Sensitivity Analysis 15-122
Solving a Parametric Setup during a Sensitivity Analysis 15-122
Performing Worst Case Analysis 15-123
Statistical Analysis Overview 15-127
Setting Up a Statistical Analysis 15-127
Setting the Maximum Iterations for a Statistical Analysis 15-129
Specifying the Solution Quantity to Evaluate for Statistical Analysis 15-129
Setting the Solution Quantity's Calculation Range 15-130
Setting the Distribution Criteria 15-131
Overriding the Distribution Criteria for a Single Statistical Setup 15-131
Changing the Distribution Criteria for Every Statistical Setup 15-132
Statistical Cutoff Probability 15-133
Edit Distribution 15-134
Modifying the Starting Variable Value for Statistical Analysis 15-136
Solving a Parametric Setup During a Statistical Analysis 15-137
Using Design of Experiments 15-137
Setting Up Design of Experiments in Ansys Electronics Desktop 15-140
Add the Design of Experiments Setup 15-140
Design of Experiments Tab 15-143

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimal Space Filling Design (OSF) 15-146
Central Composite Design (CCD) 15-147
Box Behnken Design (CCD) 15-148
Custom DOE Type 15-149
Latin Hypercube Sampling 15-149
Table Tab for Design of Experiments 15-150
Response Surface Tab for Design of Experiments 15-152
Variables Tab for Design of Experiments 15-153
Viewing Analysis Result for Design of Experiments 15-156
Min-Max Search View for Design of Experiments Result 15-159
Refinement Points Table 15-159
Performing a Manual Refinement 15-160
Response Surface Results 15-160
Response Points Table 15-163
Response Curve 15-163
Response Curve (2D Slices) 15-166
Verification Points Table 15-168
Goodness of Fit (Predicted vs Observed Chart) 15-169
Local Sensitivity Charts 15-170
Local Sensitivity Curves 15-171
Running an Optimetrics Analysis 15-173
Viewing Analysis Results for Optimetrics Solutions 15-174
Viewing Solution Data for an Optimetrics Design Variation 15-174
Viewing an Optimetrics Solution's Profile Data 15-175
Viewing Results for Parametric Solution Quantities 15-175
Plotting Solution Quantity Results vs. a Swept Variable 15-176
Viewing Cost Results for an Optimization Analysis 15-177
Plotting Cost Results for an Optimization Analysis 15-177

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Output Parameter Results for a Sensitivity Analysis 15-177
Plotting Output Parameter Results for a Sensitivity Analysis 15-178
Viewing Distribution Results for a Statistical Analysis 15-178
Plotting Distribution Results for a Statistical Analysis 15-179
Saving Field Solutions for Optimetrics Analyses 15-179
Saving Field Solutions for a Design of Experiments Setup 15-180
Saving Field Solutions for a Parametric Setup 15-181
Saving Field Solutions for an Optimization Setup 15-182
Saving Field Solutions for a Sensitivity Setup 15-183
Saving Field Solutions for a Statistical Setup 15-183
Adding an Expression in the Output Variables Window 15-184
Copying Meshes in Optimetrics Sweeps 15-185
Excluding a Variable from an Optimetrics Analysis 15-185
Linear Constraints 15-186
Setting a Linear Constraint 15-186
Modifying a Linear Constraint 15-187
Deleting a Linear Constraint 15-188
Link to DesignXplorer 15-188
Modifying the Value of a Fixed Variable 15-190
Tuning a Variable 15-191
Tuning Overview 15-194
Applying a Tuned State to a Design 15-194
Saving a Tuned State 15-194
Reverting to a Saved Tuned State 15-195
Resetting Variable Values after Tuning 15-195
16 - High Performance Computing 16-1
Remote Analysis 16-1
Prerequisites for Remote and Distributed Analysis 16-2

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configuring the Local Machine to Solve Remotely 16-3
Remote Analysis Options 16-3
Running Remote Analysis 16-4
Setting up Linux for Mechanical 16-4
Troubleshooting 16-6
Remote Solve Node = Windows 16-8
Remote Solve Node = Linux 16-9
Distributed Analysis 16-10
Beta Feature: Parallel Component Mesh Adapt for 3D Component Array 16-11
Configure a Distributed Analysis 16-11
Editing Distributed Machine Configurations for Mechanical 16-11
Test Machines 16-15
Job Distribution Tab 16-16
Adding Configurations or Accepting Edits 16-17
Selecting Optimal Configurations for Distributed Analysis 16-18
Large Scale DSO for Parametric Analysis 16-18
Prerequisites for Large Scale DSO 16-20
Job Management Interface for Large Scale DSO 16-21
Large Scale DSO Command Line Syntax 16-27
Large Scale DSO Job Outputs 16-31
Output Location and Organization 16-31
CSV File Contents 16-31
Postprocessing Large Scale DSO Dataset Solutions 16-32
Importing a Large Scale DSO Solution 16-32
Viewing Imported Solutions 16-33
Creating a Dataset Report 16-34
Cloning a Dataset Solution 16-34
Large Scale DSO Job Monitoring 16-34

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Resources for Large Scale DSO Monitoring 16-35
Large Scale DSO Deployment/Configuration 16-35
Large Scale DSO Tutorial Example 16-37
Prepare the Model for Large Scale DSO Analysis 16-37
Submit the Large Scale DSO Job: Examples 16-42
Large Scale DSO Example: Post Process the Results 16-44
Large Scale DSO Known Issues/Troubleshooting 16-46
Node Order 16-46
Parallel Task Limitation for LS-DSO Parametric Variations 16-46
Job Restart 16-47
Linux-Only Issues 16-47
Interactive Scheduler Jobs 16-48
Specifying Options for Interactive Scheduler Jobs 16-48
Batchoptions for Interactive Scheduler Jobs 16-49
DSO Configuration for Interactive Scheduler Jobs 16-50
Analysis Configuration - Machines Tab 16-51
Analysis Configuration - Job Distribution Tab 16-52
Analysis Configuration - Options Tab 16-53
Design Type Options for Interactive Scheduler Jobs 16-54
High Performance Computing (HPC) Integration 16-55
Scheduler Terminology 16-55
What a Scheduler Does 16-56
Configuring Electronics Installation for HPC 16-57
Firewall Configuration 16-58
Installation Directory Examples 16-58
Integration with Microsoft Windows® HPC Scheduler 16-58
Submitting and Monitoring Ansys EM HPC Jobs 16-59
Submitting and Monitoring Jobs for Windows HPC 16-61

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Windows® HPC Job Templates 16-78
Selecting Computation Resource Units (Job Unit Type) 16-78
Windows® HPC Non-Exclusive Jobs 16-79
Windows® HPC Job Credentials 16-91
Integration with Platform's Load Sharing Facility (LSF) 16-92
LSF Job Management 16-92
LSF Job Submission Guidelines 16-93
Integration of Ansys EM Products with LSF 16-93
Installing Ansys EM Tools on LSF Cluster 16-94
Using the bsub Command to Submit Batch Jobs 16-96
bsub Arguments 16-97
Example LSF bsub Command Lines (Linux Only) 16-98
Serial Job 16-98
Serial Job Requiring a Minimum of 4GB 16-98
Multi-processing Job using 4 Cores 16-98
Distributed Processing Job using 4 Engines 16-98
Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing 16-99
Shell Script (~/projects/OptimTee.csh): 16-99
Monitoring LSF Batch Jobs 16-99
Terminating LSF Batch Jobs 16-100
LSF Known Issues and Workarounds 16-101
LSF Troubleshooting 16-101
Integration with PBS (Portable Batch System) 16-102
PBS Job Management 16-102
Non Standard Installations for PBS 16-103
PBS Limitations 16-103
Submitting Ansys EM PBS Batch Jobs 16-105

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitoring PBS Batch Jobs 16-106
qsub Arguments 16-107
Example PBS qsub Command Lines 16-108
Integrating Ansys EM Tools with Third-Party Schedulers 16-110
Introduction 16-110
Serial Jobs 16-111
Parallel Jobs 16-111
Common Requirements for Running Jobs 16-111
Using a Shared Library (Linux) or a DLL (Microsoft Windows) 16-112
Build Information for Scheduler Proxy Library 16-113
Implementation Details for Custom Scheduler Integration 16-113
IsProductLaunchedInYourEnvironment 16-114
GetTempDirectory 16-114
GetMachineListAvailableForDistribution 16-115
GetMessageStringToRegisterForSigTerm 16-116
LaunchProcess 16-117
GetUseRsmForEngineLaunch 16-118
GetThisJobID 16-119
GetSchedulerDisplayName 16-120
Using the Command Line to Submit HPC Jobs 16-121
Distributed Jobs 16-121
Serial Job on a Single Processor 16-121
Distributed Analysis and Multi-Processing in the Same Job 16-122
Scheduler Proxy Interfaces 16-122
Testing Scheduler Integration 16-128
Testing IsProductLaunchedInYourEnvironment 16-128
Testing GetSchedulerDisplayName and GetThisJobID 16-128
Testing GetTempDirectory 16-129

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Testing GetMachineListAvailableForDistribution 16-129
Testing LaunchProcess 16-129
Testing GetUseRsmForEngineLaunch 16-130
Troubleshooting Custom Scheduler Integration 16-130
None of the Proxy Functions are Called 16-130
Troubleshooting IsProductLaunchedInYourEnvironment Function 16-131
Troubleshooting GetSchedulerDisplayName 16-131
Troubleshooting GetThisJobID 16-131
Troubleshooting GetTempDirectory 16-132
Troubleshooting GetMachineListAvailableForDistribution 16-132
Troubleshooting LaunchProcess 16-132
Troubleshooting GetUseRsmForEngineLaunch 16-133
Using an IronPython Program for Integration with a Scheduler 16-133
GetName [IronPython] 16-135
GetDescription [IronPython] 16-135
IsProductLaunchedInYourEnvironment [IronPython] 16-136
GetSchedulerDisplayName [IronPython] 16-136
GetThisJobID [IronPython] 16-137
GetUseRsmForEngineLaunch [IronPython] 16-137
GetTempDirectory [IronPython] 16-138
GetMessageStringToRegisterForSigTerm [IronPython] 16-139
GetMachineListAvailableForDistribution [IronPython] 16-139
LaunchProcess [IronPython] 16-140
Ansys Cloud Direct Support for HPC Job Management 16-141
Prerequisites for Using Ansys Cloud Direct with Ansys Electronics Desktop 16-142
Job Management UI for Ansys Cloud Direct 16-142
Environment Variable Settings for Ansys Cloud Direct 16-153
Troubleshooting and Debugging Environment Variables 16-153

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option Settings for Ansys Cloud Direct 16-154
Automatic Batchoption Settings 16-154
Selecting a Scheduler 16-154
Submitting a Job 16-156
Using Advanced Job Submission Options 16-161
Batchoptions 16-162
Environment Variables 16-163
Batch Extract 16-164
Customize Job Submission 16-165
Using the Command Line to Submit HPC Jobs 16-165
Distributed Jobs 16-165
Serial Job on a Single Processor 16-165
Distributed Analysis and Multi-Processing in the Same Job 16-166
Job Import and Export 16-166
Scripting 16-167
Multi-Step Job Submission 16-167
Windows to Linux Job Submission 16-172
Prerequisites for Job Submission 16-173
Prerequisites for Job Monitoring 16-174
Supported Schedulers 16-174
Job Submission Scripting 16-180
Monitoring Jobs 16-181
Monitor Job Window 16-181
Monitoring Ansys Cloud Direct Jobs 16-184
Monitoring Ansys Cloud Burst Jobs 16-186
Monitoring Large Scale DSO Jobs 16-187
Web Client for Batch Solve Monitoring and Reporting 16-187
Aborting an Analysis 16-193

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distribution Command Line Options 16-194
Integration with Ansys Remote Simulation Management (RSM) 16-195
Changing the AnsoftRSMService Listening Port 16-209
AnsoftRSMService Configuration 16-210
Ansoft Electromagnetics Desktop Configuration 16-211
Running HPC Diagnostics 16-213
Site-Specific Diagnostics Job 16-216
Changing Solution Priority for System Resources 16-218
desktopproxy Utility to Abort or Cleanstop a BatchSolve 16-219
17 - Running Simulations 17-1
Solving a Single Setup 17-2
Running More than One Simulation 17-3
Monitoring Queued Simulations 17-4
Disabling and Enabling an Analysis Setup 17-4
Monitoring the Solution Process 17-5
Changing a Solution Priority for System Resources 17-6
Aborting an Analysis 17-7
Re-solving after Modifying a Design 17-8
18 - Postprocessing and Generating Reports 18-1
Viewing Solution Data 18-2
Viewing a Solution Profile 18-2
Viewing Mesh Statistics 18-5
Deleting Solution Data 18-6
Creating Reports 18-7
Creating a New Report 18-9
Context Section for Reports 18-14
Using Families Tab for Reports 18-14
Modifying Reports 18-17

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Report: Selecting Use All Values or Making Selection 18-21
Modify Report: Using the Edit Sweep Dialog Box 18-22
Creating a Report from an Ansoft Report Data File 18-22
Zooming and Fitting Reports 18-23
Modifying the Background Properties of a Report 18-23
Modifying the Legend in a Report 18-26
Exporting Ansoft Report Data Format Files 18-28
Exporting Reports as Graphics 18-30
Creating Custom Report Templates and Defaults 18-31
Selecting the Report Type 18-32
Selecting the Display Type 18-34
Creating 2D Rectangular Plots 18-36
Creating 2D Rectangular Stacked Plots 18-37
Creating 3D Rectangular Plots 18-40
Creating Rectangular Contour Plots 18-46
Creating Data Tables 18-49
Delta Markers in 2D Reports 18-51
Working with Traces 18-51
Editing Trace Properties 18-53
Editing the Display Properties of Traces 18-54
Adding Data Markers to Traces 18-57
Y Markers in Stacked XY Plots 18-59
Discarding Report Values Below a Specified Threshold 18-66
Adding Characteristics to a Trace 18-67
Adding a Recently Used Trace Characteristic 18-67
Adding a Trace Characteristic from Favorites 18-67
Adding Trace Characteristics to your Favorites 18-68
Adding Characteristics Using the Add Trace Characteristics Window 18-70

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy and Paste of Report and Trace Definitions ... 18-72
Copy and Paste of Report and Trace Data ... 18-73
Removing Traces ... 18-73
Removing All Trace Characteristics ... 18-74
Limit Lines in Cartesian Plots ... 18-74
Sweeping Values Across a Distance ... 18-81
Selecting a Function for a Plot ... 18-82
Selecting Solution Quantities to Plot ... 18-88
Selecting a Field Quantity to Plot ... 18-93
Plotting Imported Solution Data ... 18-93
Setting a Range Function ... 18-93
Range Functions ... 18-94
Importing Plot Data ... 18-99
Exporting Reports ... 18-101
Deleting Reports ... 18-103
Specifying Output Variables ... 18-103
Function List for Output Variables ... 18-106
Plotting Field Overlays in Mechanical ... 18-106
Modifying Field Plots ... 18-107
Modifying Markers on Point Plots ... 18-111
Setting a Plot's Visibility ... 18-111
Color Key Settings ... 18-114
Setting Field Overlay Attributes ... 18-115
Modifying the Field Plot Scale ... 18-124
Modifying Field Plot Colors ... 18-127
Modifying Vector Field Plot Arrows ... 18-132
Modifying Scalar Field Plot Isovalues ... 18-135
Setting the Mesh Visibility on Field Plots ... 18-139

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Field Overlay Plot Defaults 18-139
Working with Scalar Field Plot Markers 18-140
Adding a Field Plot Marker 18-140
Editing Field Plot Markers 18-142
Deleting a Field Plot Marker 18-143
Exporting a Field Plot Marker Table 18-143
Clearing All Field Plot Markers 18-143
Mapping Scalar Field Plot Transparency to Field Values 18-144
Plotting the Mesh 18-144
Setting Mesh Plot Attributes 18-148
Creating Animations 18-151
Creating Geometry Animations 18-153
Creating Parametric Animations 18-156
Creating Mesh Animations 18-160
Controlling the Animation's Display 18-163
Modifying an Animation Setup 18-165
Exporting Animations 18-167
Exporting Ensight from the Select Animation Dialog 18-167
Exporting from the Animation Control Dialog 18-169
Using the Fields Calculator 18-171
Scripting Support 18-174
Preliminary Considerations for Using the Fields Calculator 18-174
Opening the Fields Calculator 18-174
Context Area 18-175
The Calculator Stack 18-176
Stack Commands 18-177
Pop 18-178
RlUp 18-178

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RlDn 18-178
Exch 18-178
Clear 18-178
Undo 18-178
Registers 18-178
Enlarging the Register Display Area 18-181
Units of Measure 18-181
Input Commands 18-181
Quantity Command 18-183
Geometry Command 18-184
Constant Command 18-188
Number Command 18-189
Function Command 18-190
Geom Settings Command 18-191
Read Command 18-192
General Commands 18-192
Scalar Commands 18-196
Vec? Command 18-197
1/x (Inverse) Command 18-197
Pow Command 18-197
(Square Root) Command 18-198
d/d? (Partial Derivative) Command 18-198
(Integral) Command 18-198
Min Command 18-199
Max Command 18-199
Ñ (Gradient) Command 18-200
Ln Command 18-200
Log Command 18-200

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Std Command 18-200
Vector Commands 18-200
Scal? Command 18-202
Matl Command 18-202
Mag Command 18-203
Dot Command 18-203
Cross Command 18-203
Divg Command 18-203
Curl Command 18-203
Tangent Command 18-203
Normal Command 18-204
Unit Vec Command 18-205
X Form 18-205
Output Commands 18-206
Value Command 18-207
Eval Command 18-209
Write Command 18-210
Export Command 18-211
Creating 2D Reports from Named Expressions 18-216
Calculating Derived Field Quantities Using Named Expressions 18-217
Named Expression Library 18-221
Exiting the Fields Calculator 18-223
User Defined Outputs (UDOs) 18-223
Named Probes and Properties in User Defined Outputs 18-223
Computation of Traces Based UDO Calculations 18-224
Dimensions Reduction by UDO Calculations 18-225
Dynamic Probes 18-225
User Defined Outputs: Python Script API 18-225

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDO Extension Implementation 18-226
Import Statements 18-226
UDOExtension Class 18-226
Validate 18-226
IUDOPluginExtension Abstract Class 18-227
GetUDSName 18-228
GetUDSDescription 18-228
GetUDSSweepNames 18-229
GetCategoryNames 18-229
GetQuantityNames 18-230
GetQuantityInfo 18-230
GetInputUDSParams 18-231
GetDynamicProbes 18-234
Compute 18-234
Data Types Used in UDO Python Scripts 18-237
Constants 18-237
Abstract Classes 18-237
IUDSInputData 18-237
GetDoubleProbeData 18-238
GetSweepsDataForProbe 18-238
GetComplexProbeData 18-239
GetSweepNamesForProbe 18-240
GetRequiredQuantities 18-240
GetVariableValues 18-241
GetInterpolationOrdersData 18-241
IUDSOutputData 18-242
SetSweepsData 18-242
SetDoubleQuantityData 18-243

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SetComplexQuantityData 18-243
Working With Properties for UDO 18-244
IPropertyList Abstract Class 18-244
IProperty Abstract Class 18-245
INumberProperty Abstract Class 18-245
ITextProperty Abstract Class 18-245
IMenuProperty Abstract Class 18-246
Other Application-Specific Classes Used in Python Scripts 18-247
Constants Class 18-247
UDSProbeParams Class 18-247
UDSDynamicProbes Class 18-248
QuantityInfo Class 18-248
IProgressMonitor Abstract Class 18-249
SetTaskName 18-250
SetSubTaskName 18-250
BeginTask 18-251
SetTaskProgressPercentage 18-251
CheckForAbort 18-251
EndTask 18-252
Using .NET Collection Classes and Interfaces in Python Scripts 18-252
User Defined Outputs: Messaging Methods 18-254
AddErrorMessage 18-259
AddInfoMessage 18-260
AddWarningMessage 18-260
User Defined Outputs: Script Organization 18-260
Using Script Libraries 18-261
Using Additional .NET Assemblies 18-261
User-Defined Documents (UDDs) 18-262

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create User-Defined Document Inputs 18-263
UDD Document Creation and Display 18-265
Managing Documents Listed in the Project Window Under Results 18-266
Documents Folder's Context Menu 18-267
Document Folder's Property Window 18-267
Viewing UDDs with an HTML Web Browser 18-268
UDD Script Libraries 18-269
User-Defined Documents: Python Script API 18-270
Import Statements 18-270
Data Types Used in UDD Python Scripts 18-271
Constants 18-271
Abstract Classes 18-271
IProgressMonitor Abstract Class 18-271
SetTaskName 18-272
SetSubTaskName 18-272
BeginTask 18-273
SetTaskProgressPercentage 18-273
CheckForAbort 18-274
EndTask 18-274
IUDDPluginExtension Abstract Class 18-275
GetUDDName 18-275
GetUDDDescription 18-276
ShowDefaultSetupDialog 18-276
GetUDDInputParams 18-276
Generate 18-278
SetupUDDInputParams 18-280
HandleUDDEvents 18-281
GetUDDSchema 18-282

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetUDDStyleSheetForHtml 18-282
GetUDDStyleSheetForPdf 18-283
GetFopExecutable 18-283
GetUDDAppContext 18-284
GetUDDDesignContext 18-284
UDDExtension Class 18-285
UDDInputData Class 18-285
UDDInputBool 18-286
UDDInputDouble 18-286
UDDInputSolution 18-286
UDDInputText 18-287
UDDInputTrace 18-287
UDDInputParams Class 18-288
UDD Input Interfaces 18-288
Explication of a Sample UDD Script 18-290
Document Generator Interfaces 18-292
Chip Model Analyzer (CMA) 18-297
PinToPinUtility 18-297
19 - Meshing 19-1
Windows® HPC Non-Exclusive Jobs 19-1
Mesh Linking 19-12
How to Import a Mesh: 19-15
Verifying Mesh Linking 19-23
Mesh Linking Requirements 19-24
Cached Link Data and Solution Invalidation 19-26
Specifying Initial Mesh Settings 19-27
Mechanical Mesh Slider 19-32
TAU Flex Meshing 19-34

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Using TAU Flex 19-35
TAU Flex Warning Messages 19-39
Reviewing TAU Flex Mesh Warnings 19-39
Addressing TAU Flex Mesh Issues 19-41
Defining Mesh Operations 19-41
Length Based Mesh Refinement 19-43
Assigning Length-Based Mesh Refinement on Object Faces 19-45
Assigning Length-Based Mesh Refinement Inside Objects 19-46
Assigning Skin Depth-Based Mesh Refinement on Object Faces 19-47
Visualization for Skin Depth Refinement Mesh Results 19-52
Modifying Surface Approximation Settings 19-54
Applying Curvilinear Elements 19-58
Assigning Cylindrical Gap Treatment 19-60
3D Clone Mesh Generation 19-61
Band Mapping Angle 19-68
Clone Mesh Density 19-69
Rotational Layer Mesh Generation 19-72
Specifying Surface Priority for TAU Mesher 19-74
Specifying the Model Resolution 19-75
Specifying a Source for the Initial Mesh 19-77
Viewing List of Mesh Operations 19-79
Reassigning a Mesh Operation 19-80
Reverting to the Initial Mesh 19-81
Deleting Previously Assigned Mesh Refinements 19-82
Plotting the Mesh 19-83
Setting Mesh Plot Attributes 19-87
Generating Mesh without Solving 19-90
Viewing Mesh Properties 19-91

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Mesh Statistics 19-91
20 - Toolkits **20-1**
Modal Toolkit Example 20-3
Thermal Toolkit Example 20-6
21 - Ansys Workbench Integration Overview **21-1**
Integrating Ansys EM Suite with Ansys Workbench 21-1
Workbench Data Integration Overview 21-3
Electronics Desktop Mechanical Systems in Workbench 21-5
Adding New Analysis Systems 21-6
Importing Existing Ansys EM Projects into Workbench 21-7
Editing Ansys EM Models in Workbench 21-8
Analyzing Ansys EM Models in Workbench 21-9
Performing Parameter Studies in Workbench 21-11
Scripting in Workbench 21-15
Ansys EM CAD Integration through Workbench 21-16
CAD Integration Functionality 21-18
CAD Integration and Geometry Sharing 21-20
Bi-Directional CAD Integration 21-23
CAD Integration Model Edits 21-24
Multiple Geometry Links for CAD Integration 21-25
Healing with CAD Integration 21-26
Important Geometry Options for CAD Integration 21-27
Ansys EM to Ansys Geometry Transfer 21-28
Material Assignment Transfer 21-30
Geometry Transfer through Ansys DesignModeler (DM) 21-30
Workbench Material Data Transfer 21-31
Detailed Behavior 21-31
22 - Minerva Remote Storage Environment **22-1**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23 - Scripting **23-1**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Welcome to Mechanical Help

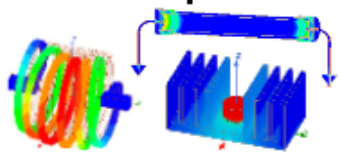
The Mechanical design type in Ansys Electronics Desktop provides system simulation for predicting the following:

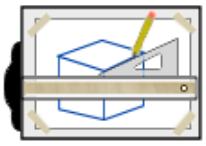
- ***Modal* solutions:** Natural frequencies of vibration, mode shapes, and related results (such as mass participation factors)
- ***Thermal* solutions:** Temperature and heat flux results
  - **Steady-State Thermal**: Determines the equilibrium state of the design
  - **Transient Thermal**: Determines solutions for multiple steps showing how the results vary with time
- ***Structural* solutions:** Thermal stress, strain, and deformation results

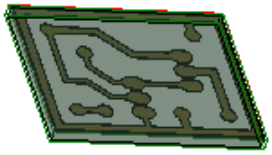
Click on the areas of interest below to quickly locate the desired help topics:

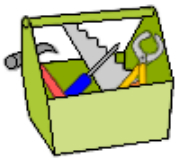
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Help:

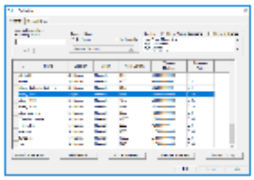

Getting Started Guides

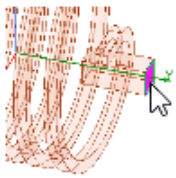

Drawing a Model


Layout Components

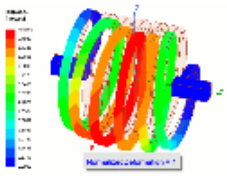

Toolkits

Modal Solutions Help:

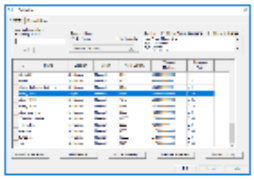

Assigning Materials

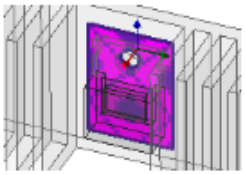

Boundaries

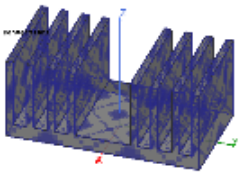
Solution Setup

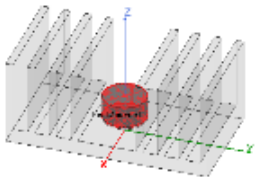

Results

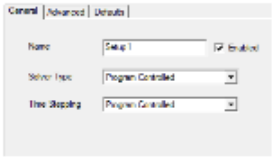
Thermal Solutions Help:

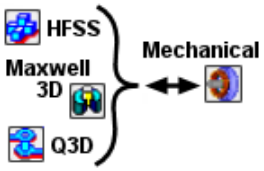

Assigning Materials


Thermal Contact


Boundaries

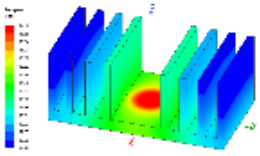

Excitations


Solution Setup

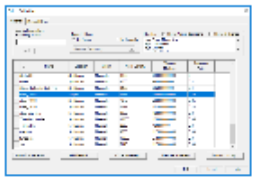


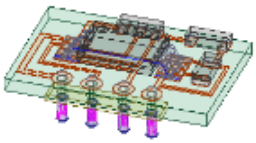
2-Way Coupling

Transient Solutions

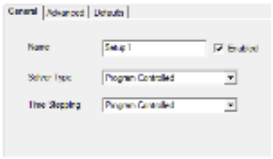

Results

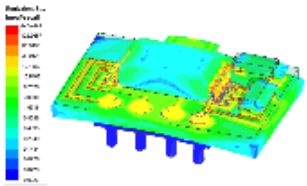
Structural Solutions Help:


Assigning Materials


Boundaries

Excitations


Solution Setup


Results

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Introduction to Mechanical

Ansys Electronics Desktop's (EDT's) *Mechanical* design type is a simulation tool for determining natural vibration frequencies (*Modal* solution), temperatures and heat flux (steady-state or transient *Thermal* solutions), and displacement and equivalent stress (*Structural* solutions). These mechanical analysis capabilities enable you to perform the following functions directly within the EDT application:

- Identify natural frequencies that might lead to structural resonance at electromagnetic excitation frequencies. In this way, you can avoid problematic excitation frequencies or modify the physical structure to avoid resonance problems. The mode shape and mass participation factors are also provided for each solved natural frequency.
- Determine steady-state or transient operating temperatures and heat flux distributions for given heat generation, applied temperatures, convection, and ambient environment temperatures. This tool can help you avoid excessive and damaging temperatures as well as over-designing heat dissipation measures.
- Import electromagnetic (EM) losses from certain other design types into your Mechanical – Thermal designs as surface or volume based heat sources.
- Estimate the heat transfer behavior between the rotor and stator of rotating electric machines.
- Perform two-way coupled analyses between Mechanical–Thermal designs and certain other design types.
- Perform trace mapping based on a source HFSS 3D Layout design of a printed circuit board. The traces, pads, vias and conductor pour areas are mapped to the solution as a metal fraction map. This method provides accurate, local thermal conductivities without actually having to include the conductors as solid objects in the Mechanical–Thermal solution.
- Determine thermal stresses and displacement based on manually defined temperatures or temperature results imported from another design.

These mechanical features are a subset of the simulation capabilities available within the *Ansys Mechanical* structural simulation product. They are included in AEDT as a convenience and an additional value for customers performing electromagnetic simulations.

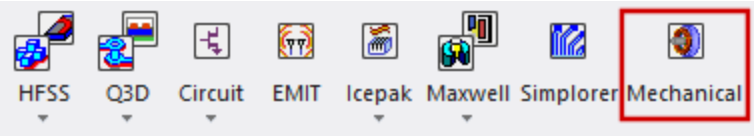
High Performance Computing (HPC) Details: You can specify the number of cores on a single machine that the solver uses for shared-memory-processing. Solver distribution is not supported for Modal, Thermal, or Structural solutions in the current release. However, you can distribute the solutions of Mechanical *Optimetrics Variations* to multiple machines.

Inserting a Mechanical Design

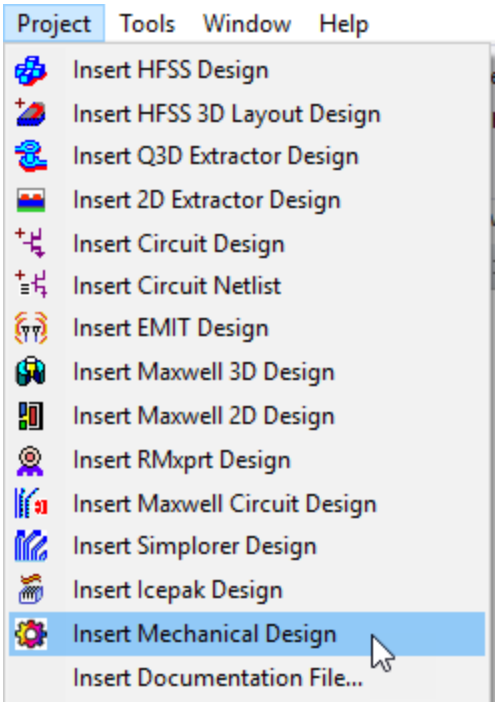
To insert a Mechanical design into your Ansys Electronics Desktop project, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

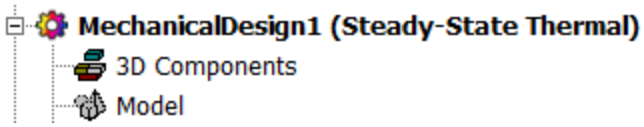
- On the **Desktop** tab of the ribbon, click **Mechanical**.



- Using the menu bar, click **Project** > **Insert Mechanical Design**.



The new design is listed in the Project Manager. It is named "MechanicalDesign*x*" by default, where *x* is a number incremented for each *Mechanical* design you add to the project. The default solution type, *(Steady-State Thermal)*, is appended to the name:



Optionally, you can choose to make any other solution type the default selection.

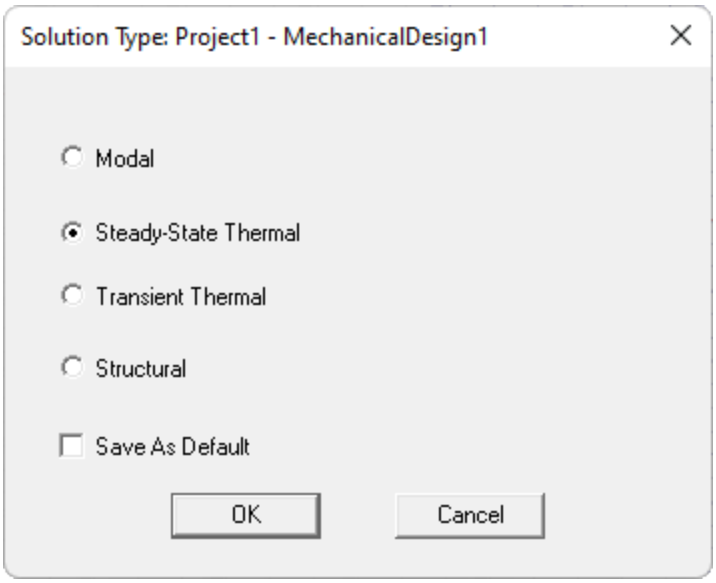
> **Note:**
>
> Click the plus sign to the left of the *Mechanical* design icon in the Project Manager to expand the branch and view specific data about the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choose Mechanical Solution Type

After inserting a Mechanical design into your project, choose the desired *Solution Type* as follows:

1. Access the *Solution Type* dialog box using *one* of the following two methods:
   - Using the menu bar, click **Mechanical** > **Solution Type**.
   - Right-click the mechanical design in the Project Manager (named **MechanicalDesign*x*** by default) and choose **Solution Type** from the shortcut menu.
2. From the *Solution Type* dialog box that appears, select *one* of the follow four available options:
   - **Modal:** To perform a natural frequency analysis, which determines the frequencies at which a structure vibrates and the associated mode shapes.
   - **Steady-State Thermal:** To determine the temperature and heat flux distribution when the model reaches a state of thermal equilibrium based on the assigned constant boundaries and excitations.
   - **Transient Thermal:** To perform a thermal analysis in the time domain, which determines how temperature and heat flux distribution vary with time. Assigned thermal boundaries and excitations can vary with time for this solution type.
   - **Structural:** To perform a stress analysis, which determines displacements and stresses resulting from thermal expansion or contraction and assigned displacements, forces, or pressures.



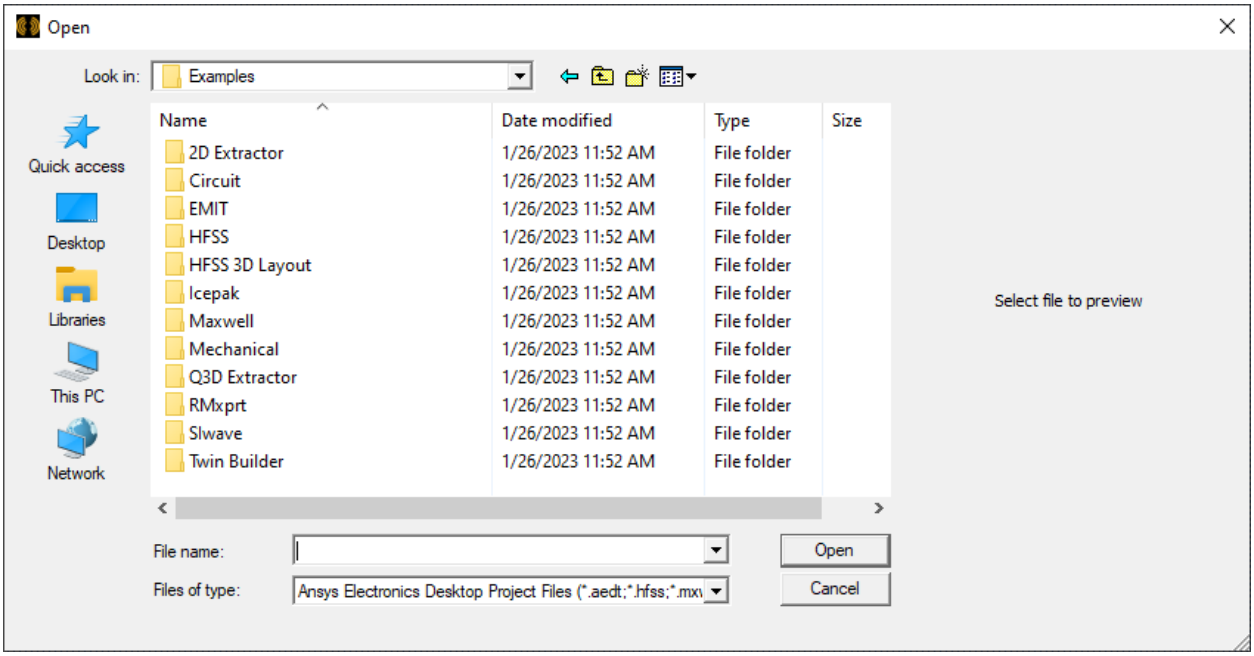
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> - Both *Modal* and *Structural* solution types have unique licensing requirements that must be satisfied for the solutions to run successfully.

3. Optionally, select the **Save As Default** option if the current *Solution Type* choice is not already the default and is what you expect to perform for the majority of *Mechanical* designs.
4. Click **OK**.

Example Projects

Your Ansys Electronics Desktop installation includes an example directory containing projects folders for several kinds of designs.



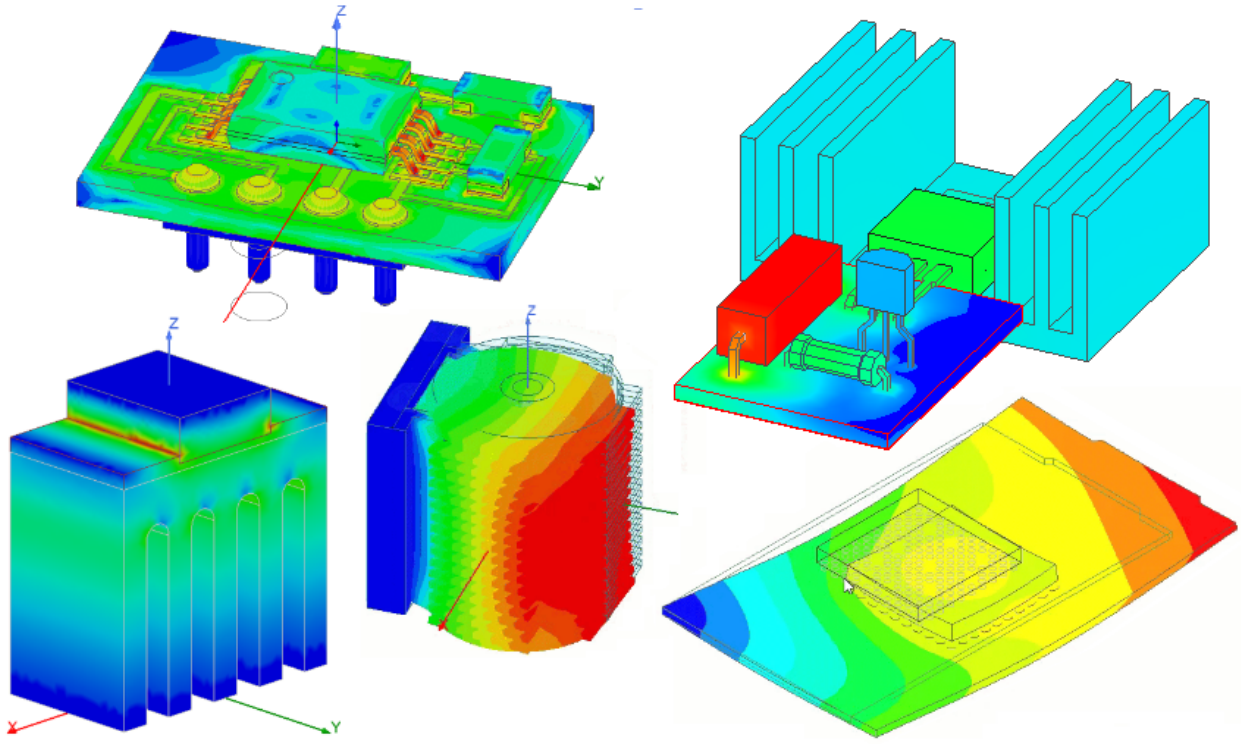
Example projects are organized by the design type.

Several of these projects are associated with detailed getting started guides.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanical Example Projects

The Mechanical examples folder contains four projects—one modal, one thermal, and two structural. These projects are fully set up with boundaries, excitations, solution setups, results plots, and mesh and field overlays. However, the solutions have not been run, and you cannot review the results until you save the projects to a working folder and solve the analyses. These projects are not associated with any Mechanical Getting Started Guides but are in addition to them.



Click one of the following links for more information:

- **Modal Example**:
  - Motor Housing
- **Thermal Examples:**
  - **Steady-State:**
    - Transistor with Heat Sink
  - **Transient:**
    - Current Regulator
- **Structural Examples:**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- PCB Stress
- Substrate Warpage

Modal Example – Motor Housing

This project is a natural frequency analysis of an electric motor housing and stator assembly, which has the following characteristics:

- **Filename and Path** (relative to the *Examples* folder of the program installation):

  Mechanical\Modal\Motor Housing.aedt

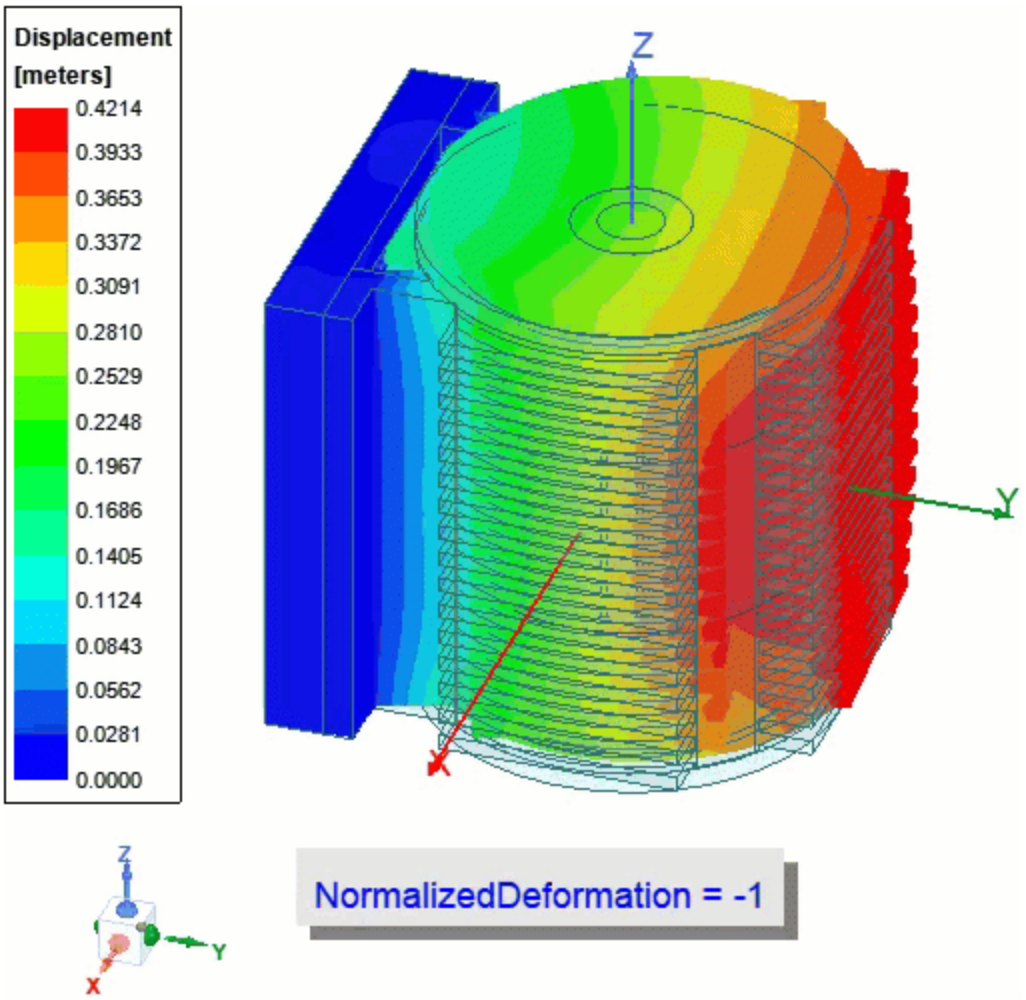
- **Number of Parts:** 7
- **Materials:** Custom defined

  "SS, Housing" (housing assembly, six parts):
  - Mass Density = 7750 kg/m^3
  - E = 193 x 10^9 N/m^2
  - Poisson's Ratio = 0.31

  "Stator" (one part):
  - Mass Density = 7750 kg/m^3
  - E = 193 x 10^9 N/m^2
  - Poisson's Ratio = 0.31
- **Mesh Operation:** Length Based, Inside Selection = 15 mm (all solid objects)
- **Boundary:** Fixed Support, base plate outer face only (-Y side of model)
- **Number of Modes Calculated:** 10
- **Results:**
  - Frequency Table
  - Effective Mass Ratio Table
- **Field Overlays:**
  - Mesh Plot
  - Displacement Magnitude

The following animation shows the vibration mode shape for mode 1 (1.3 kHz):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Steady-State Thermal Example – Transistor with Heat Sink

This project determines the resultant steady-state temperature and heat flux results for a transistor mounted to a copper plate that is in turn mounted to an aluminum heat sink. Heat is introduced by two sources—heat generation within the transistor body and heat flux applied to two copper plate faces. Cooling is by convection at selected faces of the heat sink fins.

Calculator expressions have been defined to determine the maximum, minimum, and mean temperature results.

The model has the following characteristics:

- **Filename and Path** (relative to the *Examples* folder of the program installation):

  Mechanical\Thermal\Transistor with Heat Sink.aedt

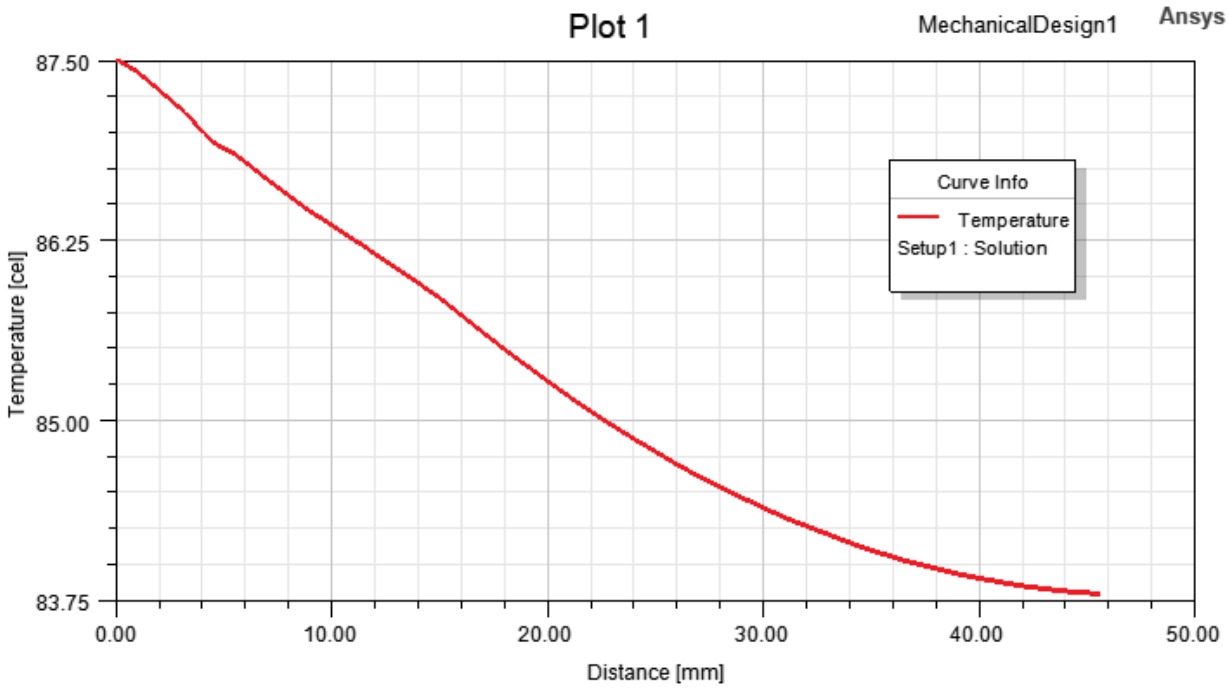
- **Number of Parts:** 3 solid, 1 polyline

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Materials:**
  - aluminum
  - copper
  - Si-Typical (silicon)
- **Mesh Operation:** Length Based, On Selection = 3.5 mm (all solid objects)
- **Boundary:** Convection (assigned to twenty-one faces)
  - Film Coefficient = 50 W/(m^2·°C)
  - Ambient Temperature = 46° C (115° F)
- **Excitations:**
  - Heat Generation = 20 W (Box3)
  - Heat Flux = 2.5 W (distributed over two faces of Box2)
- **Results:**
  - Calculation Expression Table (min, mean, and max temperature)
  - Temperature Plot (along Polyline1)
- **Field Overlays:**
  - Mesh Plot
  - Heat Flux Vectors
  - Heat Flux Magnitude
  - Displacement Magnitude

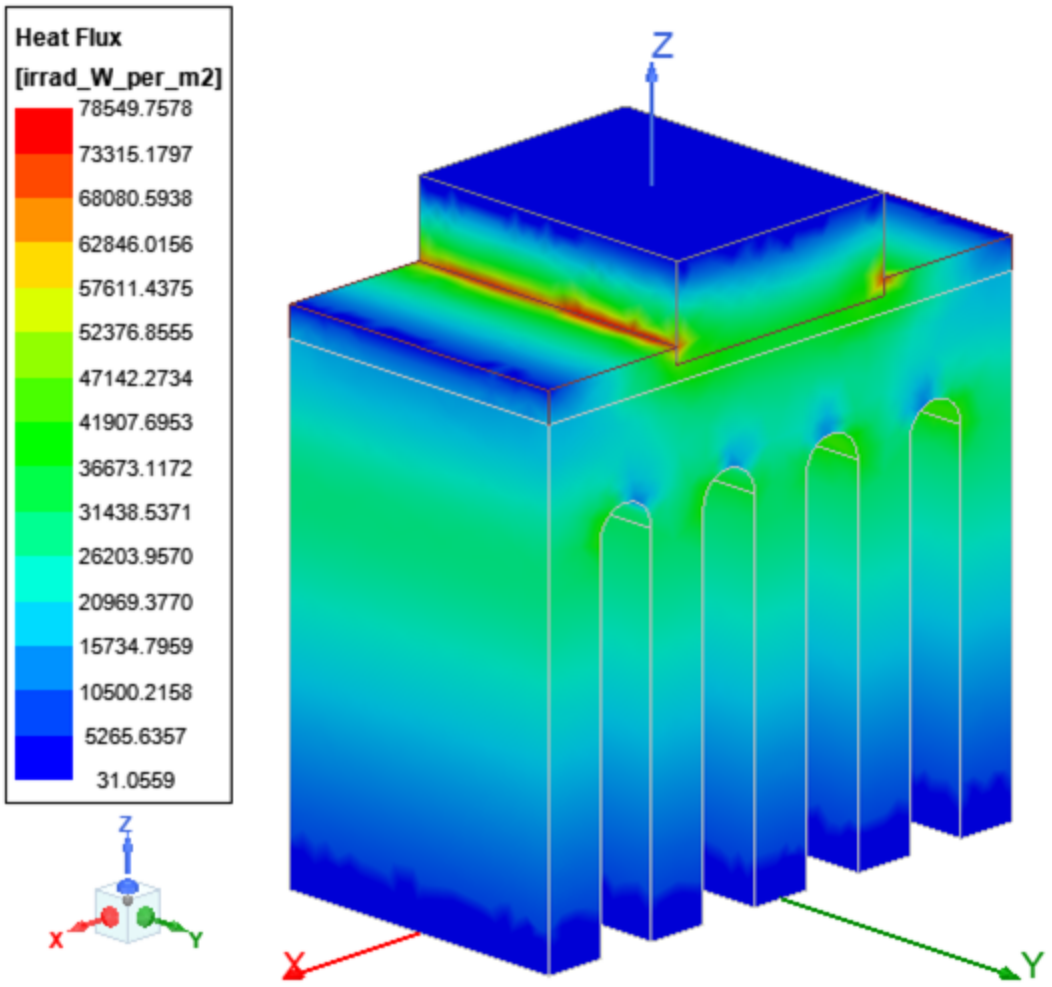
The following image is the temperature variation along Polyline1:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following image is the heat flux magnitude overlay:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Transient Thermal Example – Current Regulator

This project is a current regulator module consisting of a small printed circuit board (PCB), two bipolar transistors, two resistors, and a heat sink. It demonstrates the transient thermal solution type and trace mapping using a layout component based on a PCB created in an HFSS 3D Layout design. The transient thermal results include temperature versus time for several monitor points and mapped thermal conductivity in the PCB signal layer. Each of the discrete components generates a different amount of heat, and all of them are cycled on and off at an 80% duty cycle throughout the analysis. Convection boundaries are assigned to the PCB's top and bottom faces, the components, and the heat sink.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

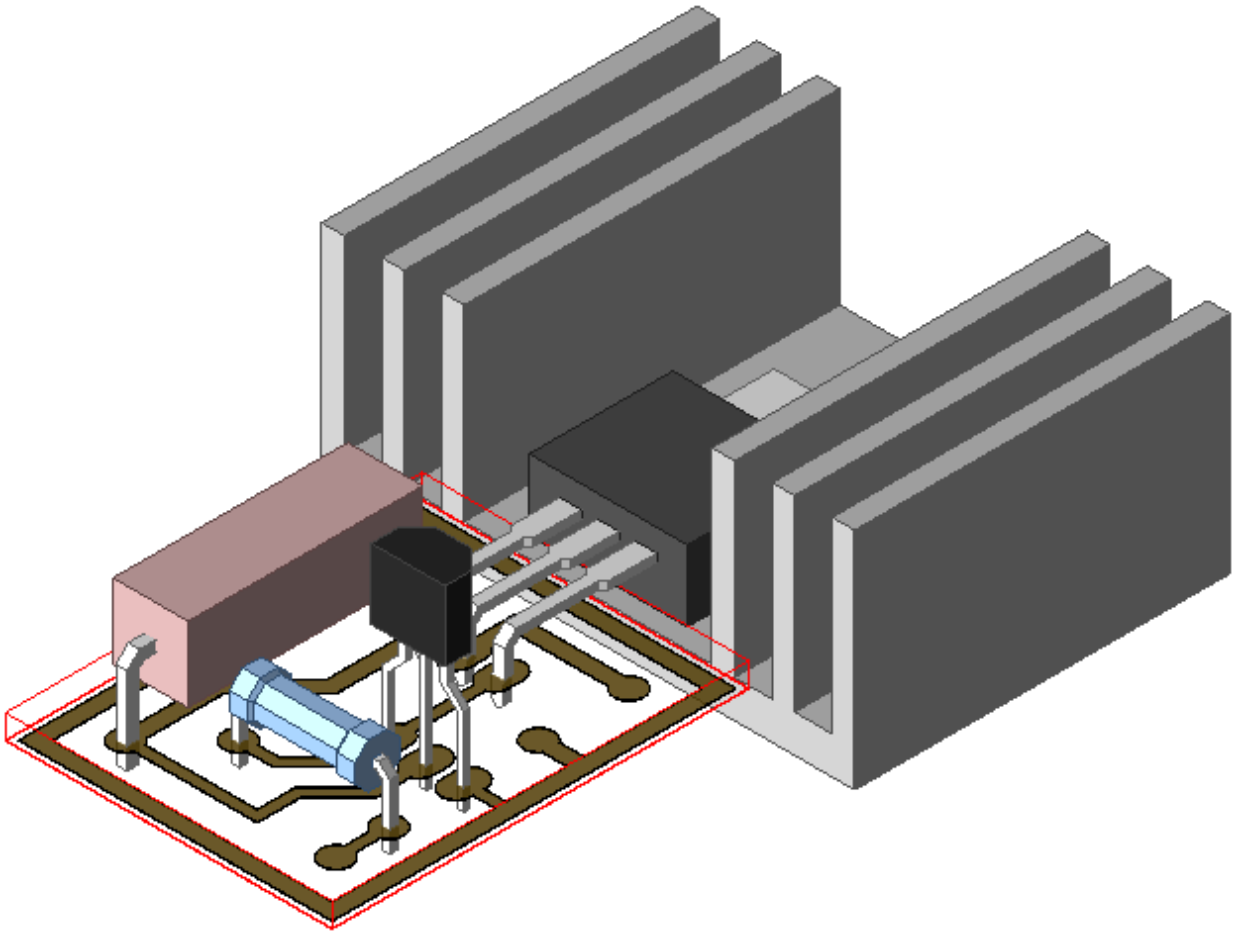


Figure 1: Current Regulator Model Geometry

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

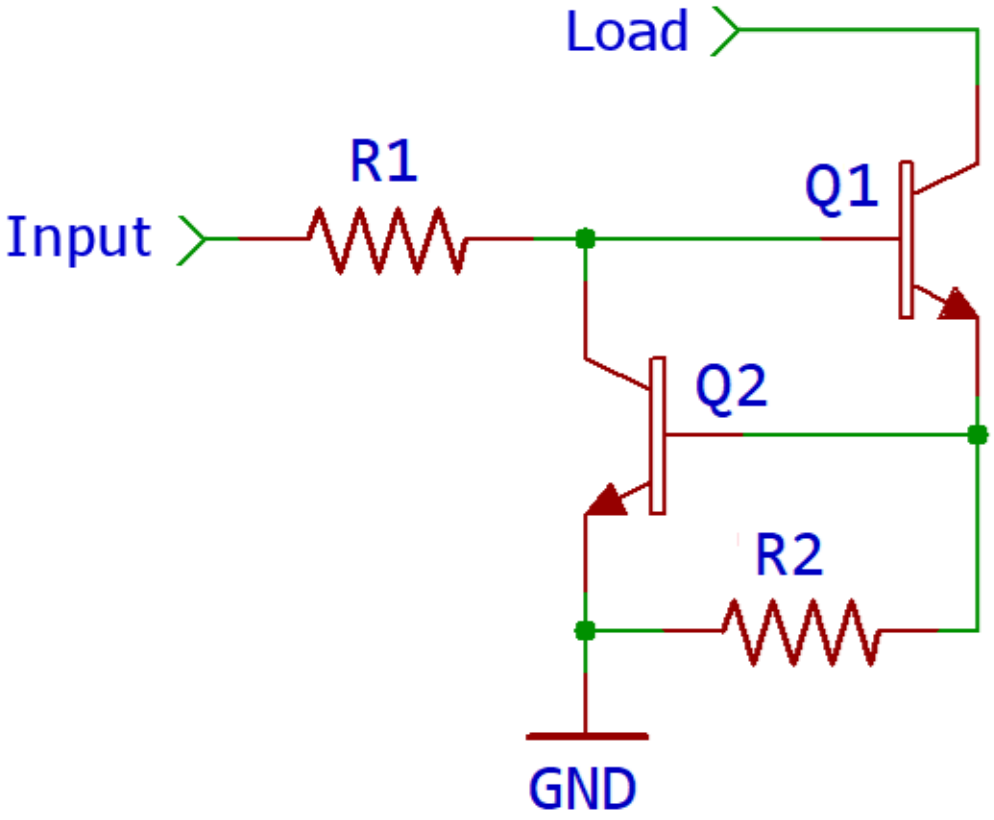


Figure 2: Current Regulator Schematic Diagram

Periodic, piecewise linear datasets are used to control the heat generation of each component versus time. Only one on/off period needs to be defined for each excitation's dataset. The datasets are extrapolated periodically for the second through last cycles.

Contact is assigned to the bottom of the power transistor (Q1) to account for thermal contact resistance between it and the heat sink. Q1 is imprinted on the heat sink to split the mounting face. The portion of the heat sink under Q1 is excluded from the assigned convection boundary, as is the bottom of Q1.

Rather than saving fields for multiple time steps, thermal monitors are used to display how temperatures vary with time. This setup saves on memory usage and solution time. Fields are available for overlaying on the model only for the first and last timestep, as defined in the solution set up.

The model has the following characteristics:

- **Filename and Path** (relative to the *Examples* folder of the program installation):

  Mechanical\Thermal\Transient_CurrentRegulator.aedt

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Number of Parts:** 1 layout component (LC_1) and 16 solid objects (including heat sink, discrete component bodies, Q1's mounting tab, and leads)
- **Materials:**
  - copper
  - FR4_epoxy
  - aluminum (Al-Extruded)
  - Stainless steel, 304, 1/2 hard
  - Transistor_Bodies (user-defined)
  - Resistor_Bodies (user-defined)
- **Design Settings:** *Enable material overrides* is Selected (due to intersection of component leads and LC_1)
- **Contact:** Assigned to the bottom face of Q1_body and Q1_tab
  - Resistance Type = Thermal Impedance
  - Thermal Impedance = 0.45 $°C \cdot in^2/W$
- **Boundaries:** Convection
  - Assigned to LC_1 (PCB), top and bottom faces (as part of the component definition)
  - Assigned to all discrete component objects except for Q1_tab and Heat_Sink
  - Assigned to exposed faces of Q1_tab and Heat_Sink (contact face excluded)
  - Film Coefficient = 20 $W/(m^2 \cdot °K)$
  - Ambient Temperature = 20° C (68° F)
- **Excitations:** Heat Generation, 80% duty cycle, 16 seconds on, 4 seconds off, repeating
  - R1_body: 162 mW
  - R2_body: 875 mW
  - Q1_body: 3.75 W
  - Q2_body: 84 mW

  A dataset has been defined for each excitation. All four are the same except for the magnitude of the power when ON. The timestep size is 4 seconds and the period of each cycle is 20 seconds. Therefore, there are five timesteps per power cycle. The dataset is designed so that power is at 100% for the first four of the five timesteps in each cycle and OFF for the fifth, reflecting the desired 80% duty cycle (4/5 = 0.8). The datasets are referenced as a piecewise periodic function so that the cycle repeats continuously every twenty seconds.

  The final timestep (at 496 seconds) is 16 seconds into the final cycle, just before power is cut OFF. The reason is so that maximal temperature field results are saved, since the solution setup only saves field results for the first and last steps. If the final timestep were defined at the conclusion of a 20-second power cycle, somewhat lower temperatures would be captured due to four seconds of cooling at the end.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the dataset associated with Q1_body's heat generation excitation:

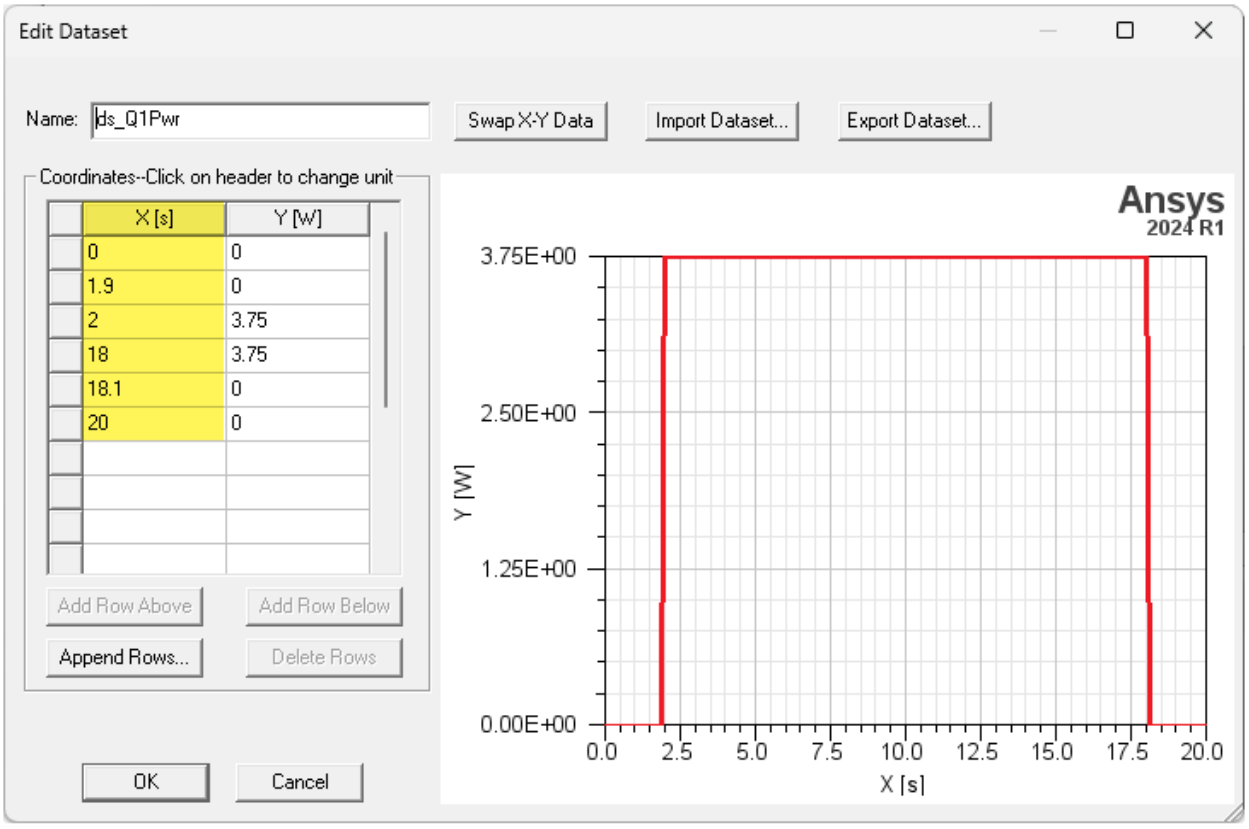


Figure 3: Definition of *ds_Q1Pwr* Dataset

- **Thermal Monitor Points** (object centroids, in addition to the global maximum) – see Figure 4:
  - R1_body
  - R2_body
  - Q1_body
  - Q2_body
- **Mesh Settings:**
  - Initial Mesh Settings: Default (*Mesh Method* = Auto, *Use Slider* selected, *Slider* centered)
  - Length-Based Mesh Operation assigned to Top and Bottom layers of LC_1: *Max Length* = 0.88 mm
- **Transient Solution Setup:**
  - Start time: 0 seconds
  - Stop time: 496 seconds (8 minutes : 16 seconds)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Time step: 4 seconds
- Field save times: None (only first and last time step fields are saved)

- **Results:**
  - Calculator Expression Table (global min, mean, and max temperatures) – see Figure 5
- **Field Overlays:**
  - Mesh Plot (all objects) – see Figure 6
  - Temperature @ 496 seconds (all objects) – see Figure 7
  - Thermal Conductivity X (bottom layer of LC_1) – see Figure 8
  - Thermal Conductivity Y (bottom layer of LC_1) – see Figure 9

Solution Time: 20 to 30 minutes for a relatively high-end workstation

Result Images

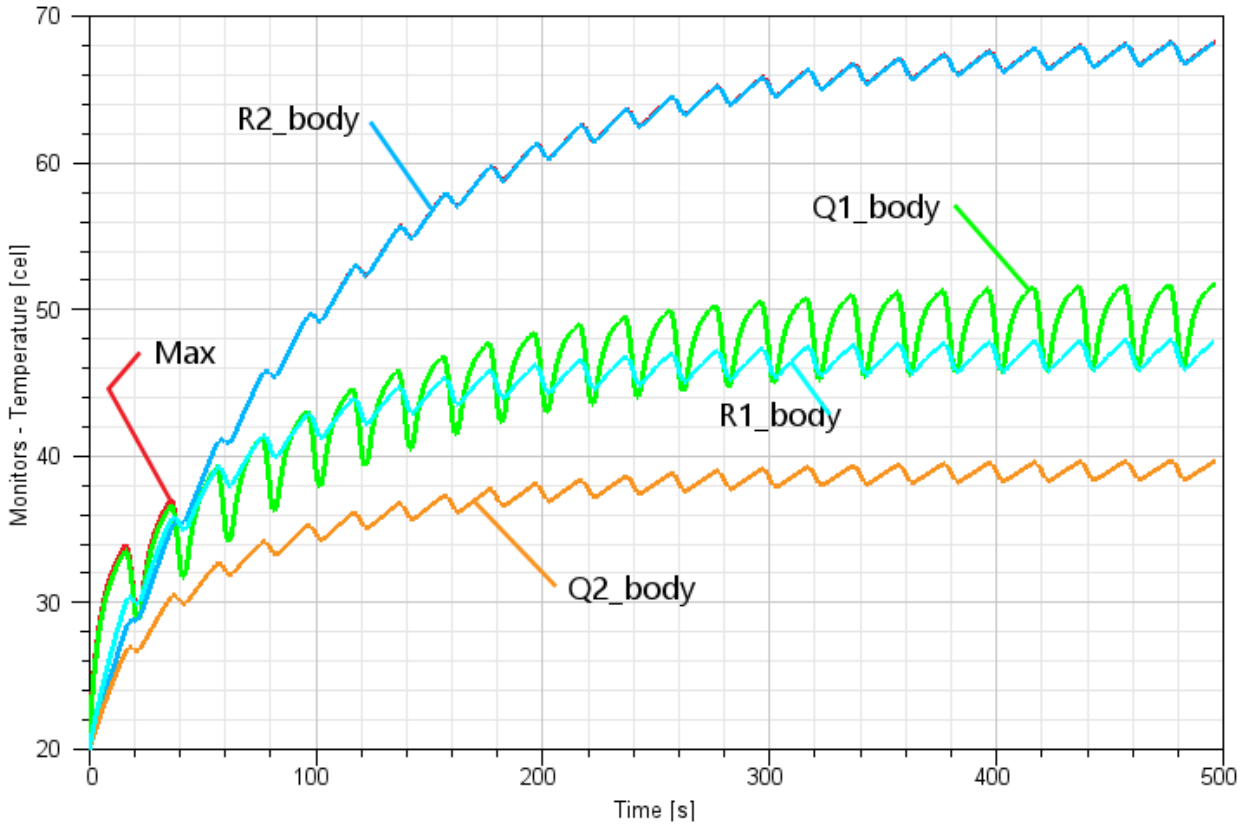


Figure 4: Thermal Monitor

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Observations:

- Initially, the maximum temperature occurs in Q1_body. However, R2_body continues to heat rapidly while Q1_body's rate of temperature increase is declining. The heat sink takes away heat more rapidly from Q1 than R2's limited convection area can. Beyond about 40 seconds, the maximum temperature coincides with the R2_body temperature, and this remains the hottest part throughout the remainder of the simulation. The red trace is completely obscured by the blue trace, and R2_body's monitor point must be disabled to see the rest of the red trace.
- The coolest of the monitored objects throughout the simulation is Q2_body, which is not surprising. It has the smallest heat generation power in addition to more lead surface area to dissipate heat via convection.
- Near the end of the transient solution, quasi-steady-state temperature cycles are approached but not quite achieved. The exception is Q2_body, which is no longer exhibiting any significant heating trend. Q1_body and R1_body have significantly leveled off and are close to quasi-steady-state temperature cycles. R2_body has a bit longer to go than the other objects before achieving a flat temperature trend.

Calculator Expressions Table 1 MechanicalDesign1 Ansys 2024 R2

	Time [s]	Global_Max_Temp Setup1 : Solution	Global_Mean_Temp Setup1 : Solution	Global_Min_Temp Setup1 : Solution
1	4.000000	27.829375	21.127810	20.139094
2	496.000000	68.564686	43.211577	33.401719

Figure 5: Global Maximum, Mean, and Minimum Temperatures – First and Last Time Steps

Only the first and last time step are included in the preceding tabular report because these fields are saved by default, and the option to save additional fields was not selected in the solution setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

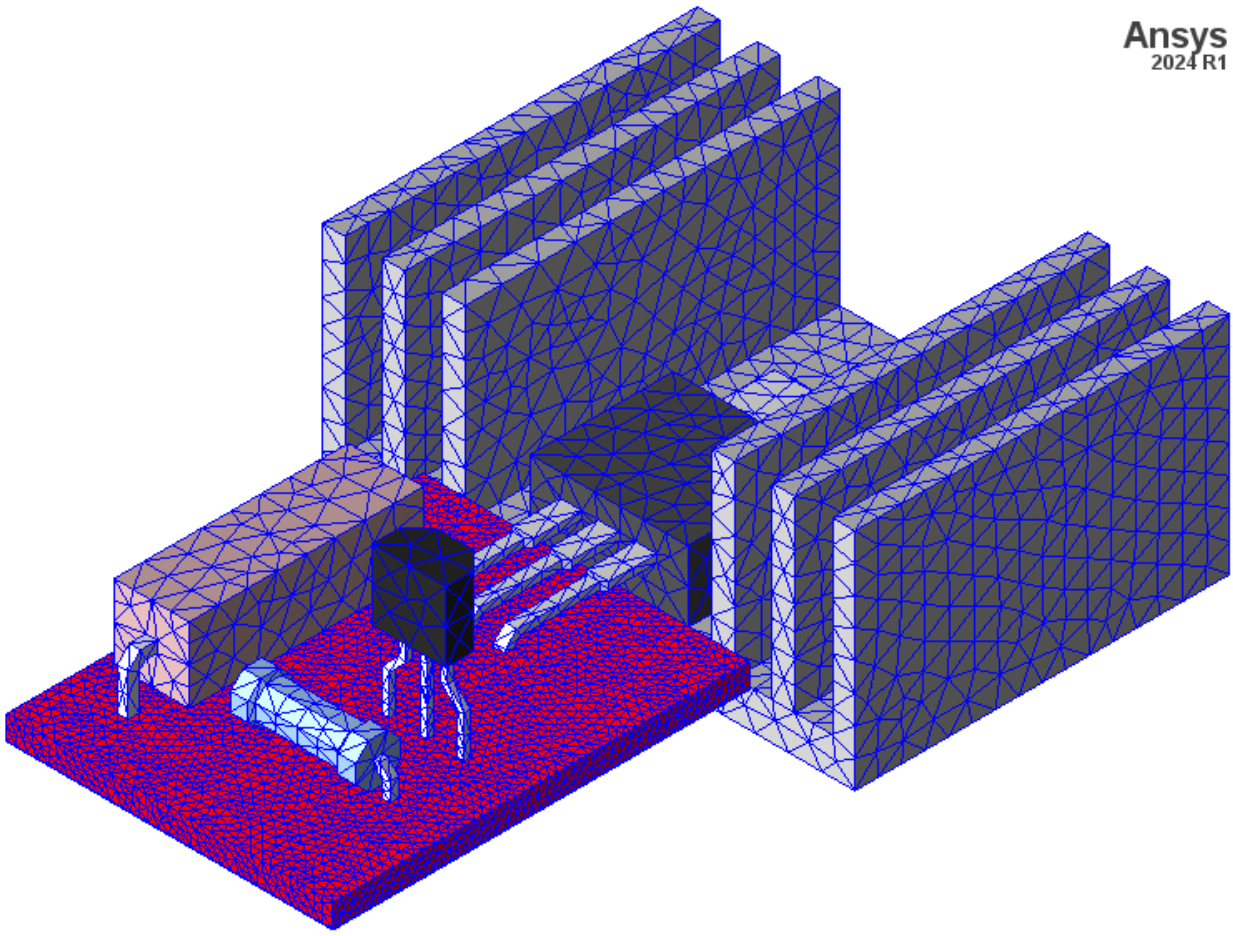


Figure 6: Mesh Plot

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

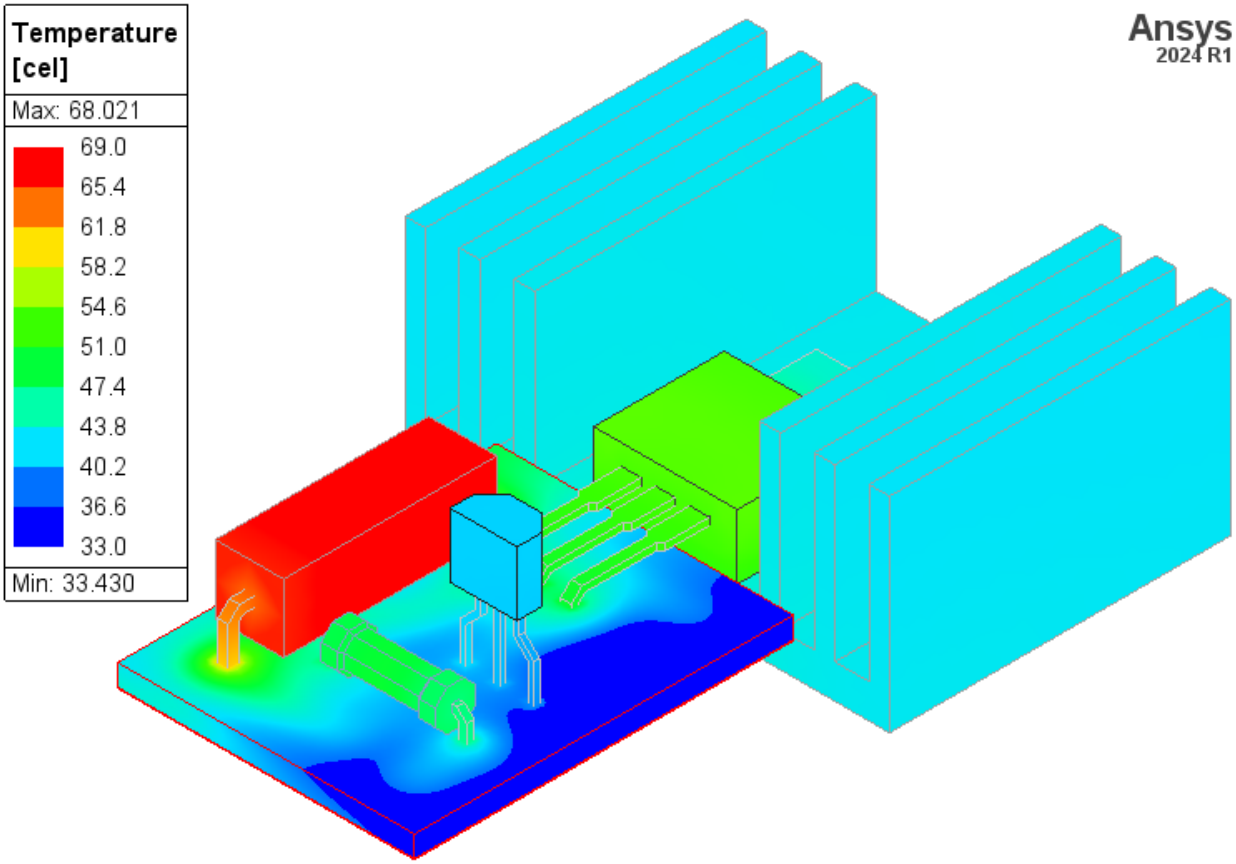


Figure 7: Temperature Overlay, Time = 496 Seconds

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

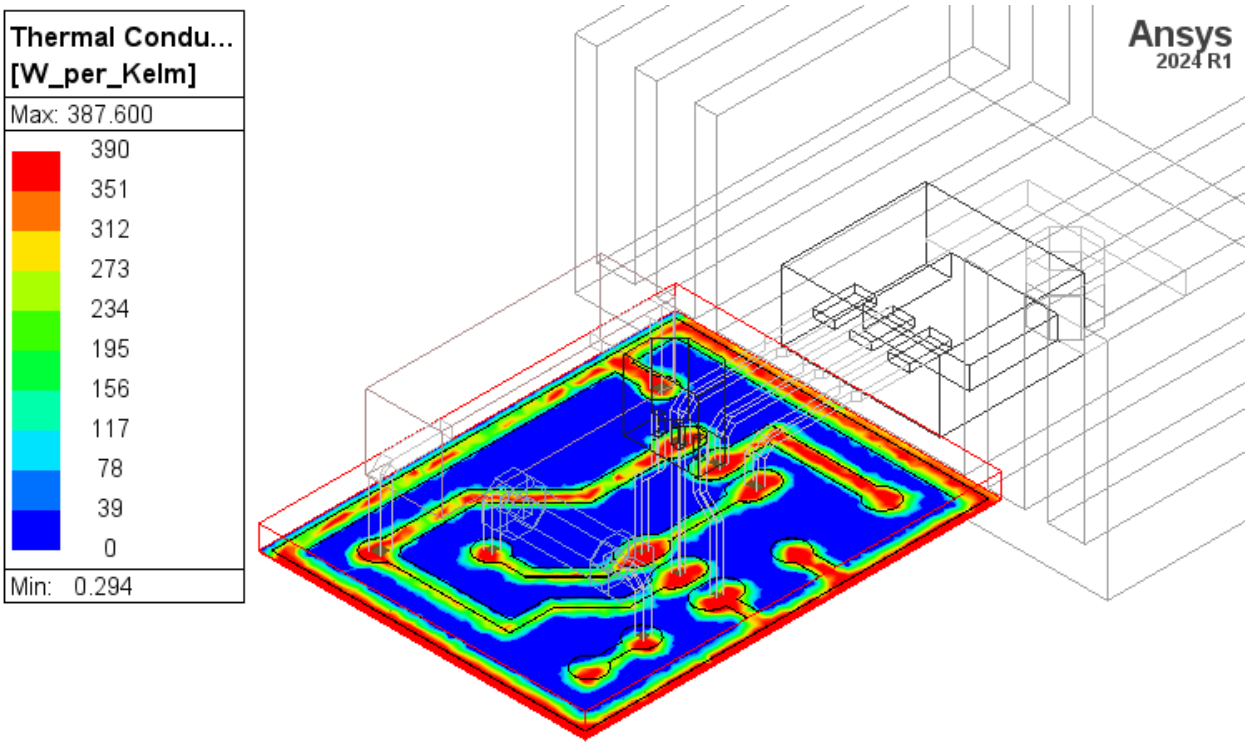


Figure 8: Thermal Conductivity, X-Direction, LC_1, Bottom (Signal) Layer

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

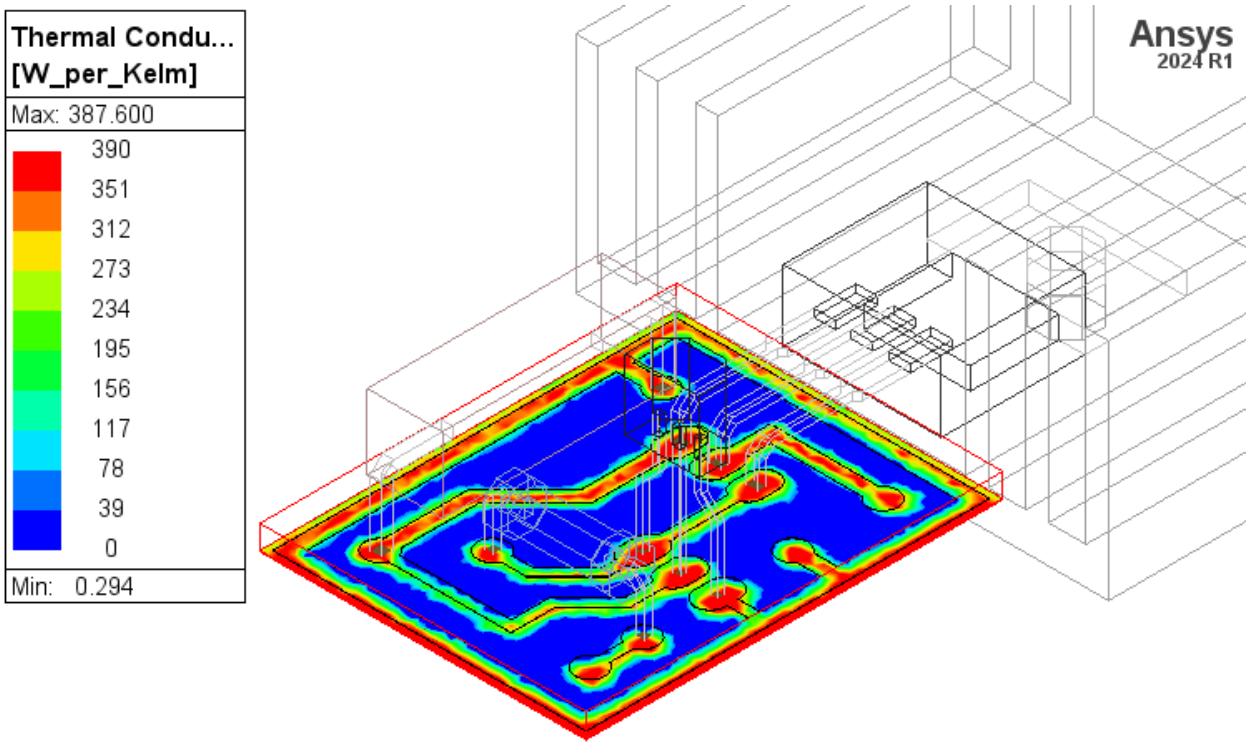


Figure 9: Thermal Conductivity, Y-Direction, LC_1, Bottom (Signal) Layer

Structural Example – PCB Stress

This project determines the displacement and equivalent stress results for a small printed circuit board (PCB) due to thermal expansion at operating temperatures. A total of three different uniform temperatures are assigned to sets of objects, as detailed below.

The model has the following characteristics:

- **Filename and Path** (relative to the *Examples* folder of the program installation):

  Mechanical\Structural\PCB_Stress.aedt

- **Number of Parts:** 51 solids, 2 polylines

  The solid objects are grouped by components and materials, as follows:

  - **Mount:** Object with soft material used to indirectly fix the PCB without impeding thermal expansion
  - **P1:** Four-pin plug used to attach the PCB to a parent circuit board, which is not included in the model
  - **PCB:** Consists of the circuit board and its conducting traces
  - **Q1:** Surface mounted transistor (consisting of body, terminals, and solder objects)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

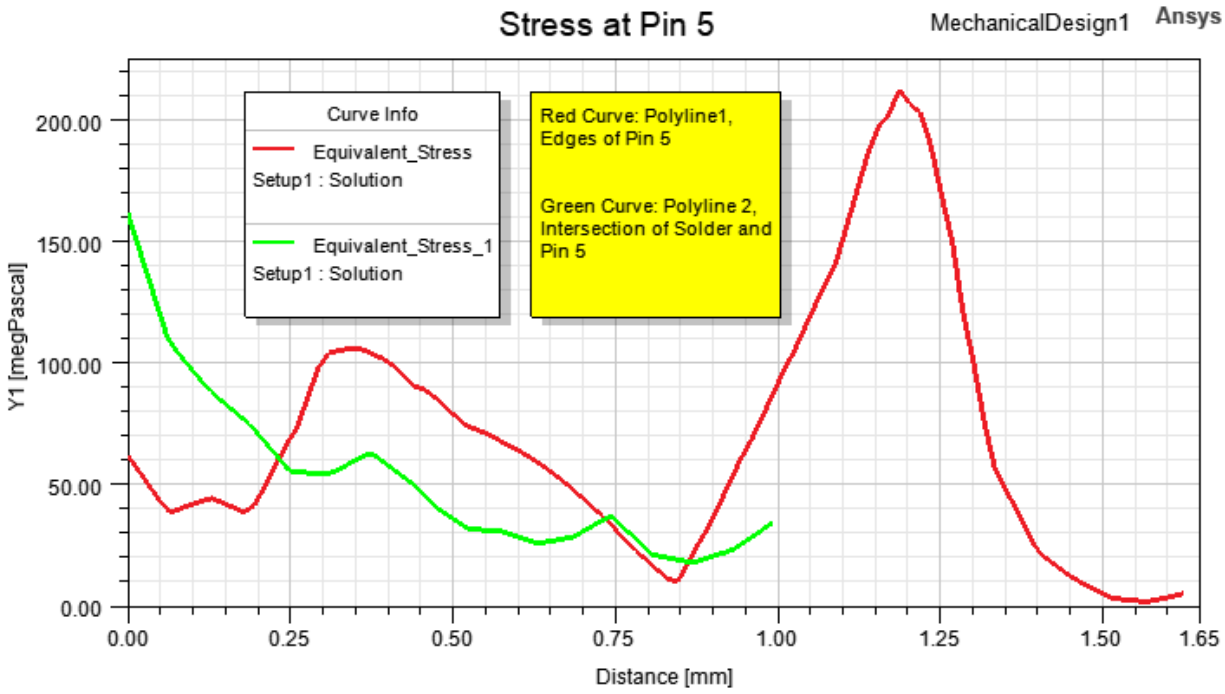
    - **R1** and **R2:** Surface mounted resistors (consisting of bodies, terminals, and solder objects)
    - **U1:** Eight-pin surface mounted integrated circuit (consisting of body, pins, and solder objects)

  The polylines are used for plotting equivalent stresses along the following most highly stressed areas:

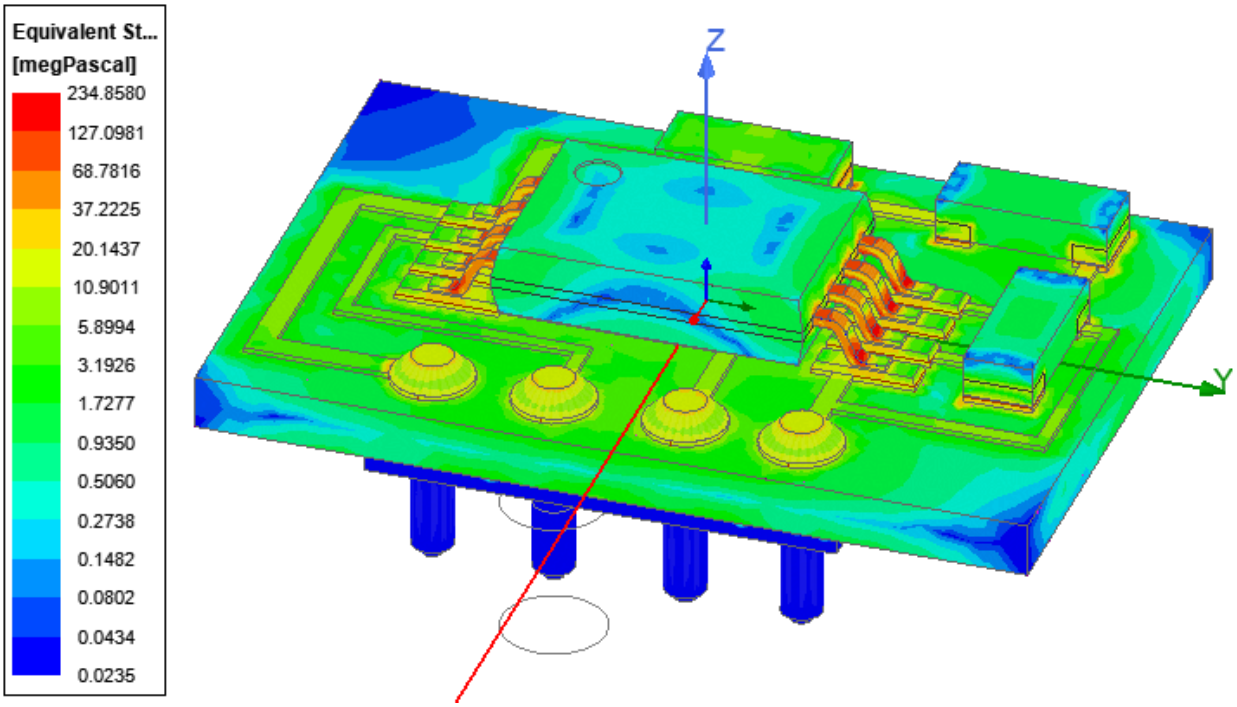
    - **Polyline1:** Length of pin 5 (runs along one corner edge)
    - **Polyline2:** Intersection of pin 5 and solder (three of four edges)
- **Materials:**
    - **copper:** Conducting traces, terminals, and pins
    - **FR-4:** PCB dielectric
    - **My_Plastic:** Plug body, user-defined — ρ = 1,850 kg/m^2; E = 1.5E7 N/m^2; v = 0.45; α = 2.5E-5/C
    - **My_Resistor:** Resistor bodies, user-defined — ρ = 1,800 kg/m^2; E = 1.1E9 N/m^2; v = 0.33; α = 7.5E-5/C
    - **My_Semiconductor:** Q1 and U1 bodies, user-defined — ρ = 2,000 kg/m^2; E = 1.2E10 N/m^2; v = 0.35; α = 7.5E-5/C
    - **SoftMount:** Mount, user-defined — ρ = 1,000 kg/m^2; E = 1E6 N/m^2; v = 0.45; α = 2E-5/C
    - **solder:** All pin/terminal-to-conducting trace interfaces
- **Mesh Operation:** Length Based, On Selection = 0.6 mm (bodies of PCB, R1, R2, Q1, and U1; P1 pins)
- **Boundary:** Fixed Support (bottom of Mount)
- **Excitations:**
    - **ThermalCondition1_38C:** Uniform (P1 body, pins, and solder; PCB body and traces)
    - **ThermalCondition1_45C:** Uniform (R1 body, terminals, and solder; U1 body, terminals, and solder)
    - **ThermalCondition1_50C:** Uniform (R2 body, terminals, and solder; Q1 body, terminals, and solder)
- **Results:**
    - **Stress at Pin 5:** Equivalent stress along Polyline1 and Polyline2
    - **Design Table 1:** Volume of Q1 body and terminals, with total
- **Field Overlays:**
    - **Mesh Plot** (undeformed)
    - **Equivalent Stress** (undeformed)
    - **Displacement Magnitude** (deformed, 150 times actual; animation defined)

The following image is the 2D plot of equivalent stress along Polyline1 and Polyline2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

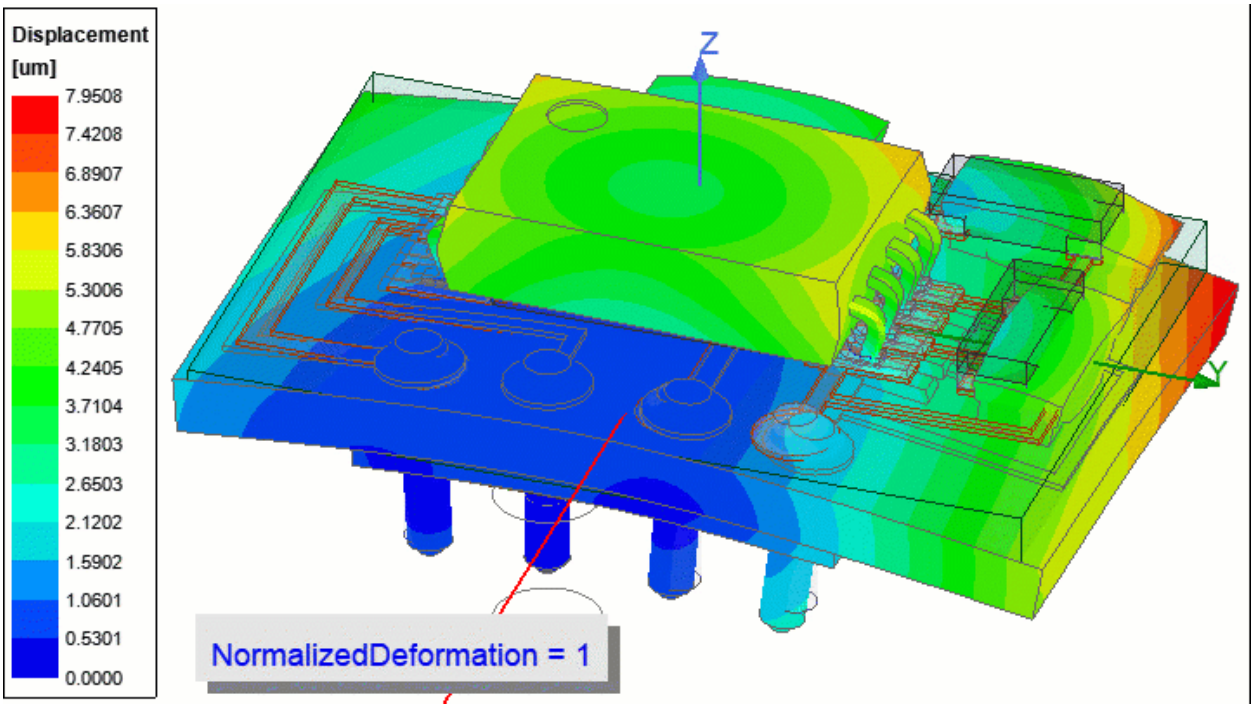


The following image is the equivalent stress overlay:



The following animation is the displacement magnitude overlay:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Structural Example – Substrate Warpage

This project determines the warpage of a semiconductor substrate and the printed circuit board (PCB) on which it is mounted. The PCB substrate, and solder objects are at an assumed uniform temperature of 120° C. The environment temperature (at which the model is stress free) is 20° C. Therefore, expansion is based on a ΔT of 100° C. The device has 90 pins arranged in a grid. The analysis looks at the equivalent stresses in all parts and particularly in the solder connecting the PCB and substrate.

Since the displacement magnitudes are dominated by elongation and lateral expansion of the parts, two calculator expressions and two face lists have been defined to provide a warpage indication. Specifically, the Z-displacement (normal direction of the substrate and PCB planes) is the best indication of warpage. The calculator expressions determine the difference between the minimum and maximum Z-displacements for the top face of the PCB and the bottom face of the substrate, which are passed to the field calculator as face lists. A data table report is also defined to tabulate the two warpage indicators (in millimeters).

An imprinting operation has been used to create a total of four very small rectangular subfaces at three different corners of the PCB. These small faces provide a means of physically constraining the model (eliminating rigid body motion to make it statically stable). At the same time, constraints were able to be assigned in a manner that neither inhibited the thermal expansion and warpage of the model nor generated any local stress exaggerations.

The model has the following characteristics:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Filename and Path** (relative to the *Examples* folder of the program installation):

  Mechanical\Structural\SubstrateWarpage.aedt

- **Number of Parts:** 92 solids

  The solid objects are grouped by components and materials, as follows:

  - **PCB:** Consists of the circuit board substate only, without copper traces
  - **Substrate:** The substrate of a very large-scale integration (VLSI) semiconductor
  - **Solder1** through **Solder90:** Interconnecting solder grid between the PCB and substrate, with each one a cylindrical object
- **Materials:**
  - **FR-4:** *PCB* dielectric
  - **silicon:** Semiconductor *Substrate*
  - **Solder Eutectic Sn-37Pb:** All *Solder* objects
- **Mesh Operations:** No mesh operations; default initial mesh settings

  > **Note:**
  >
  > The initial mesh slider has been enhanced to produce suitable thermal and structural solution meshes with minimal usage of manually defined mesh operations. Both flat and curved faces are now refined by this slider for these two solution types. In earlier versions of the software, mesh operations would have been required to produce a high-quality mesh at the critical solder objects and the areas near the solder contact points on adjacent parts. Now, the default position of the *Initial Mesh Settings* slider produces acceptable results.

- **Boundaries:**
  - **FixedSupport1** (at small imprinted subface at +X, -Y corner of the PCB)
  - **FrictionlessSupport1** (at small imprinted subface at +X, +Y corner of the PCB, bottom side)
  - **FrictionlessSupport2** (at small imprinted subface at +X, +Y corner of the PCB, front side)
  - **FrictionlessSupport3** (at small imprinted subface at -X, +Y corner of the PCB, bottom side)
- **Excitation:**
  - **ThermalCondition1:** Uniform, all objects, 120° C (stress-free *Environment Temperature* is 20° C)
- **Results:**
  - **Calculator Expression Table 1:** *PCB_Warpage_mm* and *Substrate_Warpage_mm* (ΔDz in millimeters at top face of PCB and bottom face of Substrate,

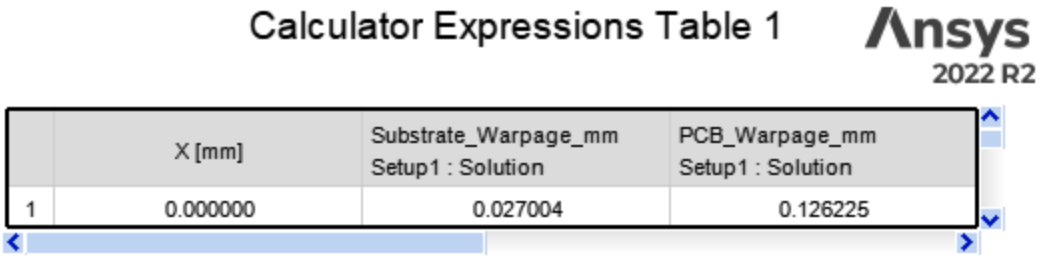
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

respectively)

- **Design Table 1:** Volume of Q1 body and terminals, with total

- **Field Overlays:**
  - **Mesh1** (undeformed): Mesh plot assigned to all objects
  - **Mesh2** (undeformed): Mesh plot assigned to solder objects
  - **Mag_Displacement1** (scaled deformation): Displacement overlay assigned to all objects; animation defined
  - **Equivalent Stress1** (undeformed): Stress overlay assigned to all objects
  - **Equivalent Stress2** (undeformed): Stress overlay assigned to solder objects

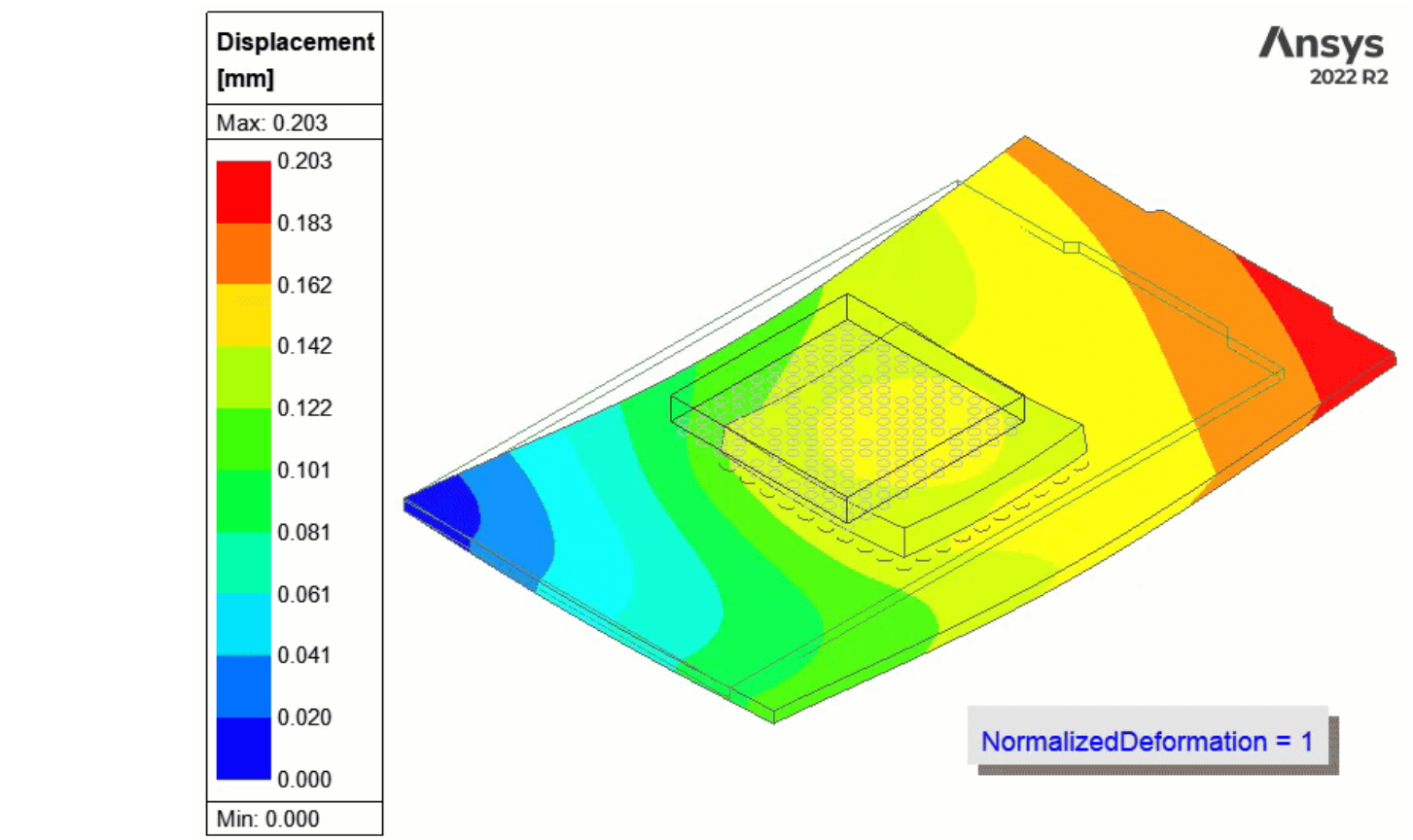
The following image is the Calculator Expression Table showing derived warpage values:



	X [mm]	Substrate_Warpage_mm Setup1 : Solution	PCB_Warpage_mm Setup1 : Solution
1	0.000000	0.027004	0.126225

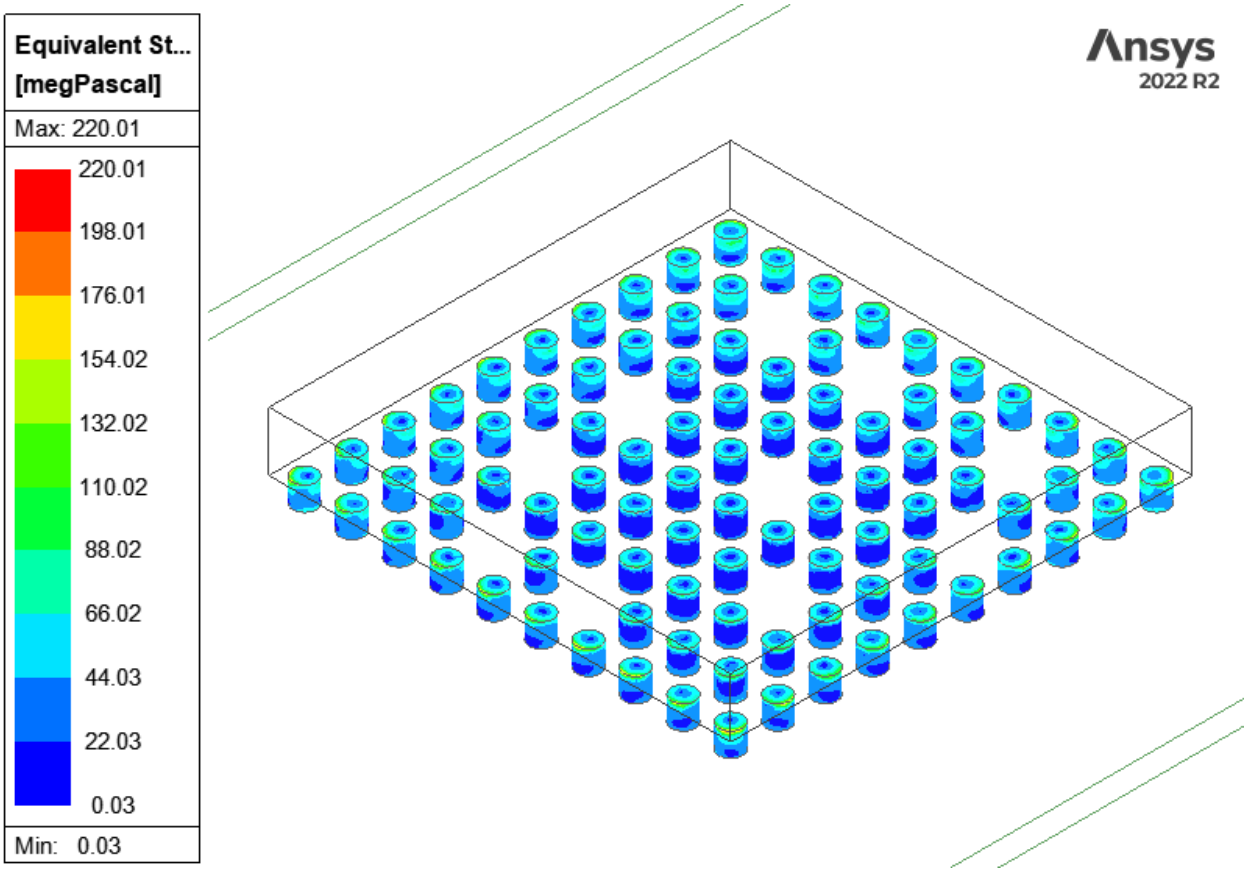
The following animation is the displacement magnitude overlay:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following image is the equivalent stress overlay assigned only to the solder objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



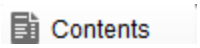
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Observations:

- The warpage of the PCB and Substrate are approximately 62.2% and 13.3% of the peak displacement magnitude, respectively.
- The maximum equivalent stress in the solder is well above the yield strength of the material, which is approximately 43 MPa. As such, all values above 43 MPa are invalid as they assume a constant Young's Modulus. In reality, the slope of the stress-strain curve decreases beyond yield, and further strain increases the stress at a much lower rate. The ultimate tensile strength of the material is 52 MPa, and breakage occurs at around 32% elongation.
- The above-yield stresses are localized at the edges of the top faces of the solder objects, particularly those along the perimeter of the solder grid. A significant portion of each solder/substrate contact face remains below the yield strength of the material.
- Solder is prone to creep at relatively low temperatures and stresses, which tends to relieve stresses in solder connections. Considering this characteristic and the high amount of elongation the material can survive, the solder objects will likely not fail despite the localized high stresses that were calculated.

Finding Information in the Online Help

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

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – 

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

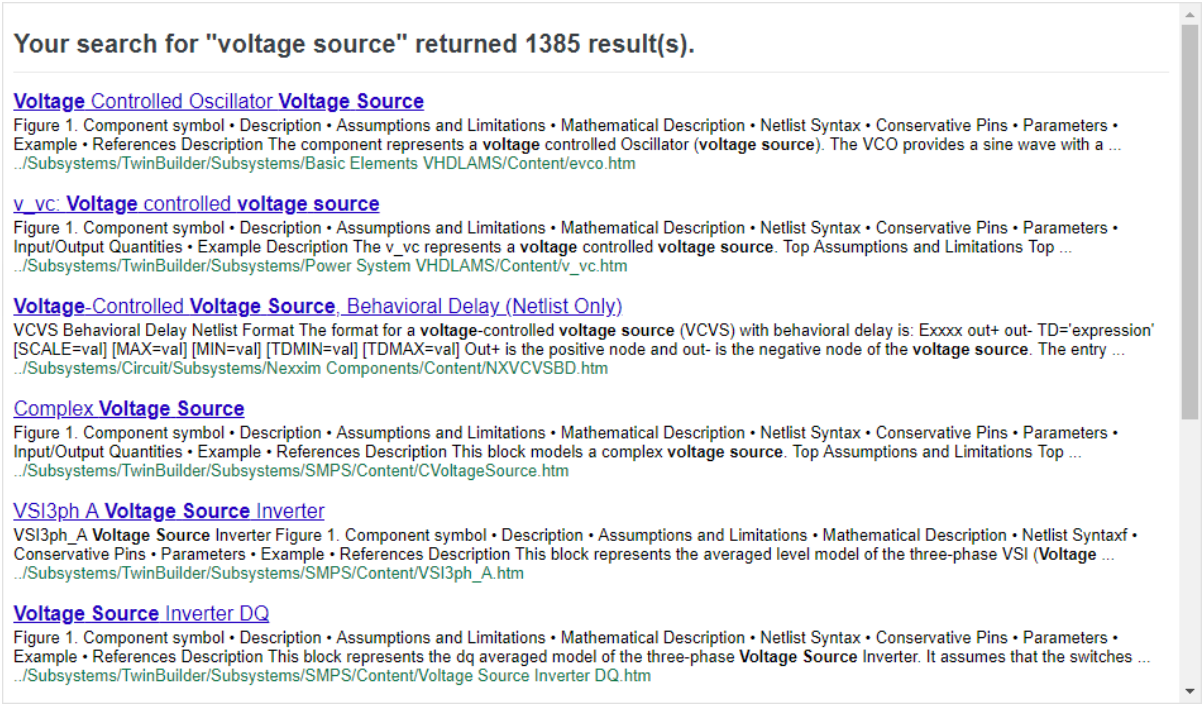
Note:

Ansys Electronics Desktop Student includes access to PDF documentation only.

Using the Search Function in the Help

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

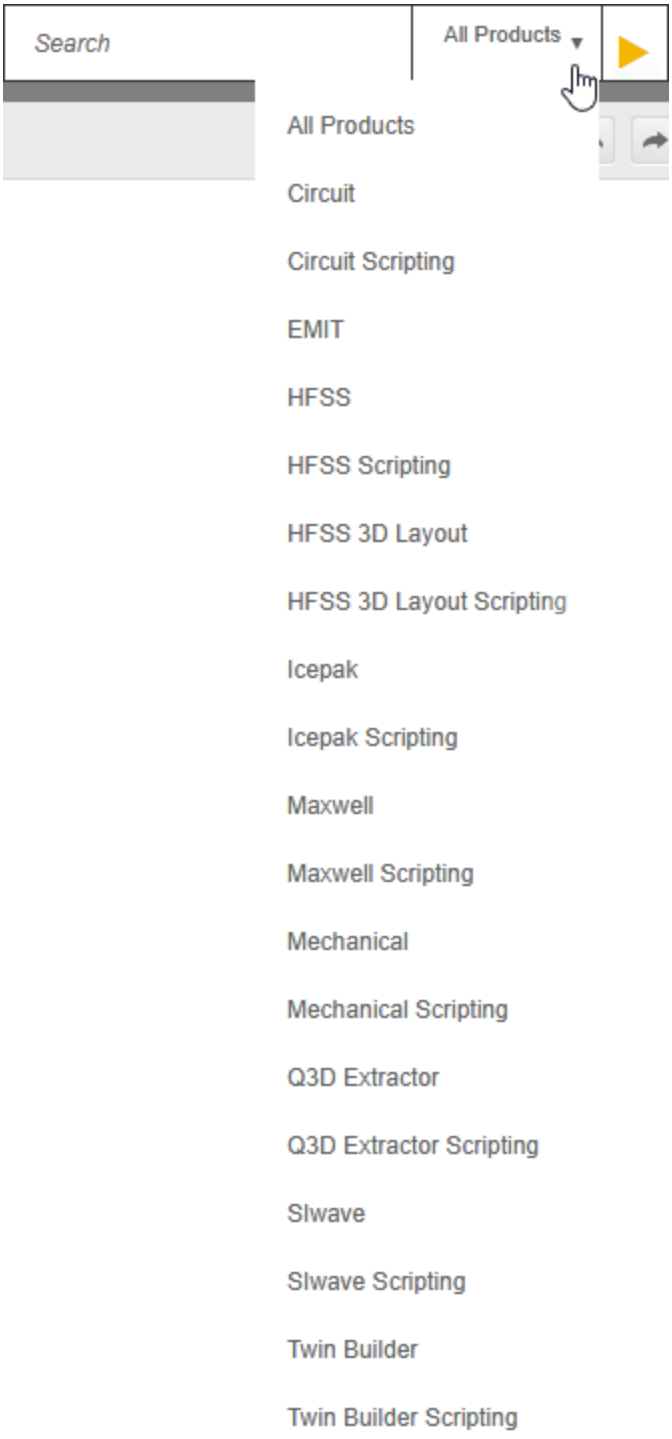


This method probably provides more hits than you want. The search function in the help provides several methods for making searches more specific.

Performing a Basic Search

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



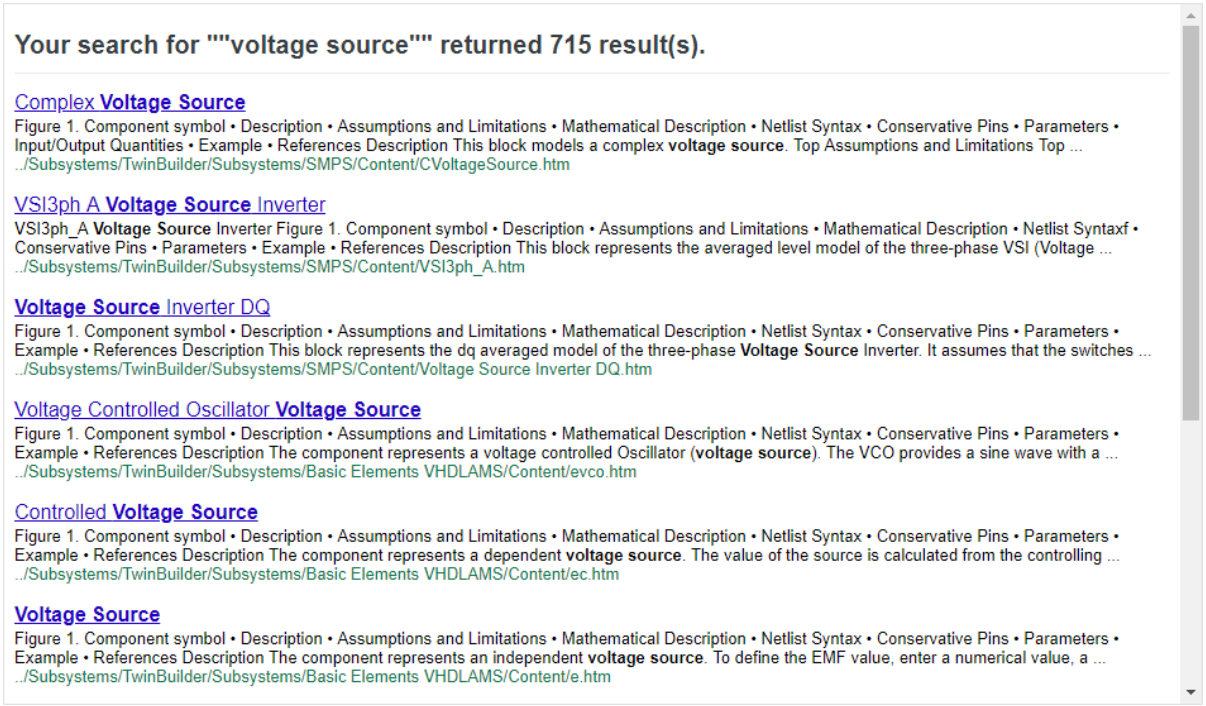
- If you accessed the help for a specific product by pressing **F1** in Electronics Desktop, the search box will have preselected that product. You can change the selection to a different product, a product's scripting guide, or all products.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Searching with Quotation Marks

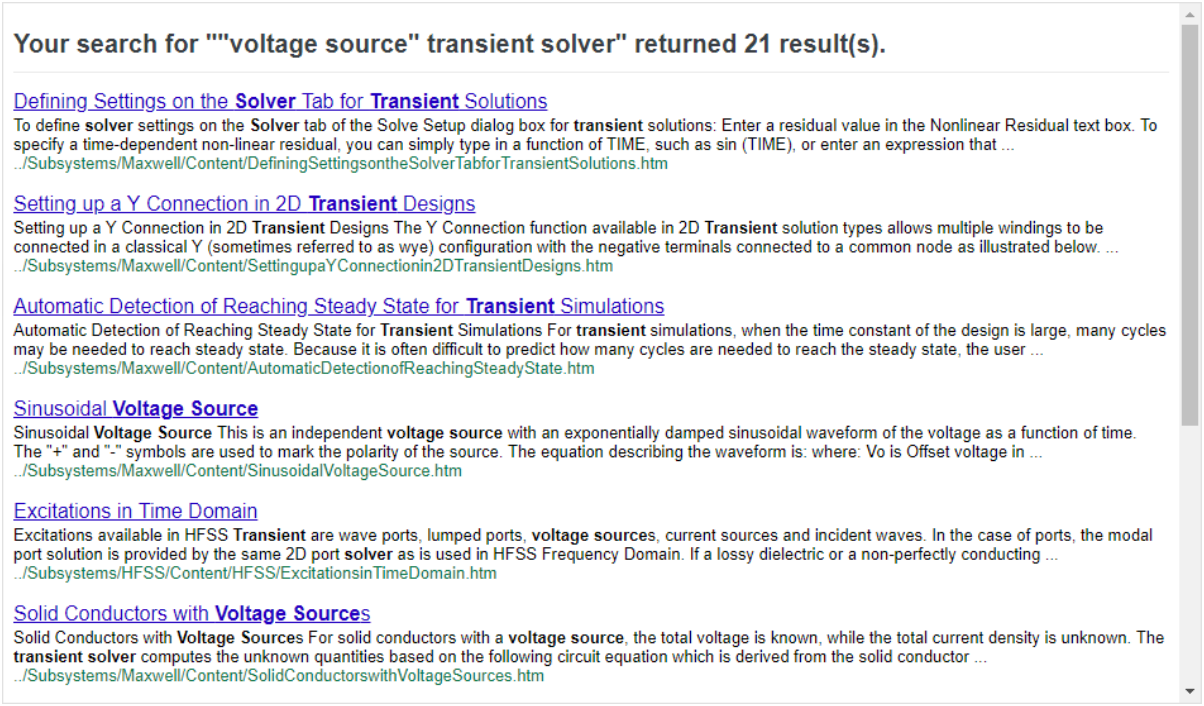
If you enter “voltage source” with quotation marks, the results show all topics that include the phrase.



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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

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

Using Boolean Operators

Note:

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Use parentheses to group terms and operators. For example:

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

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

Help Conventions

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

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

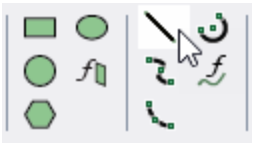
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

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

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

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

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

Getting Help: Ansys Technical Support

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Help Menu

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

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

Context-Sensitive Help

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

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

Getting Help from Ansys Technical Support

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

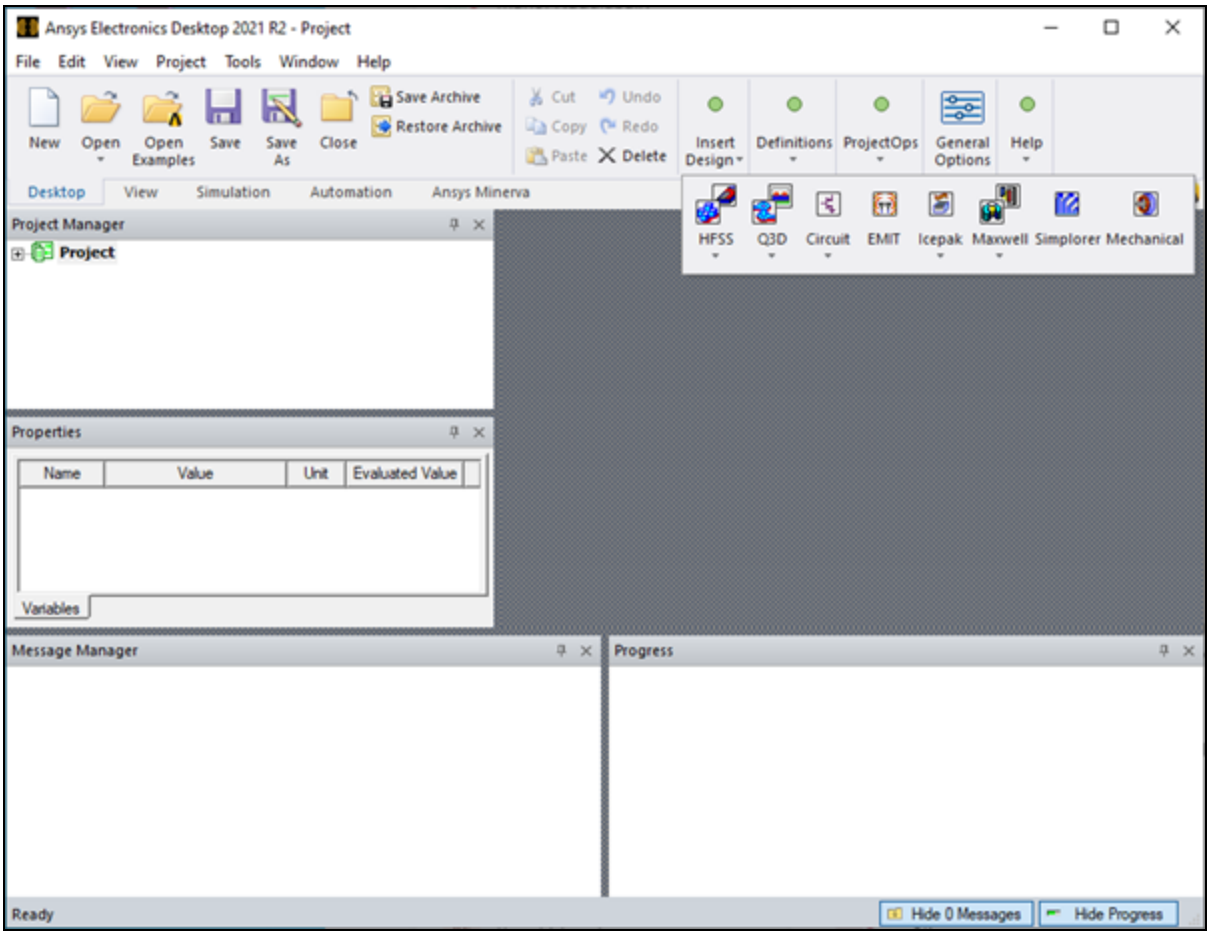
E-mail can work well for technical support. All Ansys EM software files are ASCII text and can be sent conveniently by email. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached. This allows more rapid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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 - Getting Started with Ansys Electronics Desktop

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



The desktop includes the following design types:

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

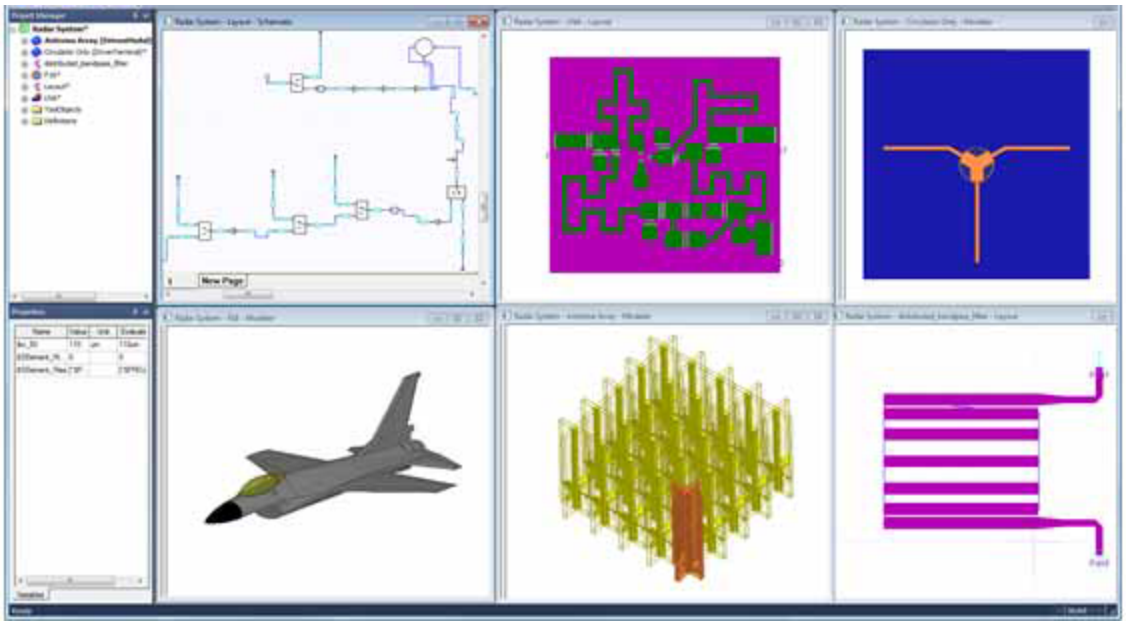
You can access all of these design types and features from the **Project** menu, and any combination of design types can be inserted into a single project file. The schematics can be used to wire up the different field solver models and create a model of a high-level system. Ansys Electronics Desktop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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



Ansys Electronics Desktop Student

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

HFSS Student Limitations

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

Q3D Extractor and 2D Extractor Student Limitations

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

Circuit Limitations

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

Maxwell and RMxprt Student Limitations

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Student Limitations

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

System Requirements

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

Ansys 2025 R1 **- Platform Support by Application / Product (PDF)**

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

Limitations of Linux Installations:

While the majority of the Ansys Electromagnetics Suite applications and features are supported on both Windows and Linux platforms, some are not supported on Linux. See the following topic for details:

Windows vs. Linux Installations

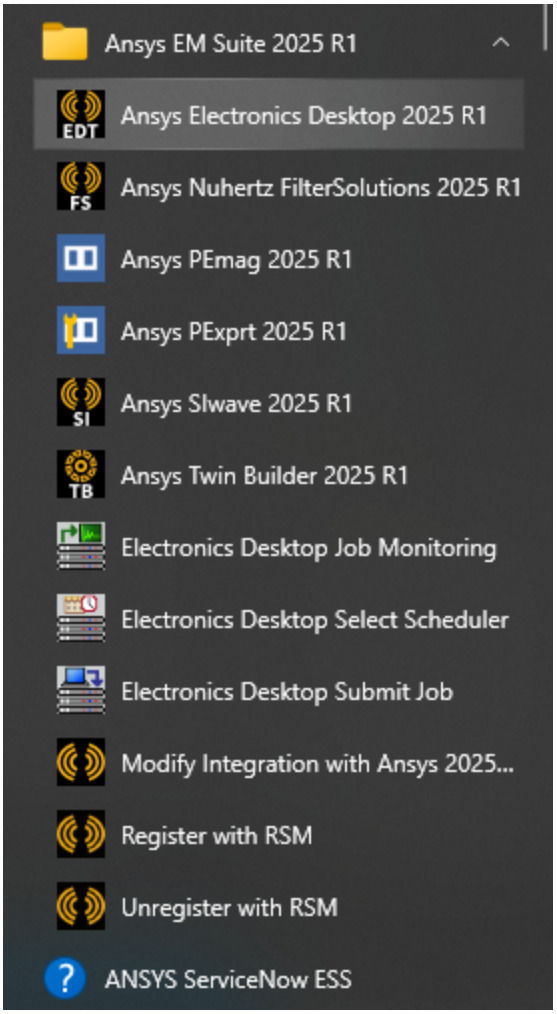
Launching Ansys Electronics Desktop

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

- On the Windows desktop, double-click the Ansys Electronics Desktop icon.

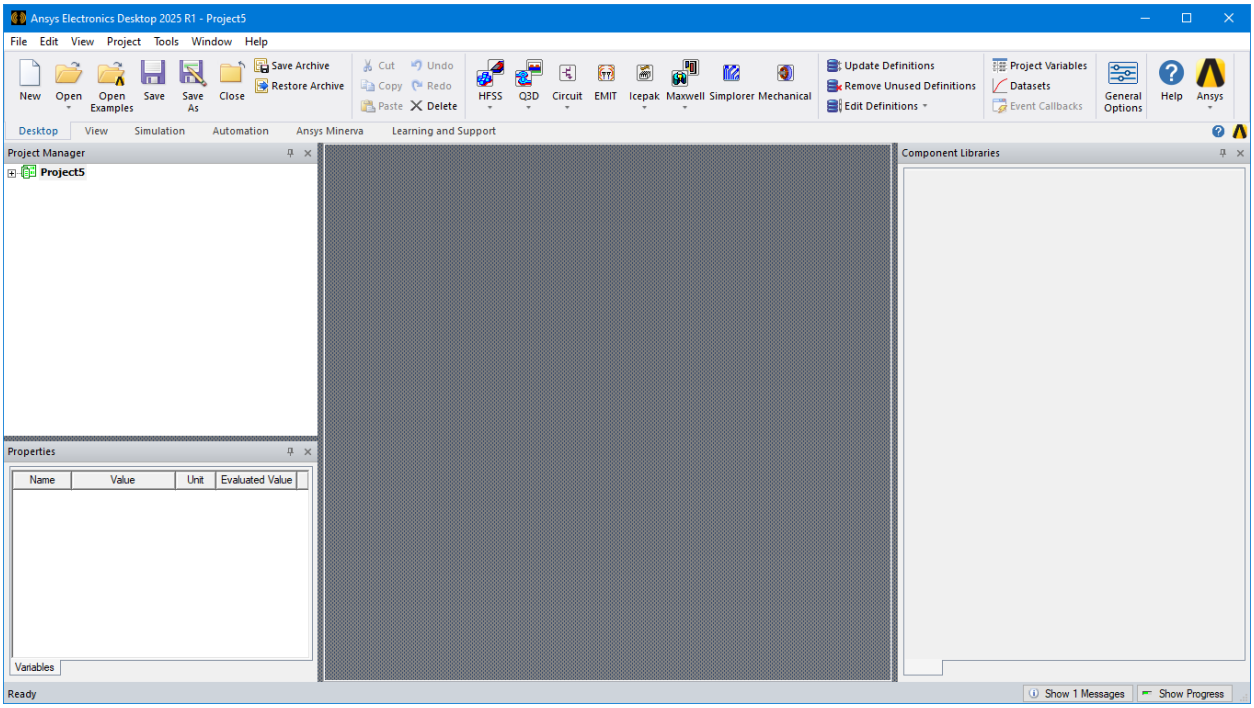
- From the Windows Start menu, select **Ansys EM Suite 2025 R1** > **Ansys Electronics Desktop 2025 R1**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electronics Desktop opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the program fails to start, ensure that:

- You have installed the licensing option provided to you. For detailed information on installing the software and licenses, see Ansys EM Suite Installation.

  This topic provides links to the Ansys installation guides (Windows and Linux) and product-specific platform support information.

- Ansys QA Services are not enabled. To disable, change the value of the ENABLE_ANSYS_QA_SERVICES environment variable to 0.

Obtaining Information about the Software and Release

To obtain information about the software and release:

1. Click **Help** > **About Ansys Electronics Desktop**
   The **About Ansys® Electromagnetics Suite [release number]** dialog box appears, listing information about the product.
2. Click the **Installed Components** tab to view a list of software installed.
3. Click the **Client License Settings** tab to view information about the following:
   - Provider Name
   - Ansys License Version

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- FlexNet Publisher Servers
- Admin Directory
- Customer Number
- FLEXlm Version
- Redirect Info

4. To export the software information:
   a. Click **Export**.
      The **Save As** dialog box appears.
   b. Browse to the location where you want to save the information as a text file.
   c. Type a name for the file in the **File name** text box. The **Save as type** drop-down menu is already specified as **Export (*.txt)**.
   d. Click **Save**.
5. Click **OK** to close the **About Ansys® Electromagnetics Suite [release number]** dialog box.

Ansys Product Improvement Program

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

How to Participate

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

How the Program Works

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

Data We Collect

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

For Ansys Electronics, we collect the following information:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Data We Do Not Collect

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

Opting Out of the Program

You may stop your participation in the program any time you wish. To do so, select menu item **Help > Ansys Product Improvement Program** or ribbon item **Help > Ansys > Ansys**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Product Improvement Program. This activates the **Ansys Product Improvement Program** dialog box. On this dialog select the **No I am not willing to participate** radio button to opt out of the program. Select the **Yes, I am willing to participate in the Ansys Product Improvement Program** radio button to opt in to the program. Click the **OK** button to accept the radio button selection. Data is no longer collected or sent if you opt out.

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

The ANSYS, Inc., Privacy Policy

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

Frequently Asked Questions

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

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

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

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

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

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

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

d. Data will no longer be collected or sent.

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

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

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

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

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

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

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

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

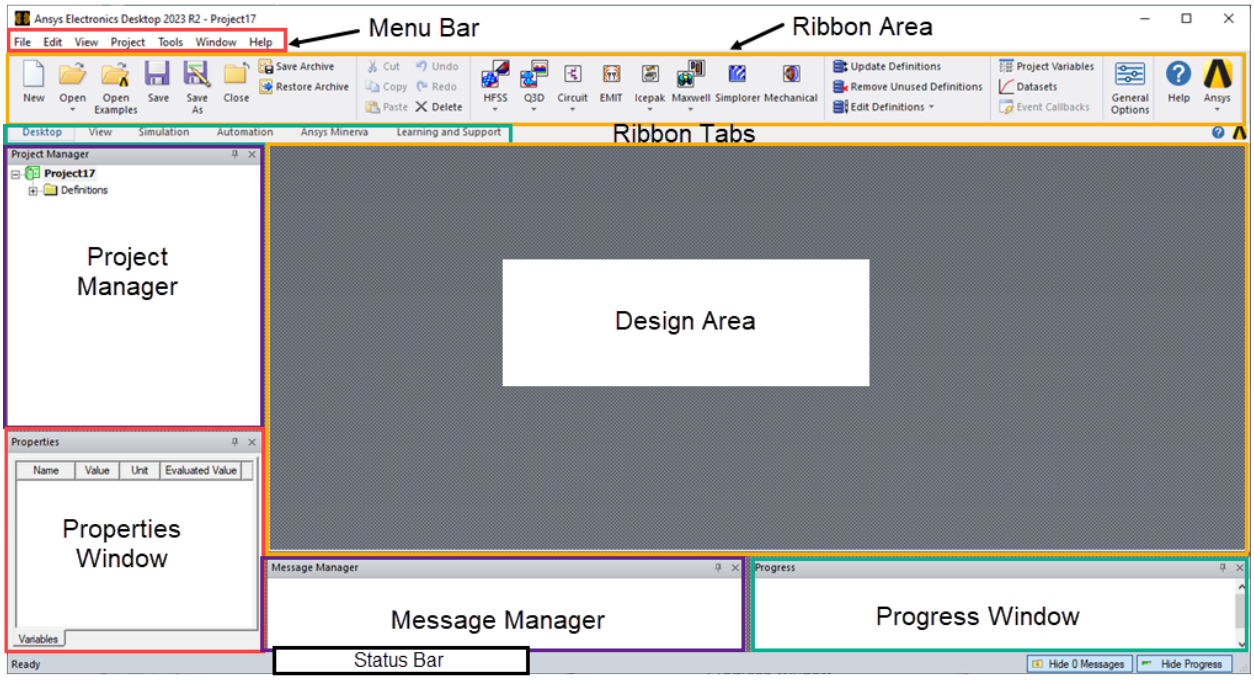
9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

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

Ansys Electronics Desktop Overview

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

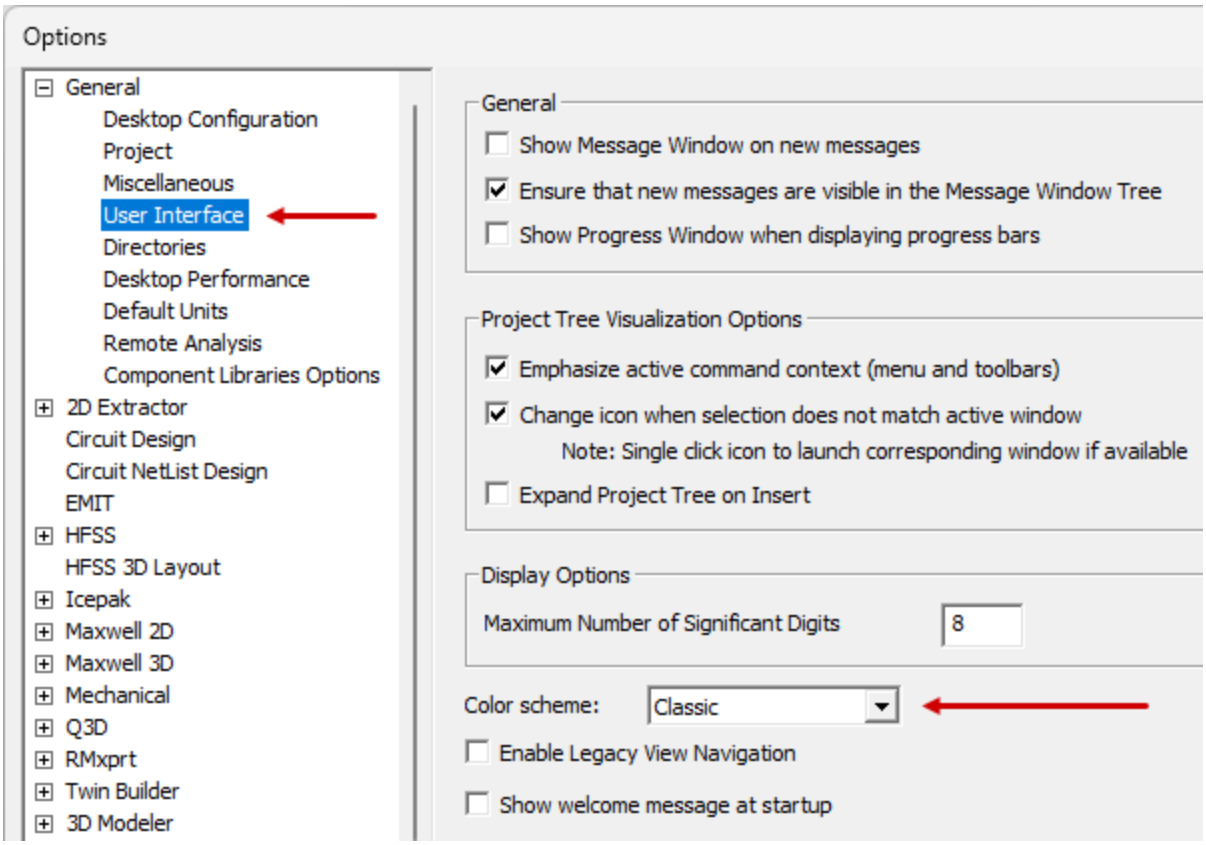


Choosing a Color Scheme [Beta]

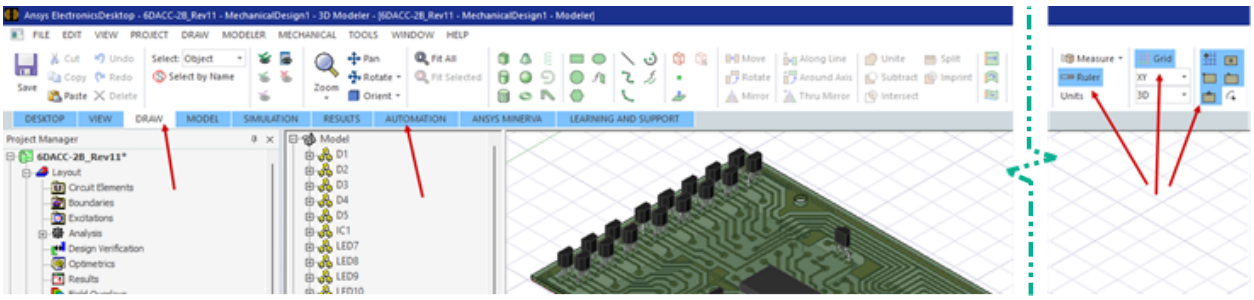
In addition to the classic Ansys Electronics Desktop color scheme, two additional color schemes are now available as beta features. Choose the desired color scheme as follows:

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

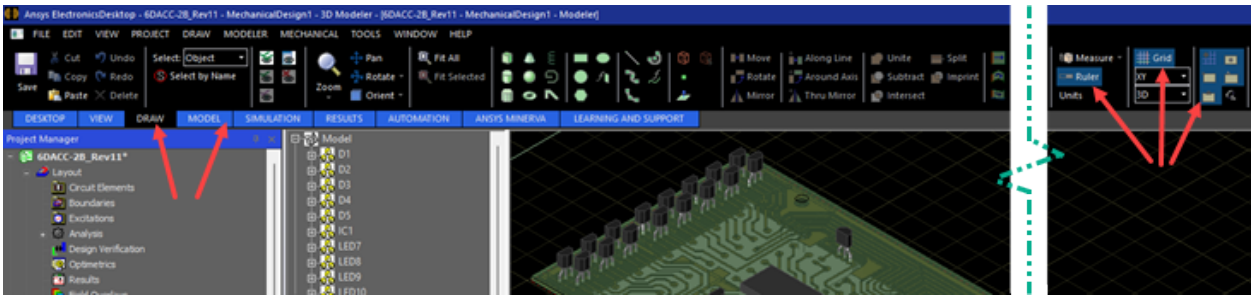


3. From the **Color scheme** drop-down menu, choose one of the following three options:
   - **Light (beta):** A light gray and white scheme with light blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a white background (including the tab):

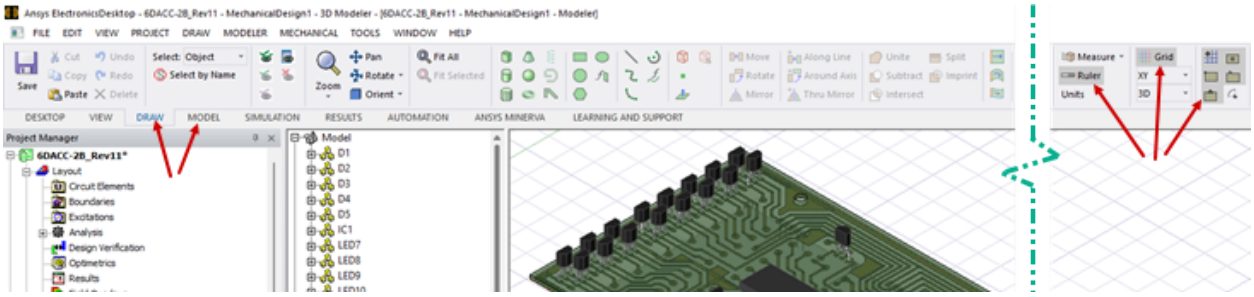


   - **Dark (beta):** A dark gray and black scheme with medium blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a black background (including the tab):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



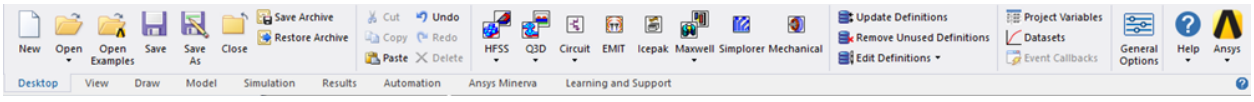
- **Classic:** A light gray and white scheme with medium gray highlighting for selected ribbon icons. The ribbon background is light gray. The name of the active ribbon tab has a blue font. A dark gray font is used for the inactive ribbon tab names:



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

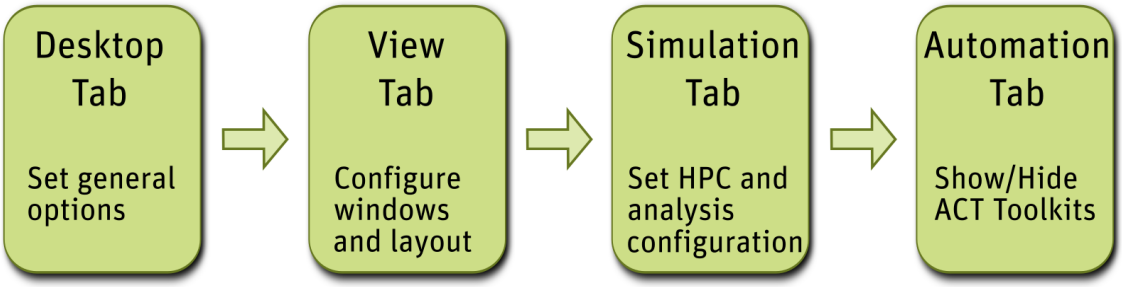
Working with Ribbons

The ribbon is the rectangular area across the top of the application. It comprises various tabs, each one representing a subset of commands available from the menus. The initial set of tabs (Desktop, View, Simulation, Automation) offers commands for adding and opening projects, selecting solvers, configuring the Desktop display (window choice, size, and position), configuring the simulation environment, setting scripted Event Callbacks or General Options, and showing Automation features for recording and using scripts.



The initial workflow for local configuration and personal customization is:

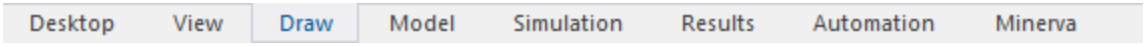
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



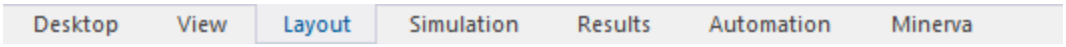
Available ribbon tabs depend on the design type. After you open or add a project and insert a design, you see additional ribbon tabs appear. The visible tabs and features are those that are appropriate for the design type and solver.

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

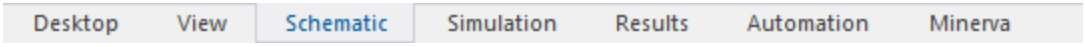
- For HFSS, Icepak, Maxwell, Mechanical, and Q3D designs, the **Draw**, **Model**, and **Results** tabs also appear:



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



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

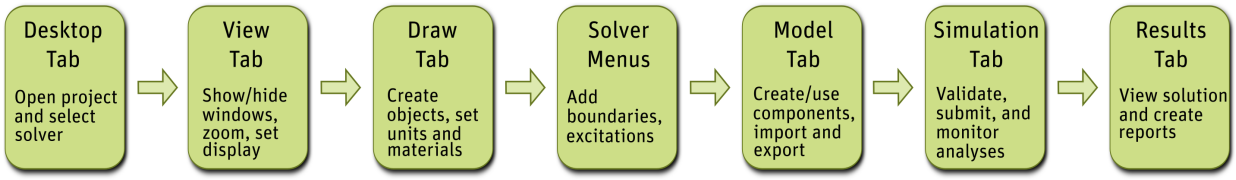


In addition to **Desktop**, **View**, **Minerva** and **Learning and Support** the ribbon tabs that are applicable to Mechanical designs are described as follows:

- **Draw Tab** – contains tools that allow you to draw objects, set units, and assign materials.
- **Model Tab** – contains tools to group and ungroup objects, create 3D components, and import or export geometry.
- **Simulation Tab** – contains tools that allow you to run simulations, perform validation checks, and schedule and monitor tasks.
- **Results Tab** – contains tools for creating reports and viewing solution data.
- **Automation Tab** – contains tools that allow you to run and record scripts, show or hide ACT extensions (Windows only), and install PyAEDT (Beta).

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

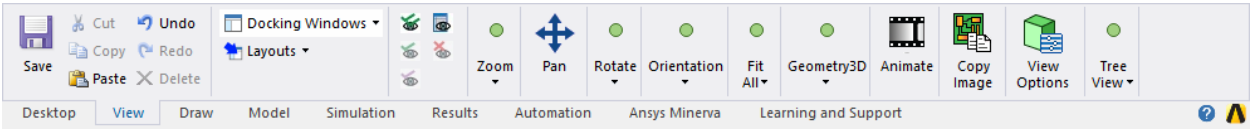


Note that workflow can vary depending on tasks at hand. For example, if you open a completed model in HFSS or use 3D components or the ACT Antenna Design toolkit (Windows only), you may not need the features of the **Draw** tab. If Validation on the **Simulation** tab identifies problems in a model,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

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

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

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



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

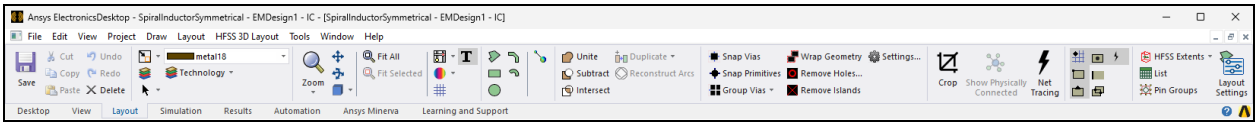
The **Learning and Support** tab provides easy access to Technical advice, instruction, and examples on Ansys websites.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ansys Innovation Courses** – opens a web page containing a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.
- **Ansys Learning Hub** – opens a web page with subscription based access to, virtual and self-paced learning across the Ansys Software portfolio.
- **Ansys Knowledge** – opens a web page to expert curated knowledge materials from FAQs to tutorials on simulation topics.
- **Ansys Learning Forum** – opens a web page to Ansys blog containing discussion and presentations from Ansys experts, partners and customers.
- **Customer Portal** – opens a web page to Ansys Product support.
- **About Electronics Desktop** - opens a dialog with version and release information, Ansys Electronics Desktop installed components, and licensing information.

IC Mode Layout Ribbon

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



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

Ansys Electronics Desktop Windows

Ansys Electronics Desktop contains the following windows:

- Project Manager
- Message Manager
- Progress

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Properties
- Component Libraries

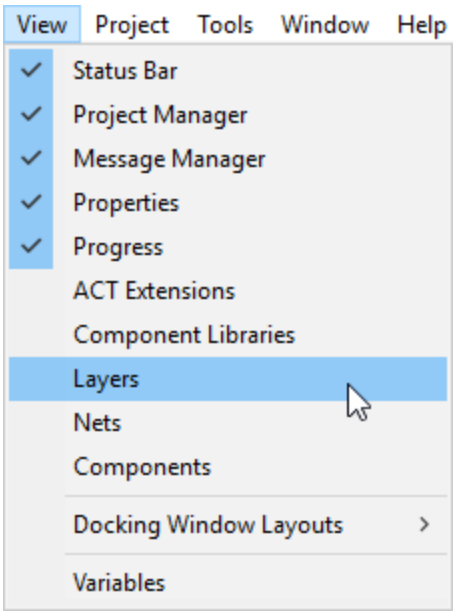
You can toggle the display of each window, auto-hide windows, or move and resize windows. Additionally, you can toggle the status bar or add ACT Extensions (Windows only).

Showing and Hiding Windows

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

From the **View** menu:

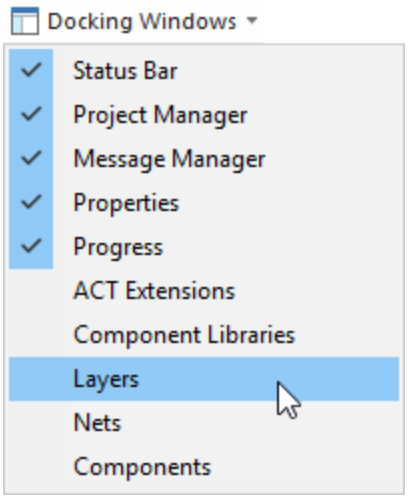
- Click a window to show or hide it.



From the **View** tab:

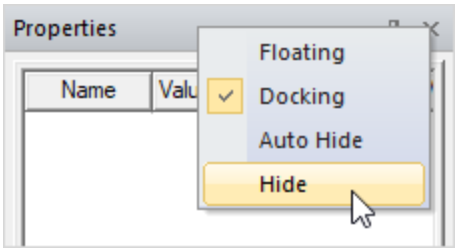
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



You can also hide a window directly from the window:

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



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

Auto Hiding Windows

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

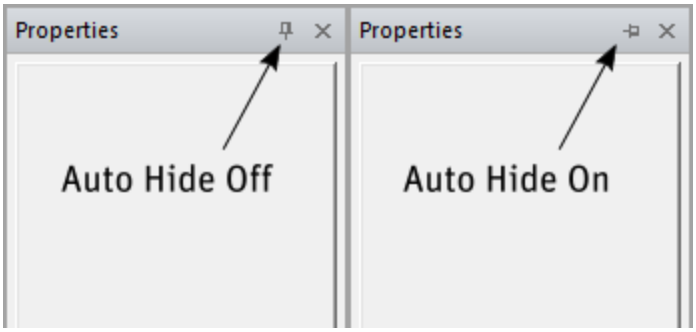
To automatically hide a window until the cursor is hovered over it:

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

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

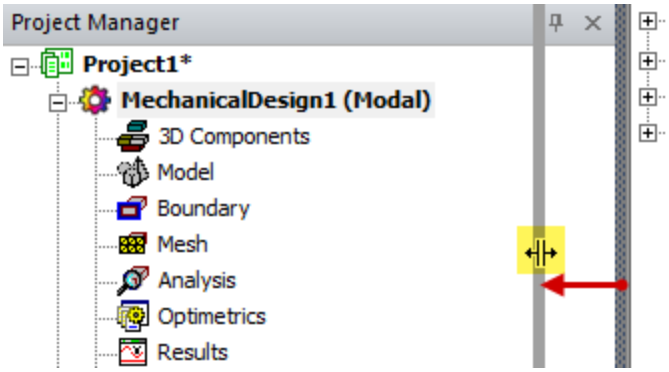
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

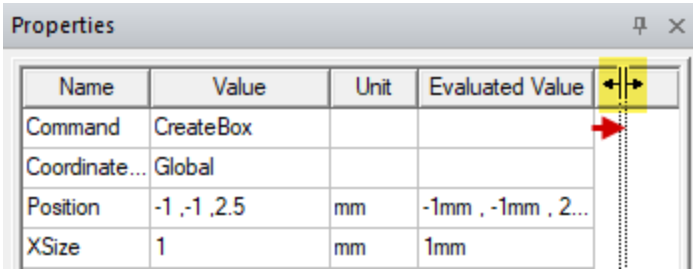


Moving and Resizing Windows

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



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



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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

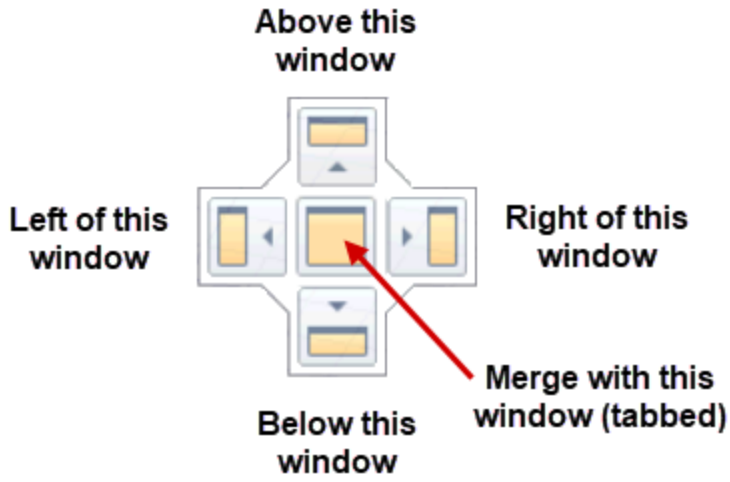
Left edge of user interface:

Right edge of user interface:

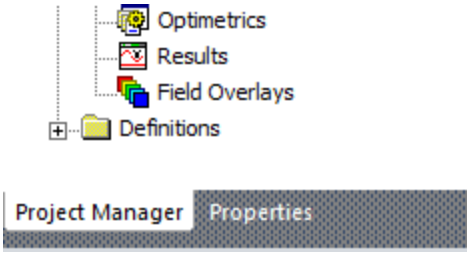
Top of user interface:

Bottom of user interface:

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



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



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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Project Manager Window

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

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

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

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

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

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

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

Working with the Project Tree

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

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

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

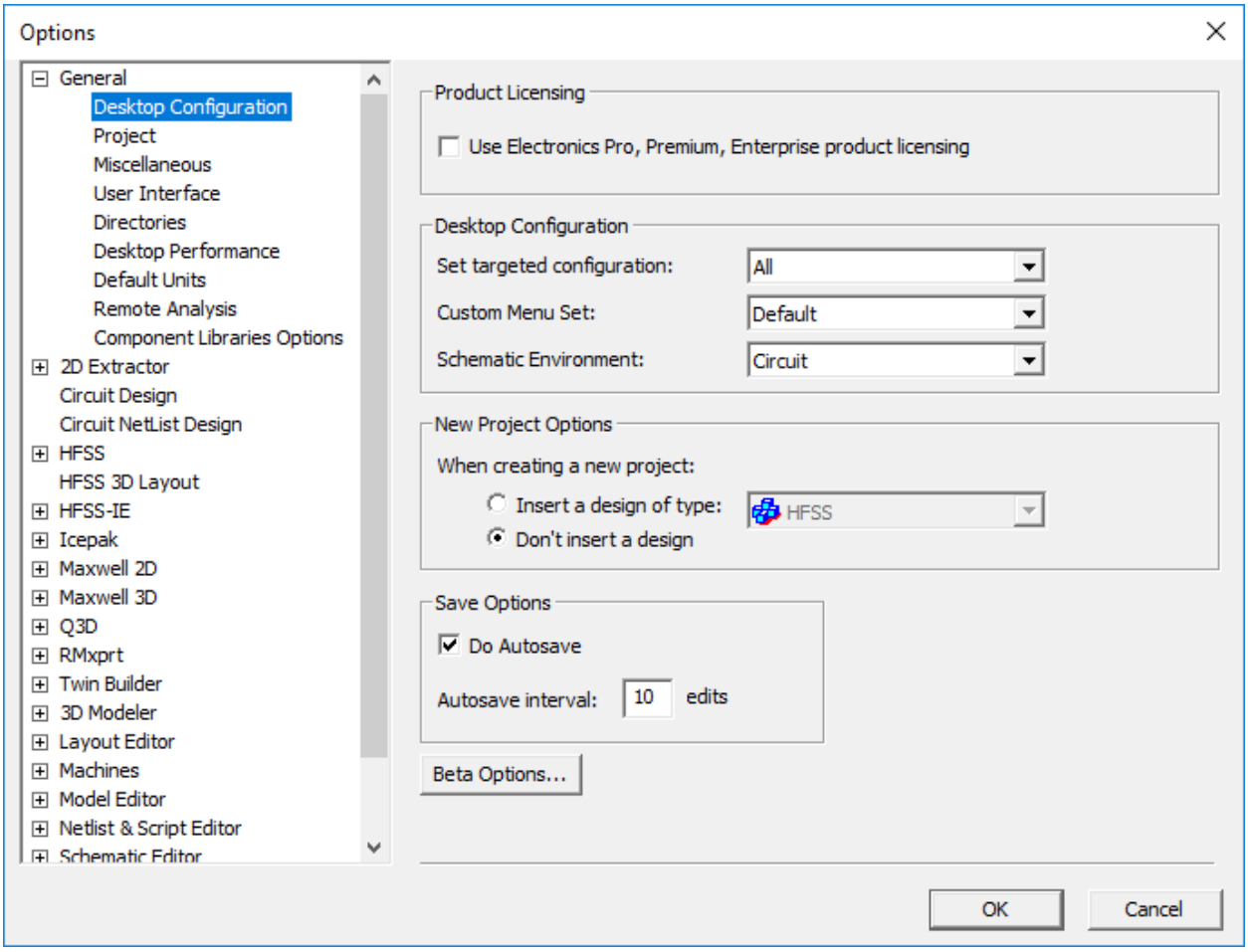
Setting the Project Tree to Expand Automatically

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

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

   The **Options** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

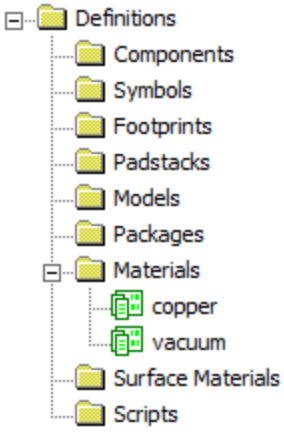


2. Click **User Interface**.
3. In the **Project Tree Visualization Options** area, check the **Expand Project Tree on Insert** check box.
4. Click **OK**.

Viewing Material Definitions

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Ansys Electronics Desktop Design Details

When you insert an Ansys Electronics Desktop design into a project, it is listed as the second node in the project tree. It is named Ansys Electronics DesktopModel*n* by default, where *n* represents the order in which the design was added to the project. Expand the design icon in the project tree to view all of the specific data about the model, including its boundary conditions and material assignments, and field solution and post-processing information.

The Ansys Electronics DesktopModel*n* node contains the following project details:

3D Components	Displays any 3D Components added to the design.
Model	Displays the model geometries in the design.
Contacts	For Mechanical–Thermal solutions only, displays user-defined thermal contact assigned between selected adjacent objects or faces.
Boundaries	Displays the boundaries assigned to a Mechanical design (for Modal, Steady-State Thermal, Transient Thermal, or Structural solutions). Available boundaries vary depending on the solution type.
Excitations	Displays the excitations assigned to a Mechanical design (for Steady-State Thermal, Transient Thermal, or Structural solutions). Available excitations vary depending on the solution type.
Analysis	Displays the solution setups for an Ansys Electronics Desktop design. A solution setup specifies how Ansys Electronics Desktop will compute the solution. Setups for Mechanical analyses vary by solution type: • Solution Setup (Modal) • Solution Setup (Thermal) • Solution Setup (Structural)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimetrics	Displays any Optimetrics setups added to an Ansys Electronics Desktop design.
Results	Displays any post-processing reports that have been generated. Available results for Mechanical analyses vary by solution type: • Results (Modal) • Results (Thermal) • Results (Structural)
Field Overlays	Displays fields, which are representations of basic or derived field quantities on surfaces or objects. Plot folders are listed under **Field Overlays.** These folders store the project's plots and can be customized. See Setting Field Plot Defaults for information on how to customize the plot folders.
Documentation	Displays files you have added as documentation.
Definitions	Displays definitions for Mechanical, including Components, Materials, and more (as applicable).

Note:

To edit a project's design details:

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

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

Message Manager Window

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

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

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

Messages in the **Message Manager** window are organized first by project, then by circuit. Because a design can contain multiple circuits and subcircuits, sometimes with multiple analyses for each, this organization helps you to quickly determine where errors have occurred.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following icons appear next to a message to indicate information, warnings, errors, or actions:

Indicates an informative message.

Indicates a warning message that may require your attention.

Indicates an error message that may require your attention.

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

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

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

Setting the Message Manager to Open Automatically

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

Showing New Messages

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

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

   The **Options** window appears.

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatically Expanding the Message Manager Tree

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

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

   The **Options** window appears.

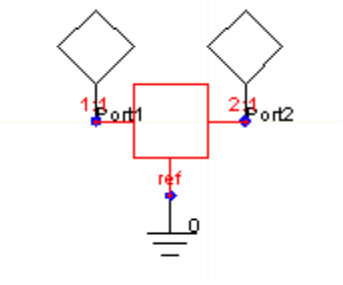
2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Ensure that new messages are visible in the Message Window Tree** check box.
4. Click **OK**.

Action Messages

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

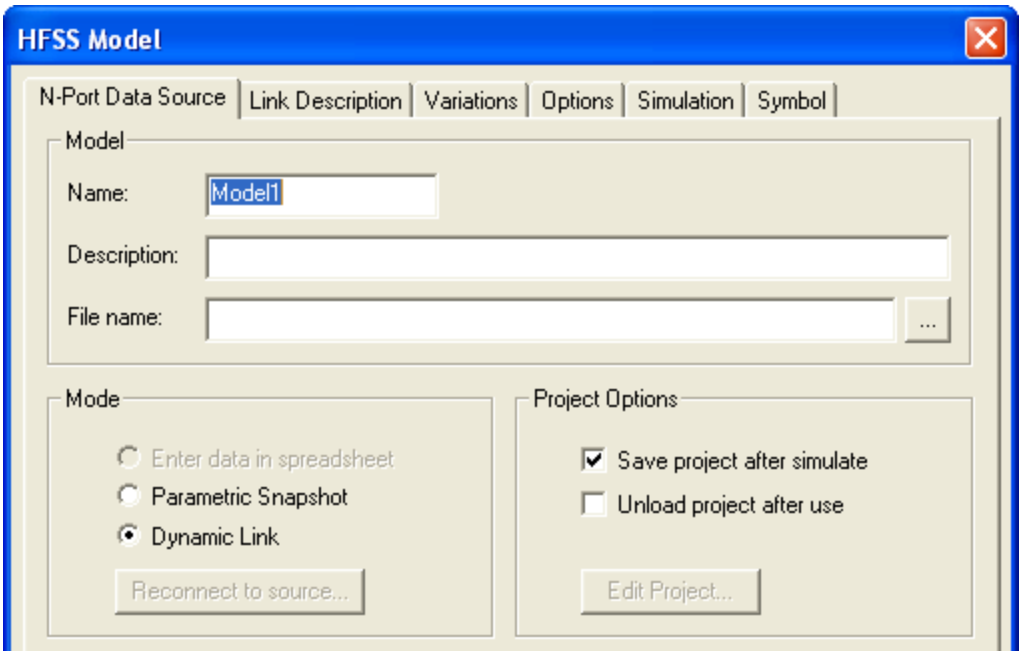
You can invoke a message action by clicking the magnifying glass icon, by double-clicking the message, or by right-clicking the message and selecting **Go To Reference** from the pop-up menu.

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



Another example is the opening of a related window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clearing Messages

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

Hiding the Message Manager Window until Messages Appear

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

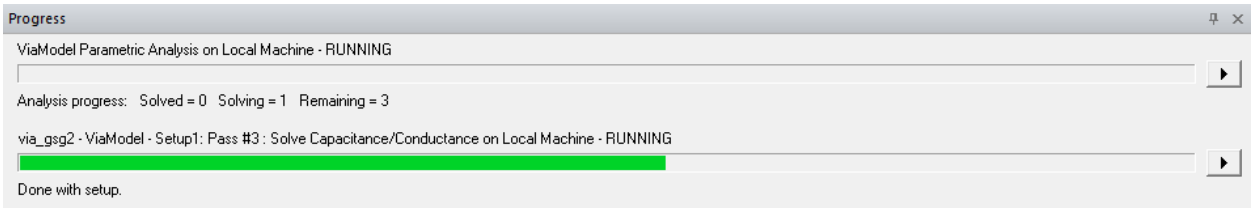
1. Select **View** > **Progress Window** to hide the window. For more information see the View drop-down menu.
2. Click **Tools** > **Options** > **General Options**.
3. Check the **Show message window on new messages** check box.

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

Progress Window

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

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

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

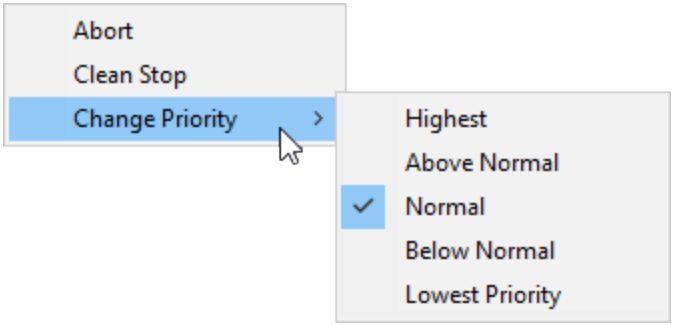
A check box appears next to this command if the **Progress** window is visible.

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

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

Stopping or Aborting Simulation Progress

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

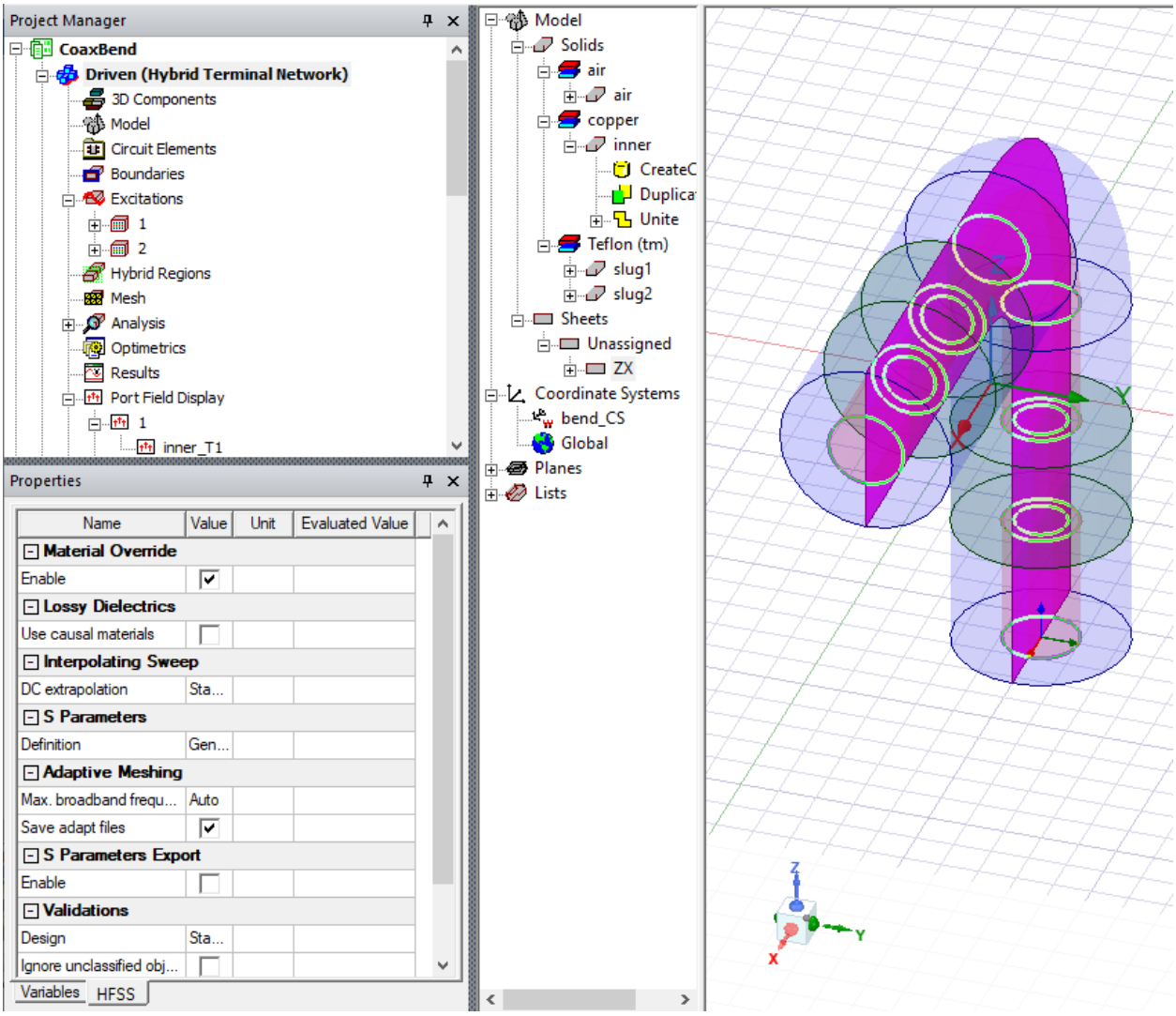


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties Window

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

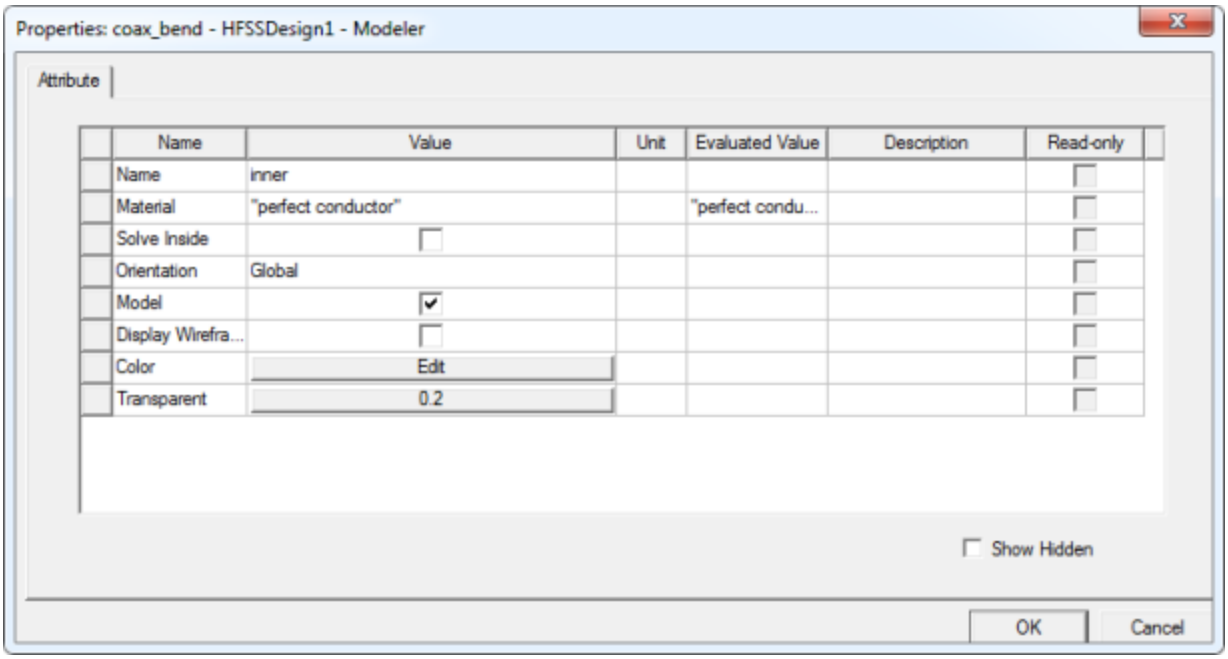
You can choose to show or hide a docked **Properties** window as part of the desktop.



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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



Opening the Properties Window

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

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

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

Setting the Properties Window to Open Automatically

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Showing and Hiding the Properties Window

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

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

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

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

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

Modifying Object Attributes Using the Properties Window

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

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

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

Modifying Object Command Properties Using the Properties Window

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

Not all command properties can be modified. In general, command properties you can typically modify are numeric values, such as position values, size values, and various other coordinate values. You can also modify many of the unit settings for a command property. You can modify names, but names must include only letters, numbers and underscores. Illegal names are not accepted and generate a message in the Message Manager window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

   Tip:

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

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

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

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

Ansys Electronics Desktop Menus

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Top Menu Bar

The **Top Menu** bar contains menus for controlling Ansys Electronics Desktop and the various editors and viewers. Click on a **Top Menu** bar entry to open its menu. In addition, the **Alt** key makes it easy to access the **Top Menu** bar menus.

- A menu in the **Top Menu** bar can be opened by clicking the **Alt** key and then pressing the underlined letter of the menu you wish to activate (the underlines appear when the **Alt** key is pressed).
- Clicking the **Alt** key enables you to scroll across the **Top Menu** bar, opening the menus, by pressing the left (⇐) and right (⇒) arrow keys.

Once a menu is open, you can use the down (⇓) and up (⇑) arrows to change the selection. You can use the right (⇒) arrow key to open a subordinate menu for the selected command. Operations on the menus can be executed in three ways:

- Clicking on the operation.
- Typing the underlined letter as shown in the menu (for example, typing the “n” in **New** on the **File** menu). Type all underlined letters in lower case.
- Using a shortcut key combination as shown in the menu (for example, Ctrl+N for the New command on the **File** menu. Shortcut key combinations are valid whether or not the menu is visible. See Shortcut Keys to locate topics with listings of various types of shortcuts.

Shortcut Menu in the Toolbars Area

Use the shortcut menu in the toolbars area of the desktop to toggle the show/hide settings for various desktop windows and toolbars. To access the shortcut menu, right-click in the toolbars area at the top of the desktop.

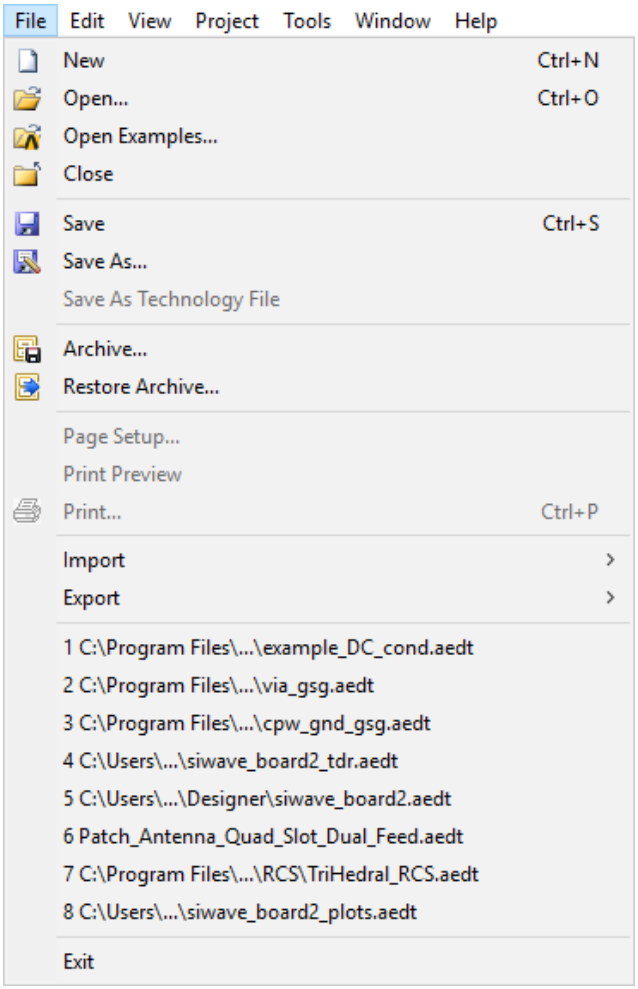
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

File Edit View Project Draw Design-Editor Tools Window Help

These menus are described in the following sections.

File Menu



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

- Click **New** (or type **n**) to set up a new project.
- Click **Open** (or type **o**) to open an existing project.
- Click **Open Examples** to open an example project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Edit Menu

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

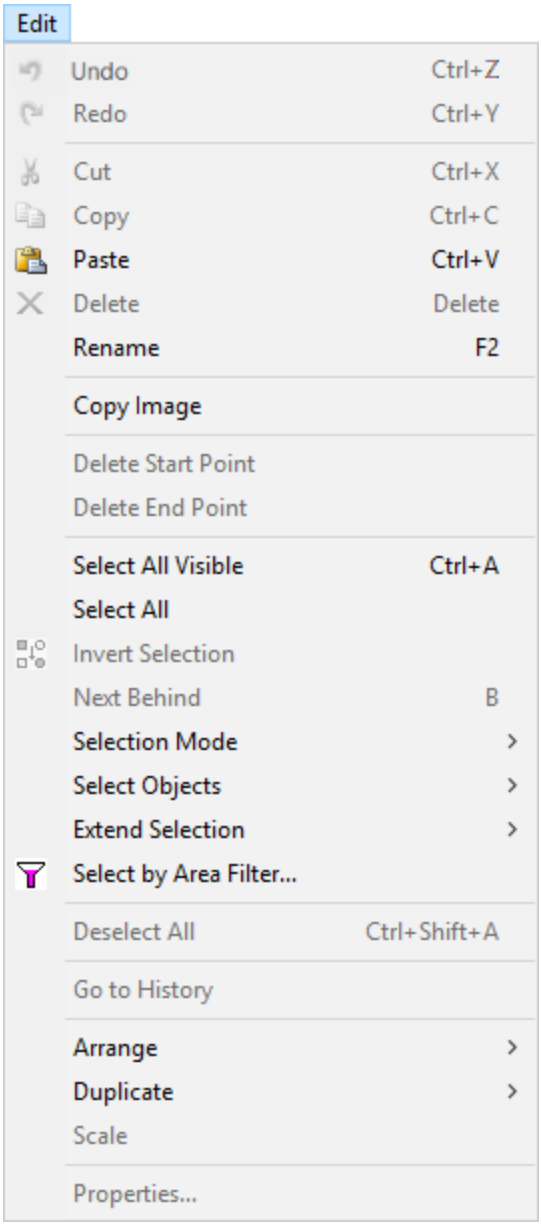
These menus include:

- 3D Modeler
- Report Window

Edit Menu for 3D Modeler

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

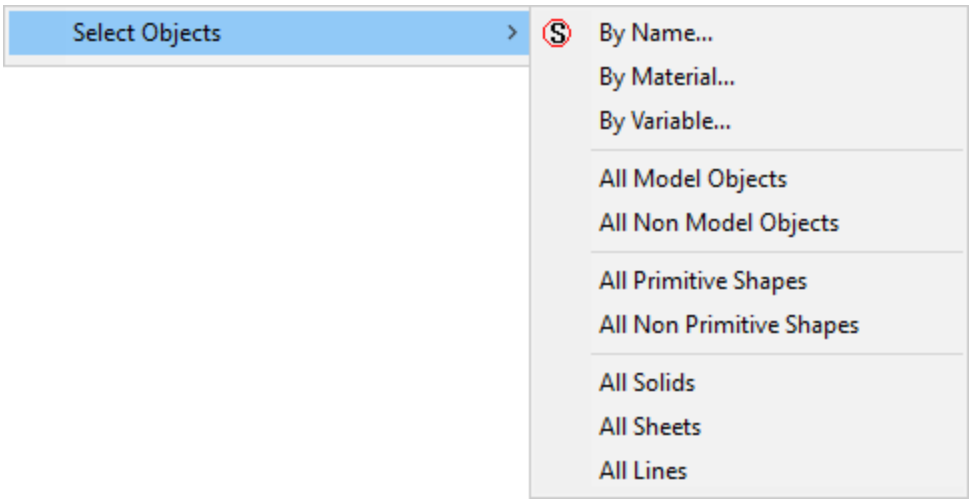


Comands will be available or not available for clicking depending on what is currently selected or on whether any operations have been performed, undone, or redone. The 3D Modeler's Edit menu contains the following options:

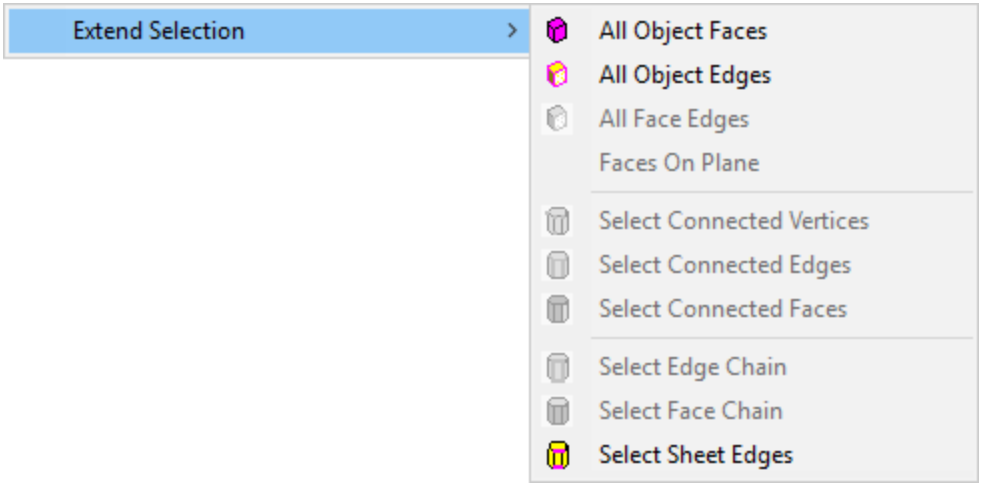
- **Undo** – allows you to undo the last action, which is specified after the word "Undo."
- **Redo** – allows you to redo an undone action, which is specified after the word "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste elements.
- **Delete** – deletes a selected element.
- **Rename**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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



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



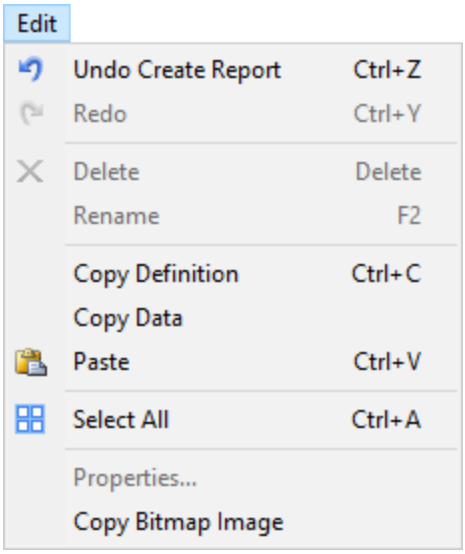
- **Select by Area Filter** – opens the **Select By Area Filters** window.
- **Deselect All** – deselects all selected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Edit Menu for Report Window

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



The Report Window Edit menu contains the following options:

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Menu

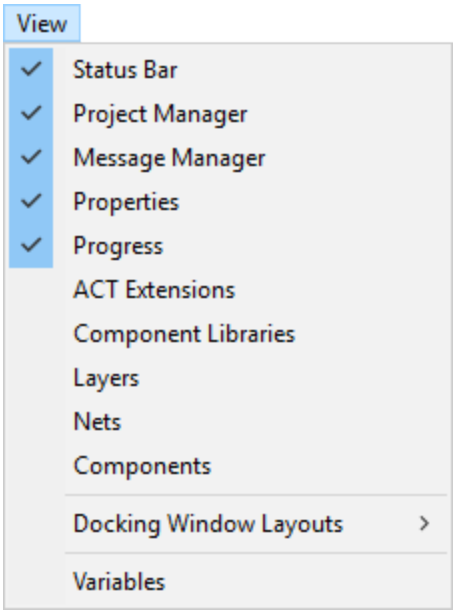
The **View** menu changes, depending on the active window in the Design Area.

These menus include:

- Basic View Menu
- 3D Modeler's View Menu

Basic View Menu

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

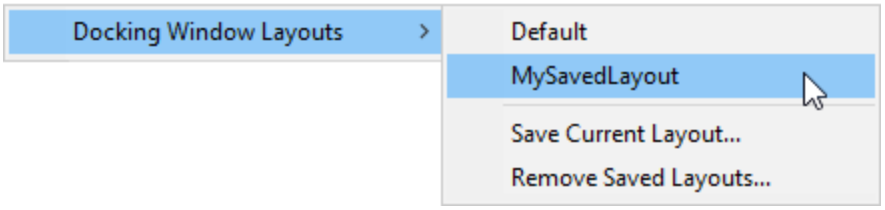


- The check boxes toggle the display of the **Status Bar**, **Message Manager Window**, **Project Manager**, **Properties Window**, and **Progress Window**. You can enable the display of **Component Libraries**, **Layers**, **Nets**, **Components** and **Variables** windows.
- The **ACT Extensions**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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

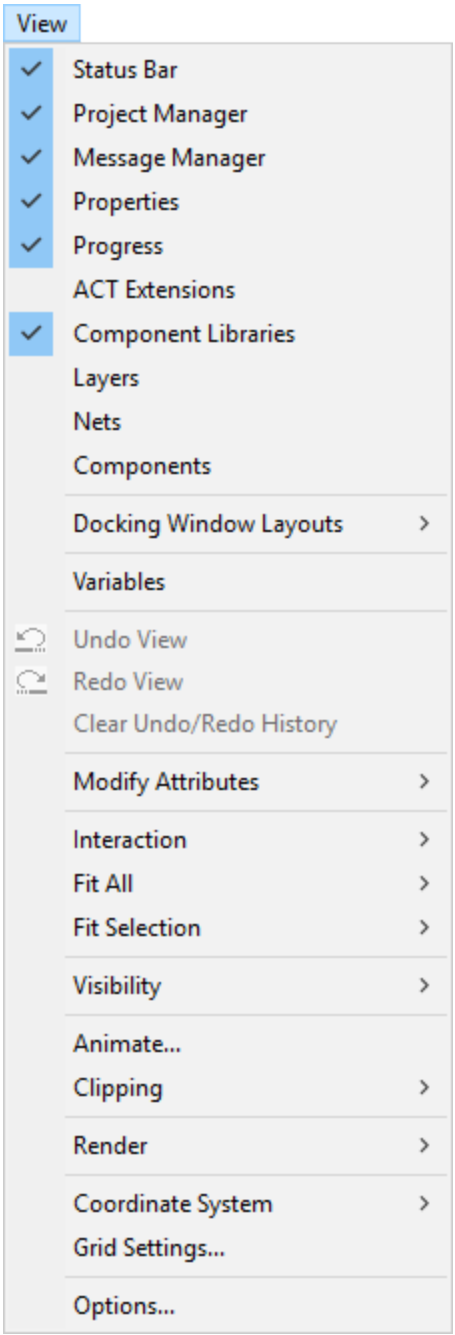


- The **Variables** option opens the **Project** and **Design** **Variables** window, allowing you to set variables.

3D Modeler View Menu

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

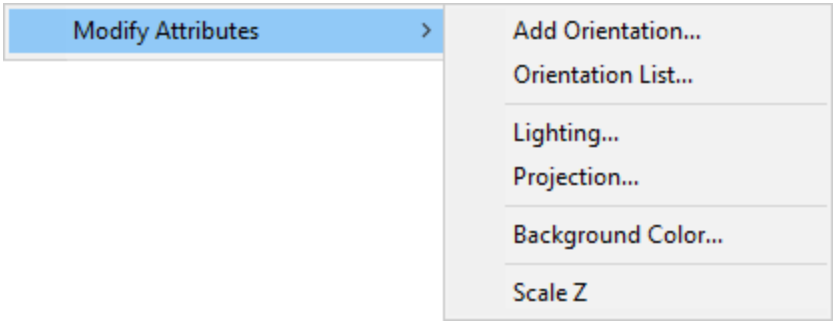


Thi View menu includes the options available on the Basic View Menu. The following additional options are available when a *Modeler* window is active. Note that a number of these options also appear in the View branch of the shortcut menu that appears when you right-click in the Modeler window.

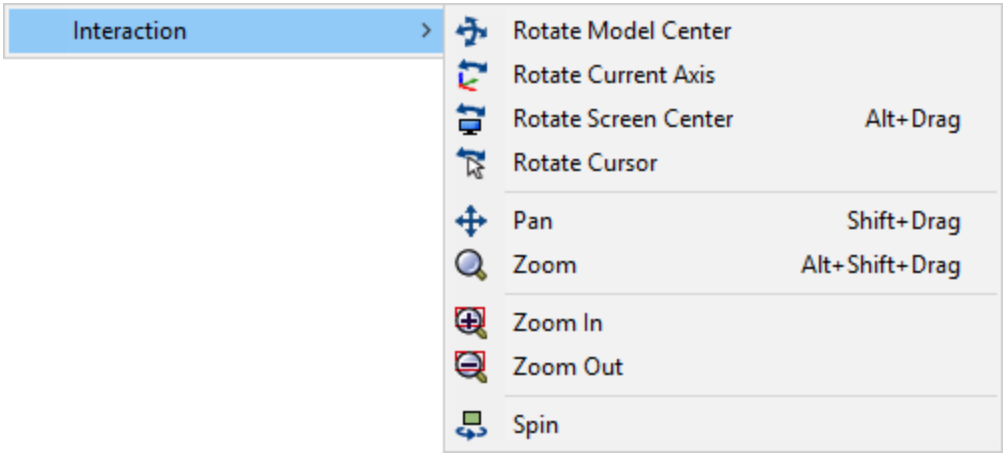
- **Undo View** / **Redo View** / **Clear Undo/Redo History** – allow you to change the view in the design area, based on view history.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Modify Attributes** – offers options to **Add Orientation**; view the **Orientation List**; change **Lighting**, **Projection**, or **Background Color**; and **Scale Z**.



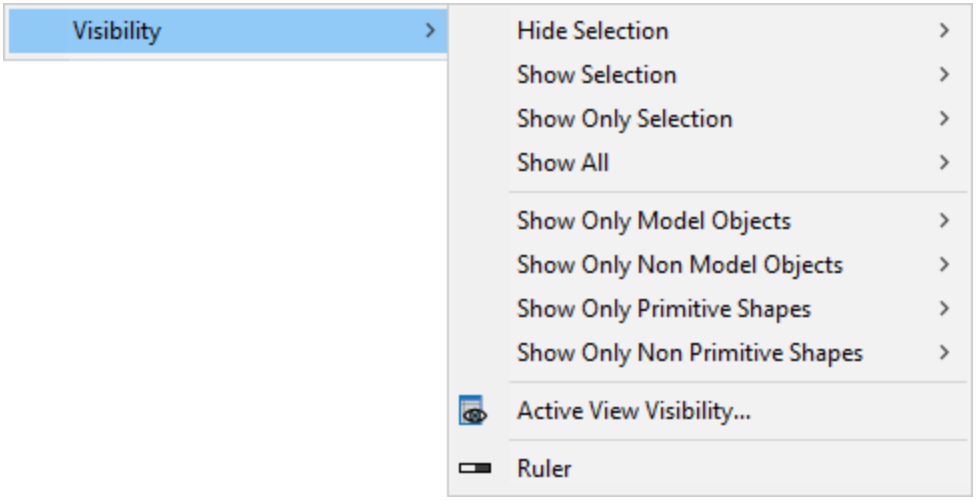
- **Interaction** – offers options to rotate, pan, zoom, and spin the drawing in the design area.



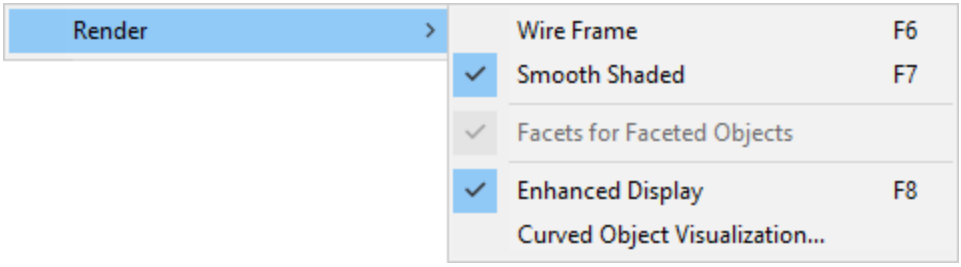
- **Fit All** – allows you to fit all objects to the view window.
- **Fit Selection** – allows you to fit selected objects to the view window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

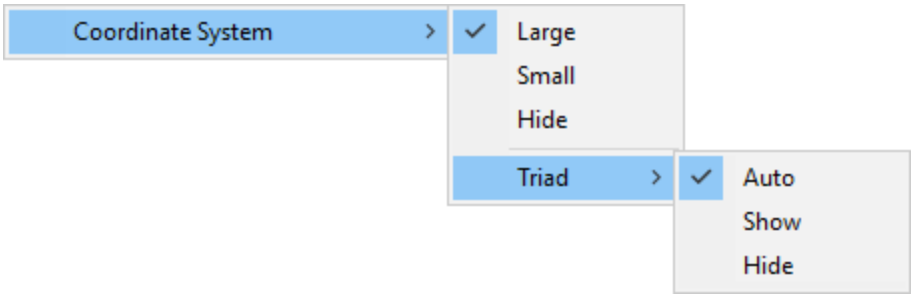


- **Animate** – opens the **Create Animation Setup** window, so that you can begin creating animations.
- **Clipping** – allows you to place a 3D coordinate system manipulator in the design area so that you can define clip planes.
- **Render** – offers options to switch between wire frame and smooth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

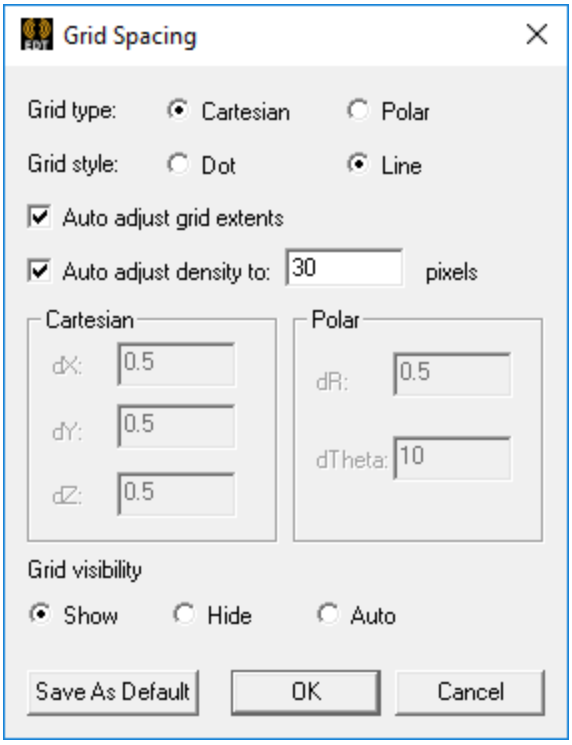


- **Coordinate System** – offers options to change how the coordinate system displays in the design area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

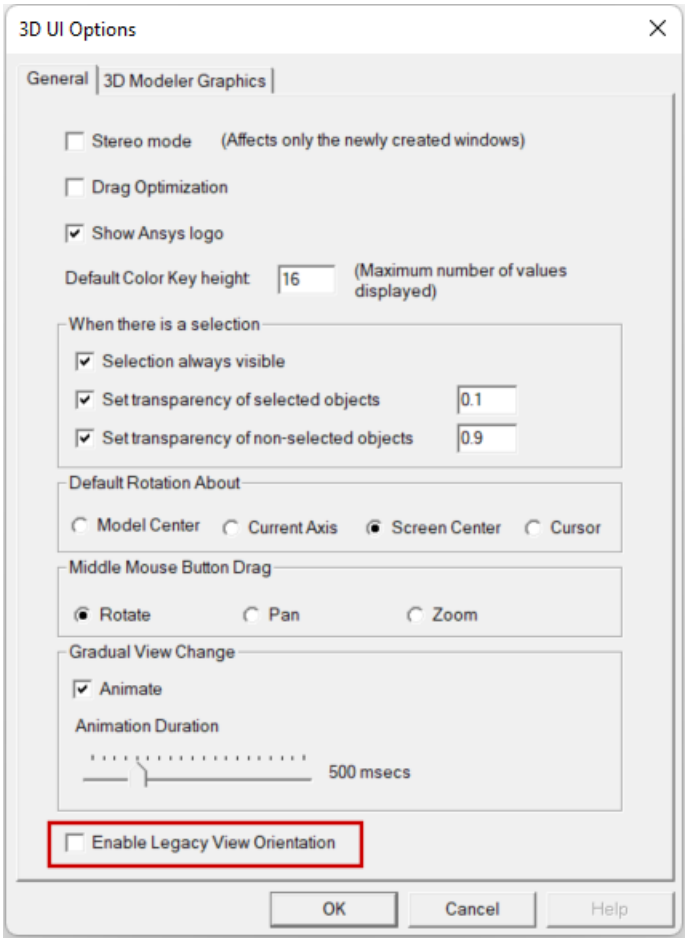


- **Grid Settings** – opens the **Grid Spacing** window, where you can adjust spacing.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

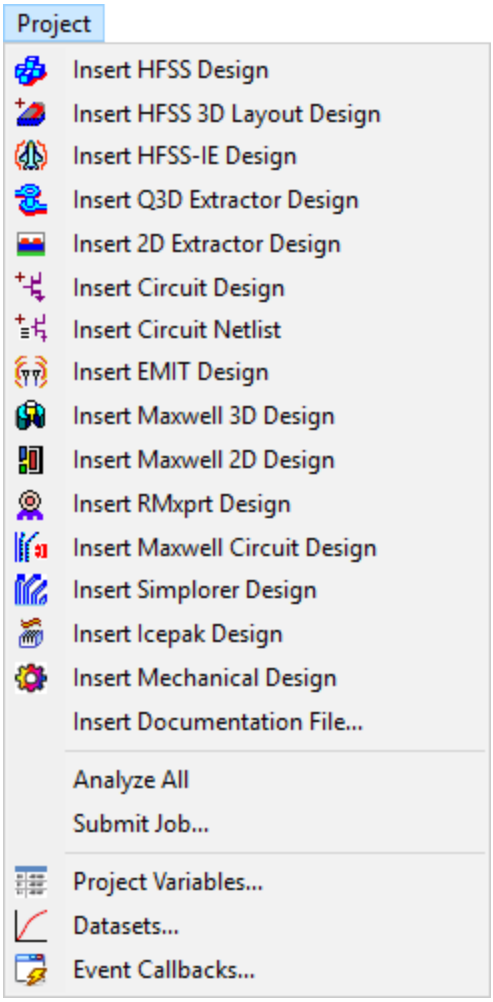
- **Options** – opens the **3D UI Options** window, where you can change additional settings.



Project Menu

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



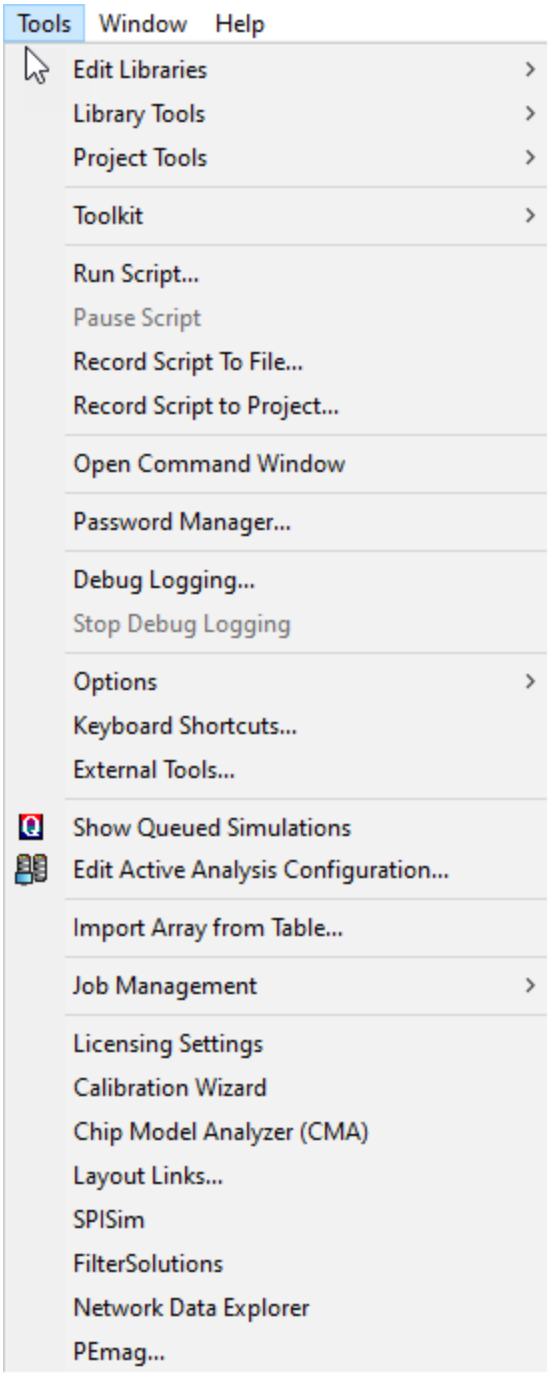
Editor and Design Specific Menus

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

Tools Menu

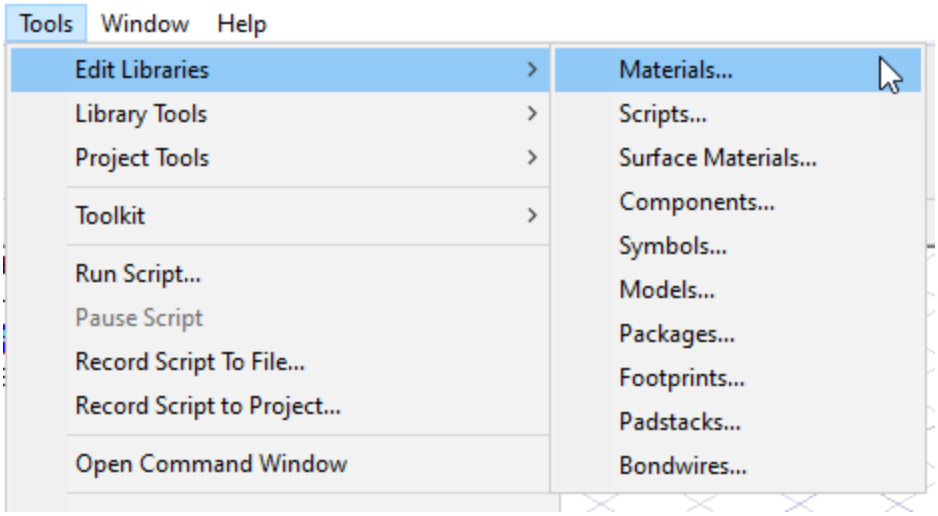
The **Tools** menu contains operations that are common to the analysis tools.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

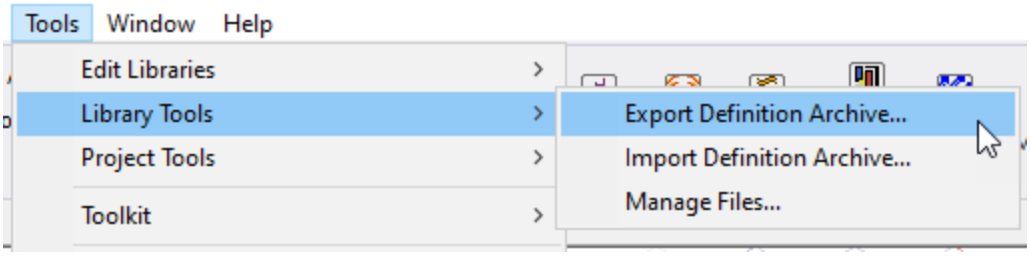


- Click **Tools>Edit Libraries** to view menus allowing access to the libraries available for the various Ansys Electronic Desktop solves, such as Materials, Scripts, and Components, Symbols and Models for Simplorer, Icepak, and Circuit.

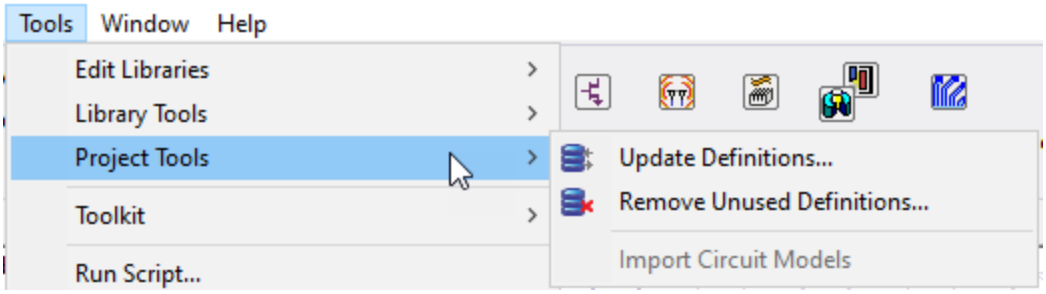
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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



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



- Click **Tools>Toolkit** to install or update, or access Python and Python-based toolkits.
- Click **Tools> Runscript...** to run scripts that you have recorded. A Pause command is available. You can also **Record Script to File...** or **Record Script to Project...**. See the Scripting Help for more information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Tools> Command Window...** to open an IronPython command window. See the Scripting Help for more information.
- Click **Tools>Debug Logging** to enable debug logging via an environment variable. When enabled, the **Debug Logging** is disabled and **Stop Debug Logging** is enabled.
- Click **Tools> Options** to set General Options and HPC options. You can also Export Optionsfiles.
- Click **Tools >Keboard Shortcuts..**. to manage your keyboard shortcut behavior.
- Click **Tools> External Tools...** to manage menu access to external tools.
- Click **Tools> Show Queued Simulations** to view a dialog listing simulations in a queue and providing tools to manage the queue.
- Click **Tools>Edit Active Analysis Configuration** to view a dialog that lets you view and manage the current configuration.
- Click **Tools>Import Array from Table** to import an array definition from a .csv file.
- Click **Tools>Job Management...** to select a scheduler, and to submit and monitor remote simulations.
- Click **Tools>License Settings** to launch the Licensing Settings tool.
- Additional Tools listed include Calibration Wizard, and Network Analyzer, and other tools used to help you with Ansys Electronics Desktop projects and solvers.

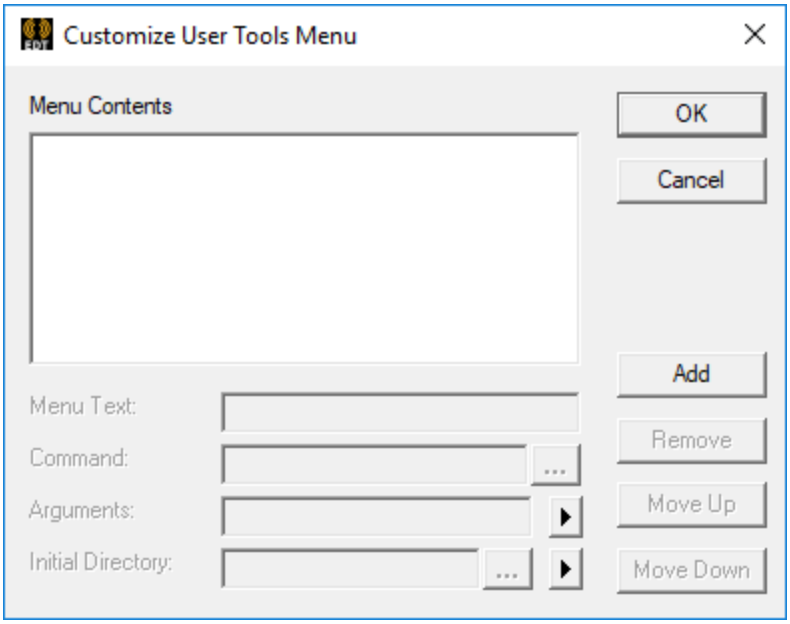
Adding External Tools to the Tools Menu

To add an executable to the Tools menu:

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

   This enables the following fields:

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

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

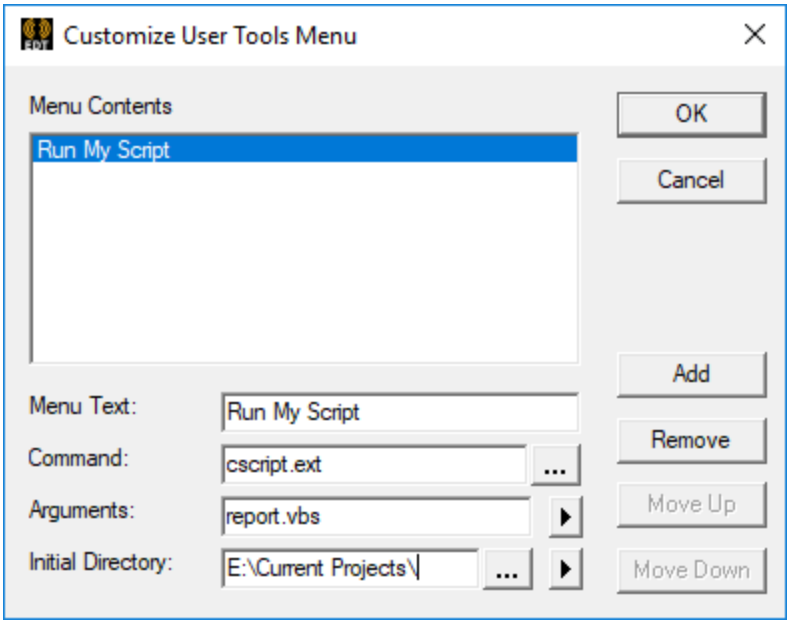
   Arguments field – accepts command arguments from the **>** button menu selections for File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

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

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

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

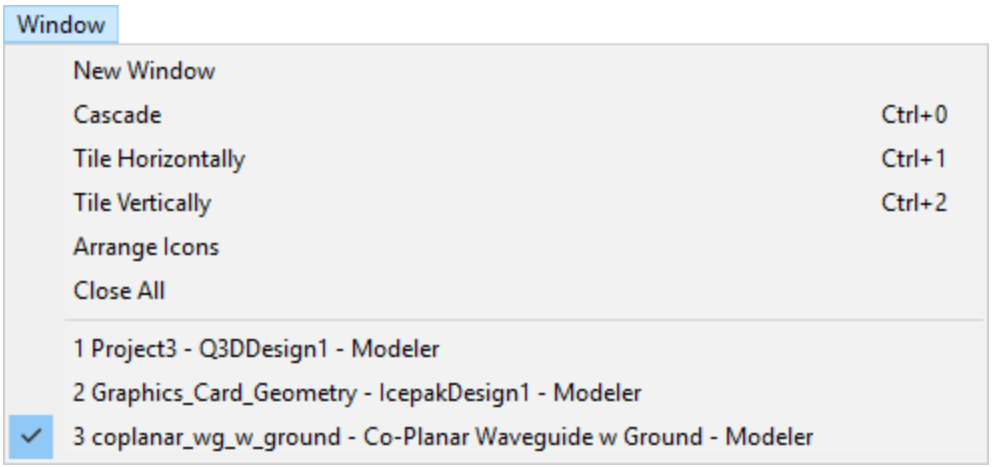
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Window Menu

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



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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

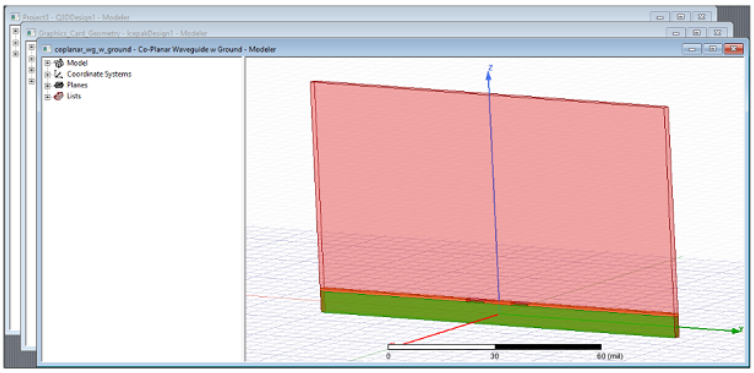
- Click **Tile Horizontally** (or type **h**) to arrange the open design windows in a top-to-bottom sequence:
- Click **Tile Vertically** (or type **v**) to arrange the open design windows in a side-to-side sequence:

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

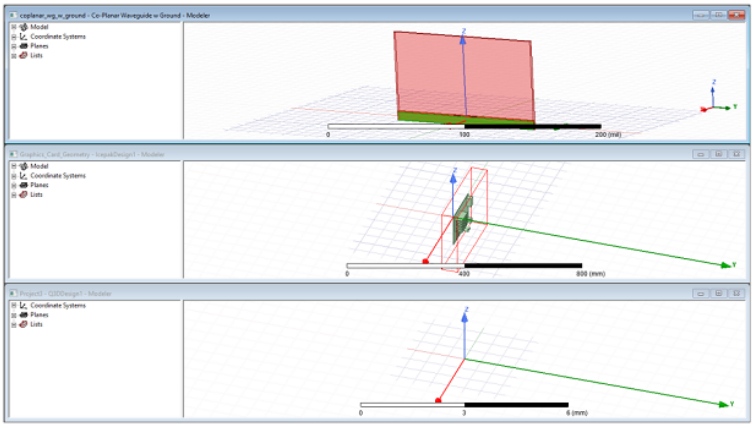
- Click **Arrange Icons** (or type **a**) to restore iconized windows to a neat row at the bottom of the Design Area, after the icons have been repositioned manually.
- Click **Close All** (or type **l**) to close all the editor windows in the Design Area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

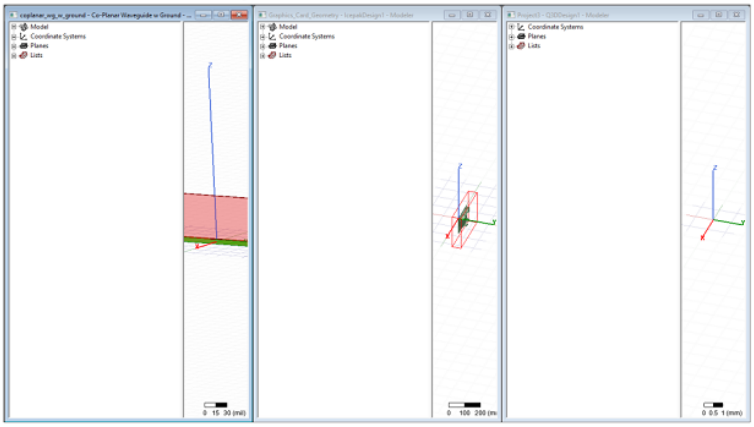
Cascade



Tile Horizontally



Tile Vertically



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Help Menu

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

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

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

- **Mechanical Help** – opens the Mechanical help within the Electronics help system. You can also access PDF versions from within the help system.
- **Mechanical Scripting Help** – opens the Mechanical scripting help.
- **Mechanical Getting Started Guides** – opens a list of links to the Mechanical Getting Started Guides, which walk you through projects that demonstrate product features.
- **Ansys Customer Support** – opens a browser page to the Ansys Customer Portal. At the website you can learn more about Ansys products and services and log on to contact Ansys technical support staff.
- **What's New in this Release** – opens a PDF that describes *What's New in Ansys Electronics Desktop* for 2025 R1.
- **Ansys Product Improvement Program** – opens a window describing the Product Improvement Program option.
- **License Settings** – Opens help for the License Settings tool.
- **About Ansys Electronics Desktop** – opens a dialog box that displays the Ansys Electromagnetics Suite release number and contains tabs that show information about the **Installed Components** and **Client License Settings**.
- **Ansys Innovation Courses** – opens a web page containing a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.
- **Ansys Learning Hub** – opens a web page with subscription based access to, virtual and self-paced learning across the Ansys Software portfolio.
- **Ansys Knowledge** – opens a web page to expert curated knowledge materials from FAQs to tutorials on simulation topics.
- **Ansys Learning Forum** – opens a web page to Ansys blog containing discussion and presentations from Ansys experts, partners and customers.
- **Customer Portal** – opens a web page to Ansys Product support.
- **About Electronics Desktop** - opens a dialog with version and release information, Ansys Electronics Desktop installed components, and licensing information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Context-Sensitive Help

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

Obtaining Information about the Software and Release

To obtain information about the software and release:

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

Shortcut Menus

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

Shortcut menu in the 3D Modeler window	Use the shortcut menu in the **3D Modeler** window to select, magnify, and move objects (zoom, rotate, etc.); change the view; perform boolean

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	operations; assign materials, boundaries, excitations, or mesh operations to objects; and work with field overlays.
Shortcut menus in the Project Manager window	Use the shortcut menus in the **Project Manager** window to manage Ansys Electronics Desktop project and design files and design properties; assign and edit boundaries, excitations, and mesh operations; add, analyze, and manage solution setups; add Optimetrics analyses; create post-processing reports; insert far- and near-field radiation setups; edit project definitions; and run Maxwell SPICE.
Shortcut menus in the History Tree	Use the shortcut menus in the **History** tree to expand or collapse groupings. If you select particular objects in the history tree, the shortcut menu lists the commands that you can apply to the selected object(s).
Shortcut menus in the Progress window	Use the shortcut menus in the **Progress** window during a simulation to **Abort** or **Clean Stop**.

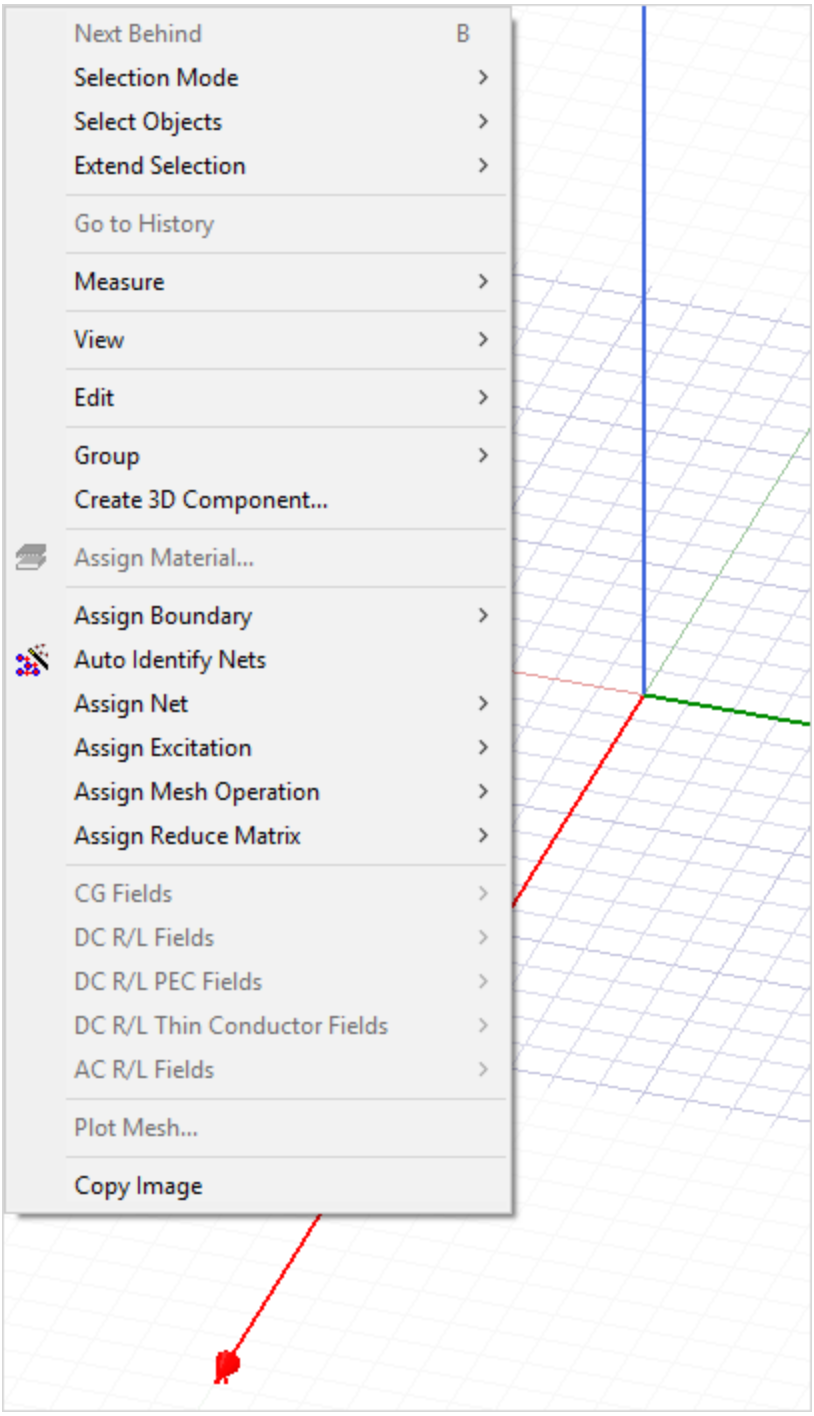
> **Note:**
>
> All commands available on shortcut menus are also available from the menu bar.

Shortcut Menu in the 3D Modeler Window

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

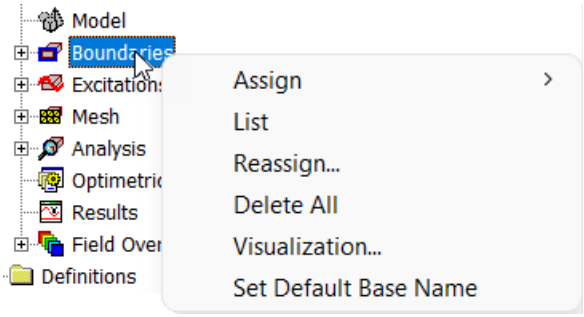


Shortcut Menus in the Project Manager Window

Each node, or item, in the project tree has a shortcut menu. For example, from the **Boundaries** shortcut menu, you can assign boundaries to selected entities; list previously assigned

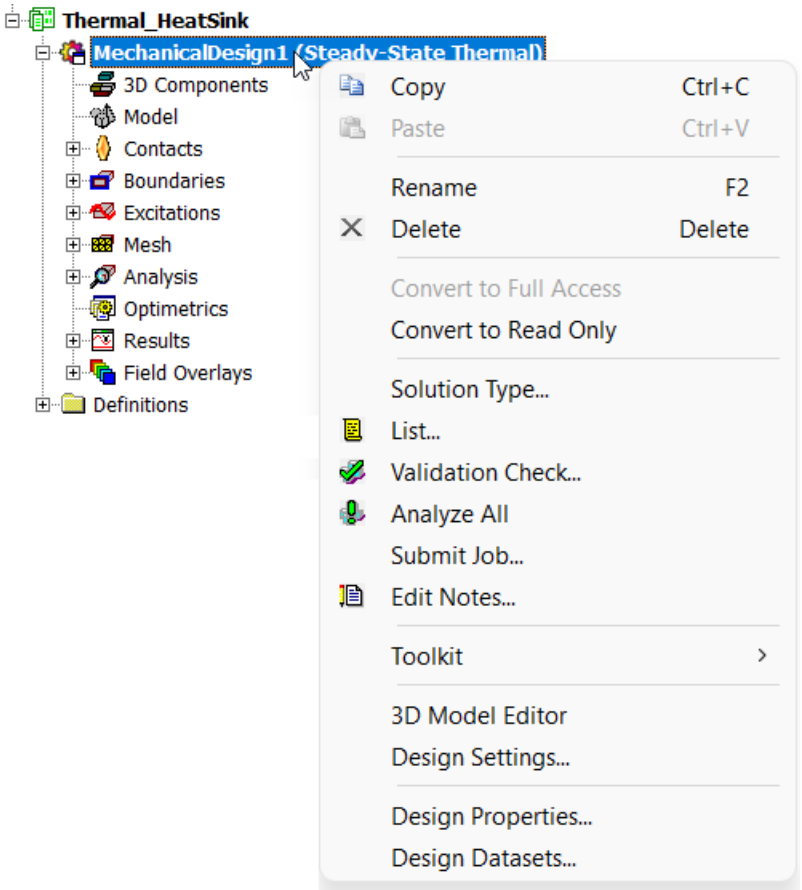
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundaries for the active design and review their information; access the *Reassign Boundary* dialog box; remove all boundary assignments; control the visualization options (that is, show or hide a boundary's geometry, name, or vectors); and access the *Set Base Boundary/Excitation Name* dialog box.



Other nodes have shortcut menus appropriate for the context.

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



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Customizing Ansys Electronics Desktop Menus

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

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

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

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

Select Different UI Types

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

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

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

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

Add a New Customized UI Type

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Names of the XML Files

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

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

Valid XML Elements and Attributes

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

- **Context** – child of DesignerMenu. It appears at least one time and has a required “name” attribute. For details on setting the context name, see the “Context Name” section below.

2. Child elements of **Context**:
   - **TopMenu** – For RightClickMenu.xml, do not use this element. You can specify child elements of TopMenu listed below under Context. For any other xml files, you need at least one TopMenu child element for any Context.

     Child elements of TopMenu:

     - **MenuName** – required, appears only one time.
     - **popupMenu** – may appear one or multiple times; child elements are the same as those of Topmenu.
     - **LeafMenu** – may appear one or multiple times.

       Child elements of **LeafMenu**:

       - **MenuName** – optional string.

         Pay attention to the character reference "&" in XML. In order for the name of the menu to be displayed as “Tools”, the XML syntax must be:

         <MenuName>&Tool</MenuName>

         “&“ is the character reference for ”&” in XML, while “&Tool” tells Ansys Electronics Desktop that Alt+T is the shortcut key for this menu.

       - **MenuID** – optional number.

         For a list of valid MenuIDs, please see ***Command IDs for Customizing AEDT Menus.xlsx*** under <*install_dir*>/v<*release_number*>/[Win64 or Linux64]/Help.

       - **ShowBitMap** – optional, “Yes” or “No” (default)
       - **Accelerator** – optional string (Example: Ctrl+N)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> To add a new LeafMenu, both “MenuName” and “MenuID” are required.

- **CustomMenu** – may appear one or multiple times; used to add a customized menu to run an external script (vbs or python).

  Child elements of **CustomMenu**:

  - **MenuName** – see LeafMenu for Usage
  - **ShowBitmap** – see LeafMenu for Usage
  - **Accelerator** – see LeafMenu for Usage
  - **ScriptPath** – required string; used to supply path to the script (use of $PROJECTDIR, $PERSONALLIB, $USERLIB, $SYSLIB variables is also allowed).

    Example:

    <ScriptPath>C:/Users/jwei/Python/HelloWorld.vbs</ScriptPath>

    or

    <ScriptPath>$PERSONALLIB/HelloWorld.py</ScriptPath>

> **Note:**
>
> To add a new CustomMenu, both MenuName and ScriptPath are required.

- **Separator** – may appear one or multiple times; has no child element.

3. All elements have an optional attribute of **action**.

   Valid values are:

   - **add** – can be applied to all elements other than “DesignerMenu” or “Context” . This element and all its child elements will be added to its parent. An optional attribute of “position” can be followed here to specify the position of this newly added menu. Position starts from 1. If no position is specified, this menu will be appended to its parent menu.
   - **useDefault** – default menus will be used for this menu(matched by MenuName or MenuId). Any of it’s child menu will be processed according to its action setting.
   - **delete** – can be applied to all elements other than “DesignerMenu” or “Context” . This menu (matched by MenuName or MenuID) will be deleted from its parent.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

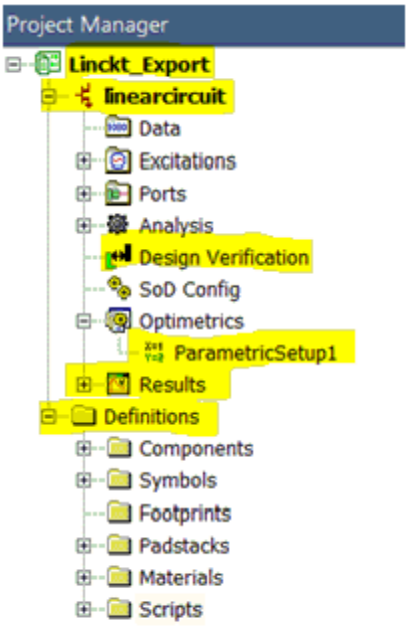
- For “DesignerMenu” and “Context”, only no action setting or “useDefault” is valid.
- The “DesignerMenu” action setting is significant because it is how you specify whether you want to construct your own menus or modify existing default menus. If this is set to “useDefault”, default menu settings are used for any menus not listed in your XML files. If no action is set, only the “File”, “Window” and “Help” menus display; no other default menus display. Menus listed in the XML file display after the “File” menu and before the “Window” menu. In this case, no action is needed for any other elements in the XML file. If any action is specified for any element, you will receive an error message and a default menu will be displayed for that context. A “Revert To Default” menu will be appended to the “Tools” menu. If no “Tools” menu is specified as a TopMenu, a “Tools” menu is created before the “Window” menu with one “Revert To Default” menu item.
- For any other elements, a missing action setting means that the parent action will be used.
- If any element’s action is set to “add”, then no action is needed for any of its child elements. If any action is specified, it will be ignored. This menu and all of its child menus will be added to the default menu.
- If any TopMenu or pop-upMenu’s action is set to “delete”, you don’t need to list any of its child menus. The menu will be deleted with all child menus.

Context Name

1. The “name” attribute specifies the name of the context for this product. You can have an "All" context if you set the action of DesignerMenu to UseDefault to specify a menu setting that you want to apply to all contexts for that product.
2. Valid context name for different xml files:
    - 2D Extractor.xml — “All”, “3d modeler”, “report2d”
    - CircuitDesign.xml — “All”, "SchematicEditor", “Layout”, “Netlist”, “report2d”
    - Circuit Netlist.xml —“All”, “Design”, “Netlist”
    - HFSS.xml — “All”, “3d modeler”, “report2d”
    - HFSS 3D Layout Design.xml — “All”, “Design”, “Layout”, “Layout3D Editor”, “report2d”
    - HFSS-IE.xml — “All”, “3d modeler”, “report2d”
    - NoDesignUI.xml — “All”, “No Context”, “project”, “FilterDesign”
    - Q3D Extractor.xml — “All”, “3d modeler”, “report2d”

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- RightClickMenu.xml — the following figure illustrates valid context names and how they display in Ansys Electronics Desktop.



  - **Project Folder** – "Project"; in the above figure, “Project” represents the “Linckt_Export” project folder.
  - **Design Instance Folder** – “Circuit Design”, “HFSS 3D Layout Design”, “HFSS”, “Q3D Extractor”, “Circuit Netlist”, “HFSS-IE”, “2D Extractor”
  - **Definitions Folder** – “Definitions”; in the above figure, the Definitions folder is highlighted.
  - For any Folder or Item under the Design Instance or Definitions folder, use the “/” notation to specify the path to the Folder or Item for context name. In the above figure: Use “Circuit Design/Results” for the Results folder. Use “Circuit Design/Design Verification” for the Design Verification folder.
    - For any folder or item under this, use the context name for the folder appended with “/Item”. In the above figure, “Circuit Design/Optimetrics/Item” represents ParametricSetup1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For the Report folder under Results, please use the context name for the Results folder appended with "Report"; for example: "Circuit Design/Results/Report".
>
> For the Trace folder under Report, please use the context name for the Report appended with "Trace"; for example: "Circuit Design/Results/Report/Trace"

Sample XML Files

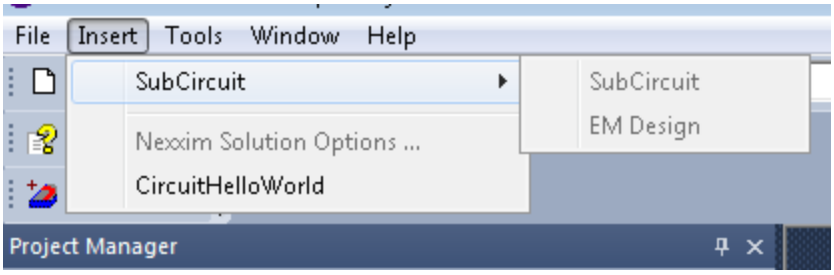
1. **Circuit Netlist.xml Sample**

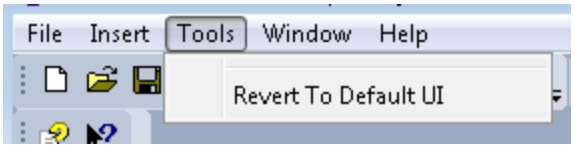
```
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">
   <Context name="Design">
      <TopMenu>
         <MenuName>Insert</MenuName>
         <pop-upMenu>
            <MenuName>S&amp;ubCircuit</MenuName>
            <LeafMenu>
               <MenuName>&amp;SubCircuit</MenuName>
               <MenuID>55500</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
            <LeafMenu>
               <MenuName>&amp;EM Design</MenuName>
               <MenuID>55502</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
         </pop-upMenu>
         <Separator></Separator>
         <LeafMenu>
            <MenuName>Nexxim Solution &amp;Options ...</MenuName>

            <MenuID>38460</MenuID>
            <ShowBitMap>No</ShowBitMap>
            <Accelerator></Accelerator>
         </LeafMenu>
         <CustomMenu>
            <MenuName>CircuitHelloWorld</MenuName>
            <ScriptPath>$USERLIB/HelloWorld.vbs</ScriptPath>
            <ShowBitMap>No</ShowBitMap>
         </CustomMenu>
      </TopMenu>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    </Context>
</DesignerMenu>
```

Menus created by processing the above sample, Circuit Netlist.xml





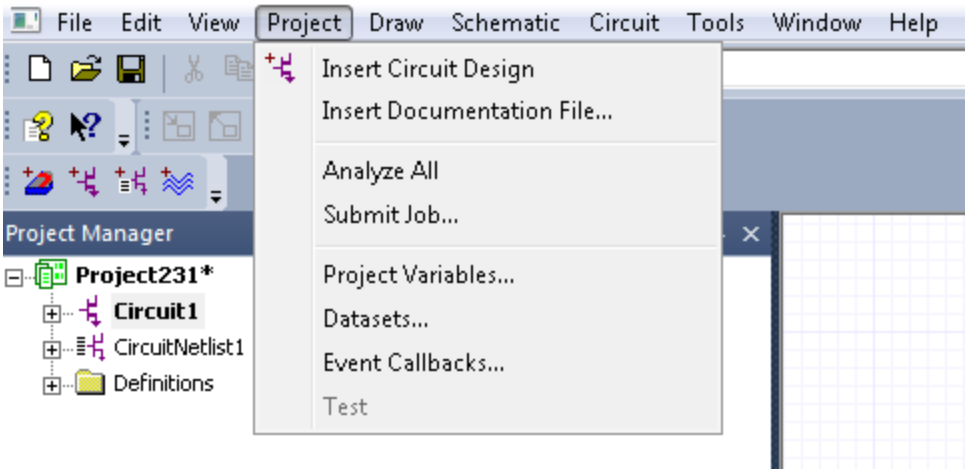
2. **Circuit Design.xml Sample**

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu"
action="useDefault">
  <UseProjectWindowSelectionContext>false</UseProjectWindowSelectionC
  ontext>
  <Context name="All">
    <TopMenu>
          <MenuName>&amp;amp;Project</MenuName>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS 3D &amp;amp;Layout
        Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS-IE Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert Q3D Extractor Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
```

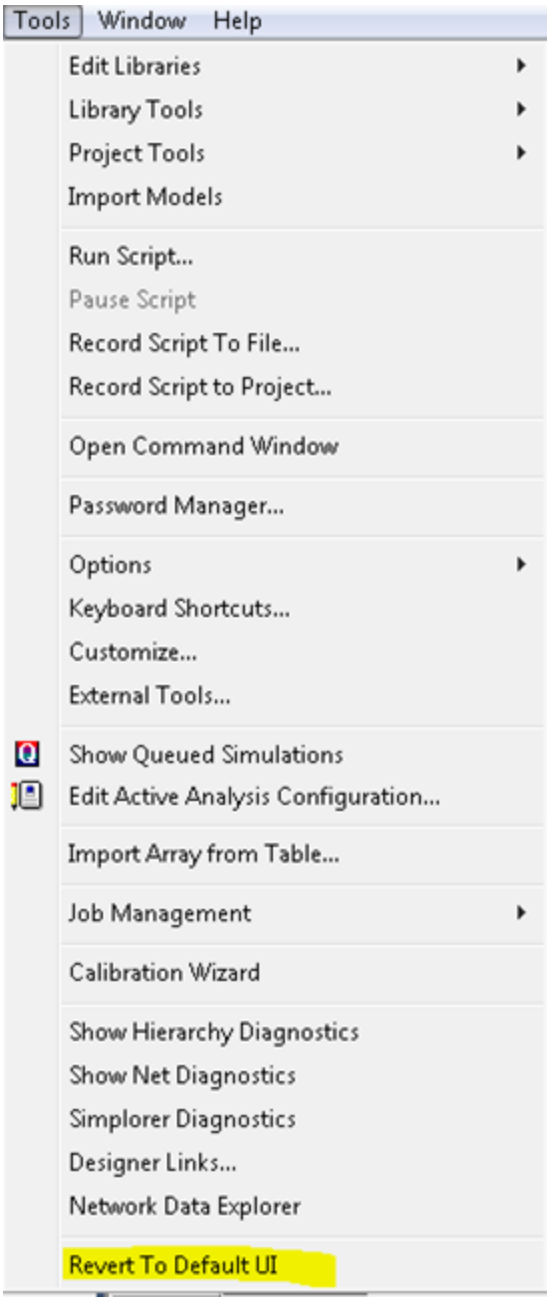

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                <MenuName>Insert 2D Extractor Design</MenuName>
        </LeafMenu>
        <LeafMenu action="delete">
                <MenuName>Insert Circuit &amp;amp;Netlist</MenuName>
        </LeafMenu>
        <LeafMenu action="add">
                <MenuName>Test</MenuName>
                <MenuID>3333</MenuID>
        </LeafMenu>
      </TopMenu>
   </Context>
</DesignerMenu>
```

Menus created by processing the above XML sample, Circuit Design.xml



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



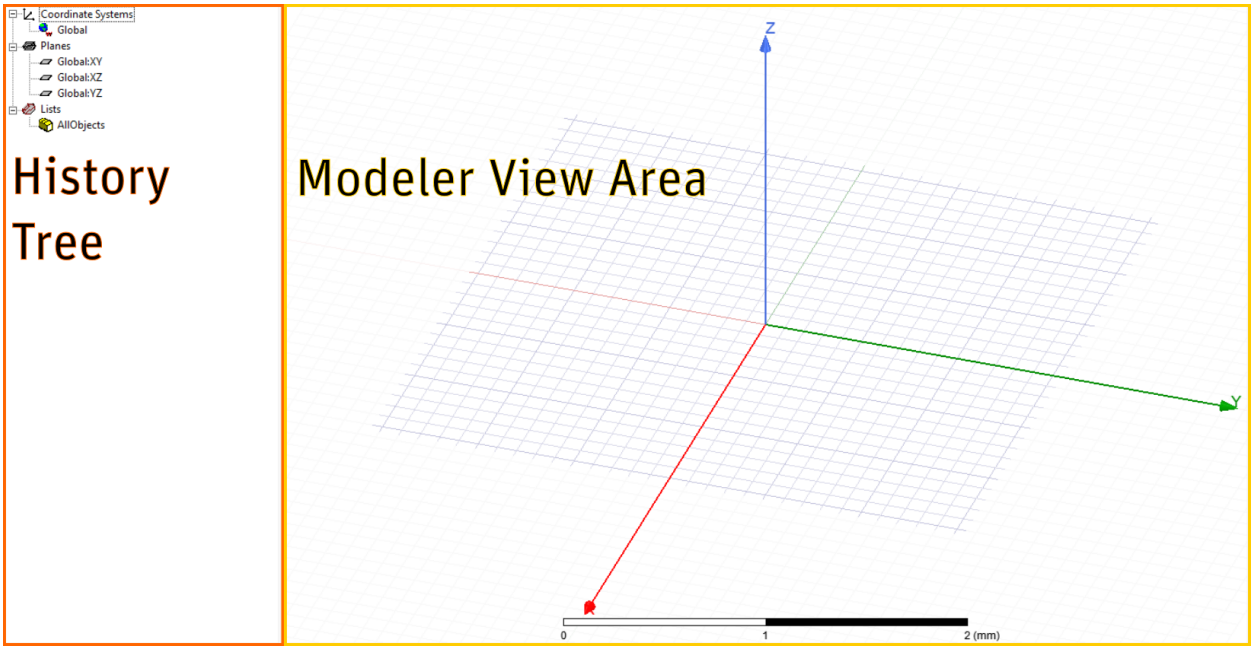
Ansys Electronics Desktop Design Area

The Design Area of the desktop can display one or more editor windows and report windows, depending on the type of designs you create or load. See the Desktop Windows topic for ways to manipulate the windows in the Design Area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Window

The **3D Modeler** window is the area where you create the model geometry. It appears to the right of the **Project Manager** window after you insert an Ansys Electronics Desktop design into a project. The **3D Modeler** window consists of the modeler view area, or grid, and the History Tree. The two areas are highlighted in the following figure:



You can open a new 3D Modeler window in one of two ways:

- Insert a new Ansys Electronics Desktop design into the current project, or
- Double-click an Ansys Electronics Desktop design in the Project Tree

The model you draw is saved with the current project when you click **File** > **Save**.

Objects are drawn in the **3D Modeler** window. You can create 3D objects using Ansys Electronics Desktop's Draw menu commands, or you can draw 1D and 2D objects and manipulate them to create 3D objects. See: Drawing a Model.

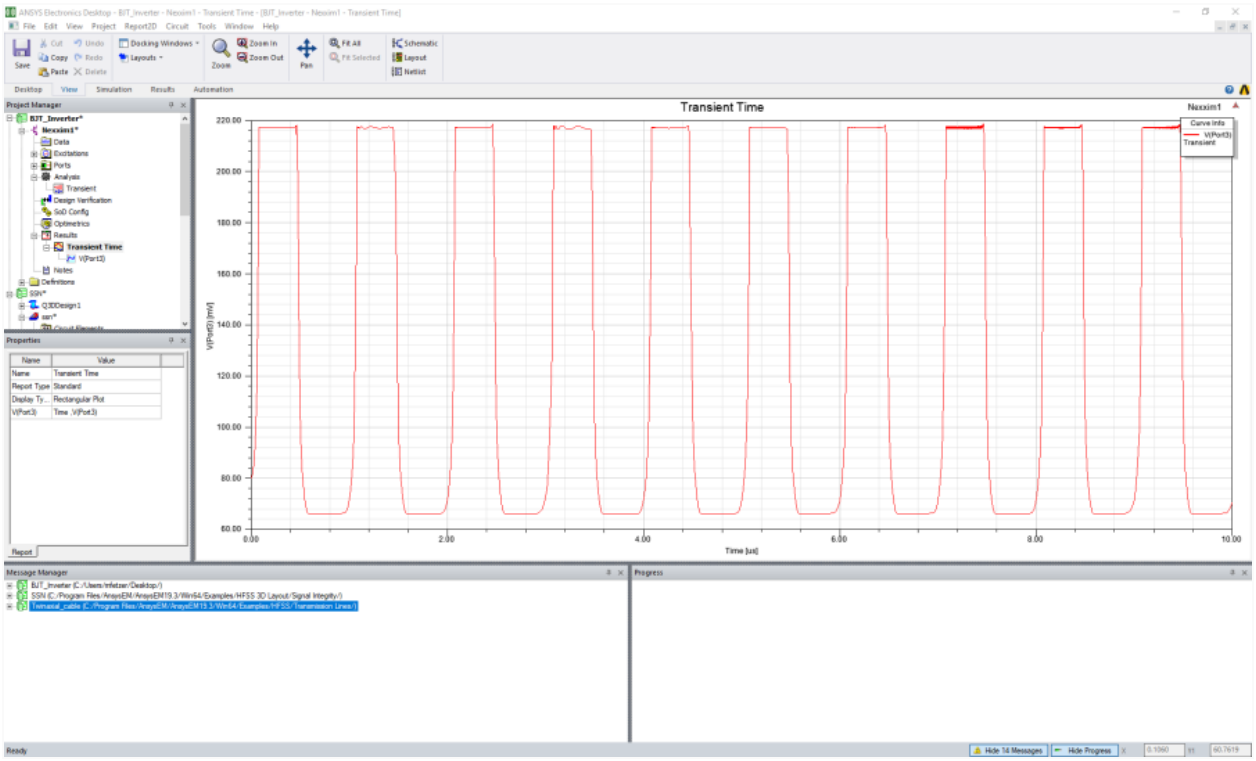
You can modify the view of objects in the **3D Modeler** window without changing their actual dimensions or positions. Besides menu and icon commands, you can also use **Alt+click**, mouse buttons, and keyboard shortcuts. See: Modifying the Model View.

Report Window

When a design has been successfully simulated, you can generate a report of results in a wide variety of forms, including XY graphs, polar graphs, 3D graphs, Smith charts, and data tables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Various attributes of each can be customized to your liking. The following shows a 2D report window in Electronics Desktop:

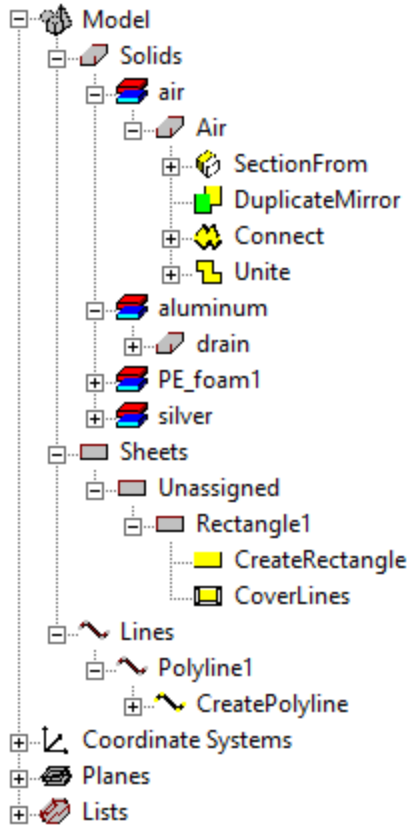


For more information, see Post Processing and Generating Reports.

Ansys Electronics Desktop History Tree

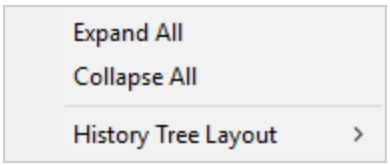
The history tree in the **3D Modeler** window lists the active model's structure and grid details.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Expand or collapse items in the tree using the **+** and **-** boxes.

Expand or collapse all items in a category by right-clicking to open a shortcut menu:



> **Note:**
>
> Some objects have more extensive shortcut menus than others, but all of them have these three options.

Selecting any item in the History Tree also selects it in the design area and populates the **Properties** window with its properties.

You can perform the following tasks with the History Tree:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Organize the History Tree
- Control the View of Objects in the History Tree
- Use Group Commands for Modeler Objects
- Select Objects in the History Tree
- View and Edit Commands on History Tree Objects

The History Tree can contain the following model details:

Invalid	Lists all invalid objects.
3D Component Names	Lists any 3D components added to the design. The History Tree lists their Solids, Coordinate Systems, and Planes separately from objects drawn in the modeler or imported to the modeler.
Solids	Lists the model's solid objects and a history of the commands carried out on each object.
Sheets	Lists the sheets in the model 3D design area.
Lines	Lists all line objects included in the active model. See Drawing a Straight Line for information on how to draw a line object.
Points	Lists all point objects included in the active model. See Drawing a Point for information on how to draw a point object.
Groups	Lists folders for Groups you have created or imported. See Group Commands for Modeler Objects.
Coordinate Systems	Lists all the coordinate systems for the active model. See Setting Coordinate Systems for more information on this model detail.
Planes	Lists the planes for all the coordinate systems. When you create a coordinate system, default planes are created on its xy, yz, and xz planes.
Lists	Lists the object or face lists for the active model. By default, a list called “AllObjects” appears. You can add a specific group of objects or surfaces for field plot or for calculation. See: Creating an Object List and Creating a Face List.

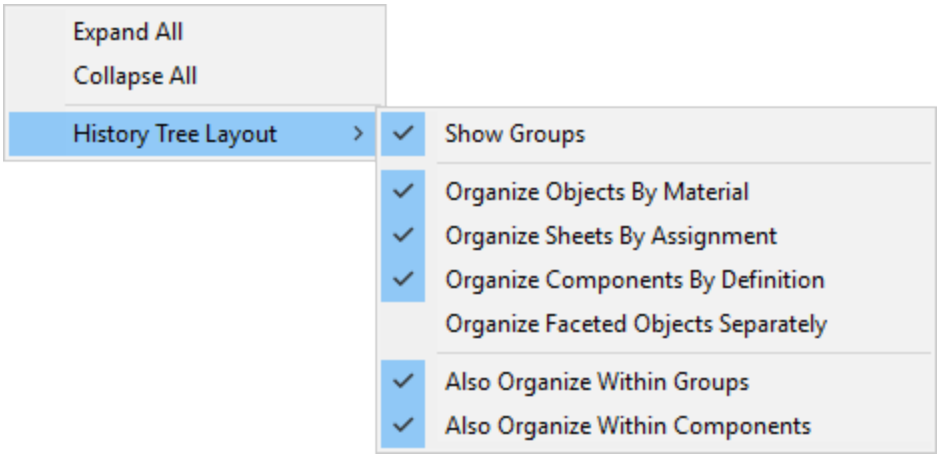
Note:

While objects created in Mechanical can always be classed in the History Tree as a solid, sheet, or wire, some imported objects may have mixture of these. Mechanical places such objects in an Unclassified folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Organizing the History Tree

To organize and sort objects in the History Tree, right-click any object in the history tree to display the shortcut menu and select **History Tree Layout**.



The shortcut menu contains the following commands:

- **Show Groups**
- **Organize Objects By Material:** This option is applicable to 3D objects and also to sheet objects that have the *Shell Element* attribute selected and a material selection specified (Mechanical - Thermal designs only). Imported solid objects that have no material specified yet are listed in a group named *Unassigned*. Non-model objects are listed in a *Non Model* group.
- **Organize Sheets By Assignment:** This option is applicable to sheet objects only (including Mechanical – Thermal shell objects with a boundary or excitation assigned). Sheet objects are grouped according to the boundary or excitation assigned. When a sheet is assigned both a boundary and an excitation, it is grouped according to the last assigned excitation (that is, the group name is the last assigned excitation). When multiple boundaries are assigned to a sheet, the sheet is grouped according to the boundary with the highest priority, and that boundary is used as the group name. The boundary priority depends on the design type, as follows:
  - **HFSS and Maxwell:** Priority is specified by the user.
  - **Icepak, Mechanical, and Q3D:** The last created boundary has the highest priority
- **Organize Components By Definition**
- **Organize Faceted Objects Separately**
- **Also Organize Within Groups**
- **Also Organize Within Components**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

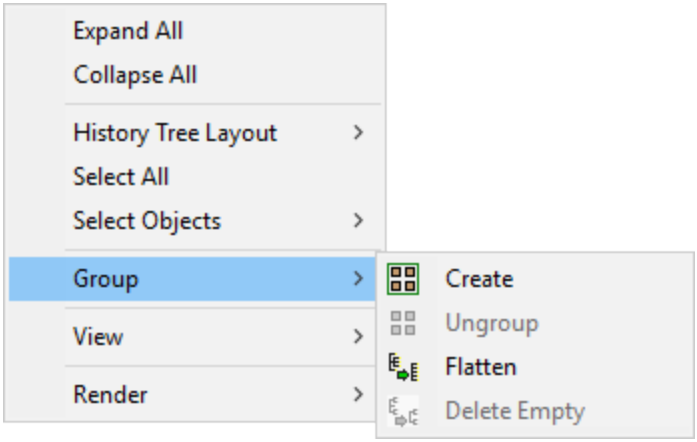
Controlling the View of Objects in the History Tree

To control the view and visibility of an object such as a box, right-click an object in the history tree to display the shortcut menu and select **View**. The shortcut menu contains the following commands:

- **Fit in Active View**
- **Hide in Active View**
- **Show in Active View**
- **Fit in All Views**
- **Hide in All Views**
- **Show in All Views**

Group Commands for Modeler Objects

The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.

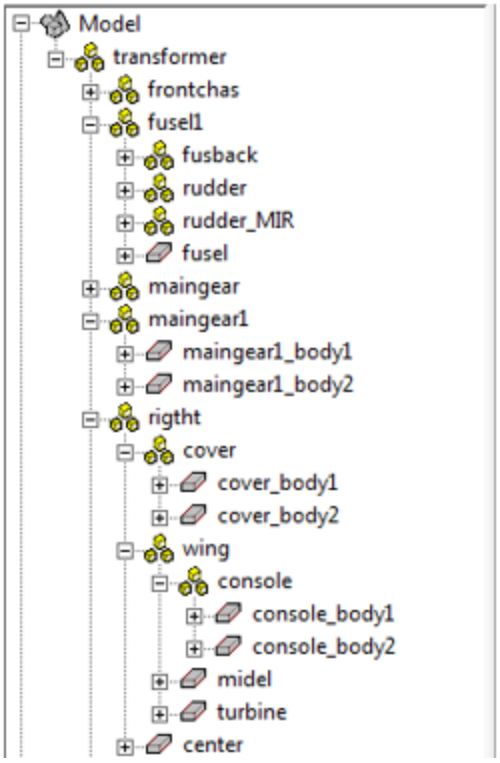


The **Group** commands support the following features:

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-group containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Accessing Group Commands

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout>Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Operations on Groups

Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Copy and Paste of Objects in Groups

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

You can select group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

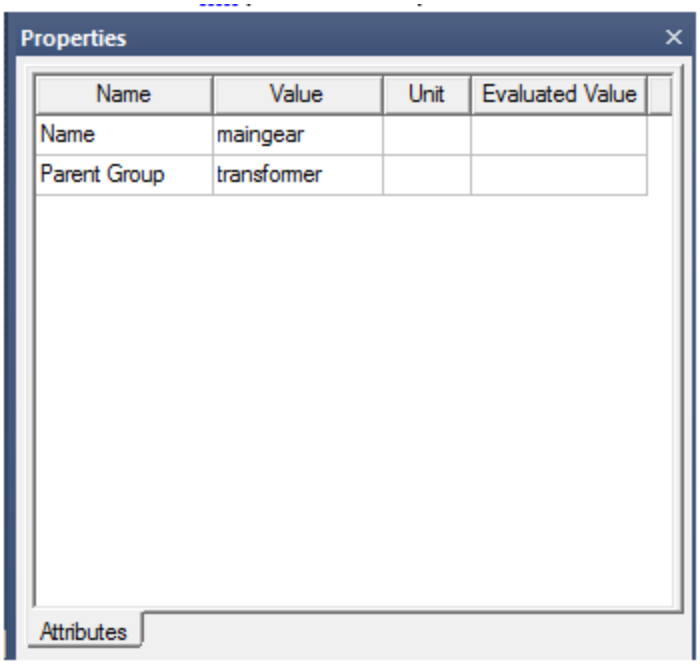
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See: 3D Modeler Options: Group Options.

Group Properties

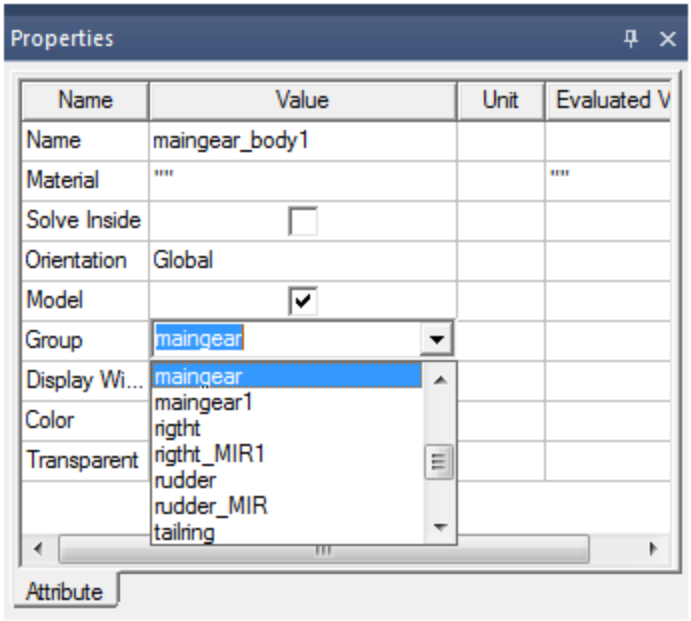
A Group’s Property tab is shown when you select a group in the history tree. The Group’s properties include Name and parent Group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Group Property Window

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the existing groups listed or create a new group.



Group assignment could be changed in Object's property window

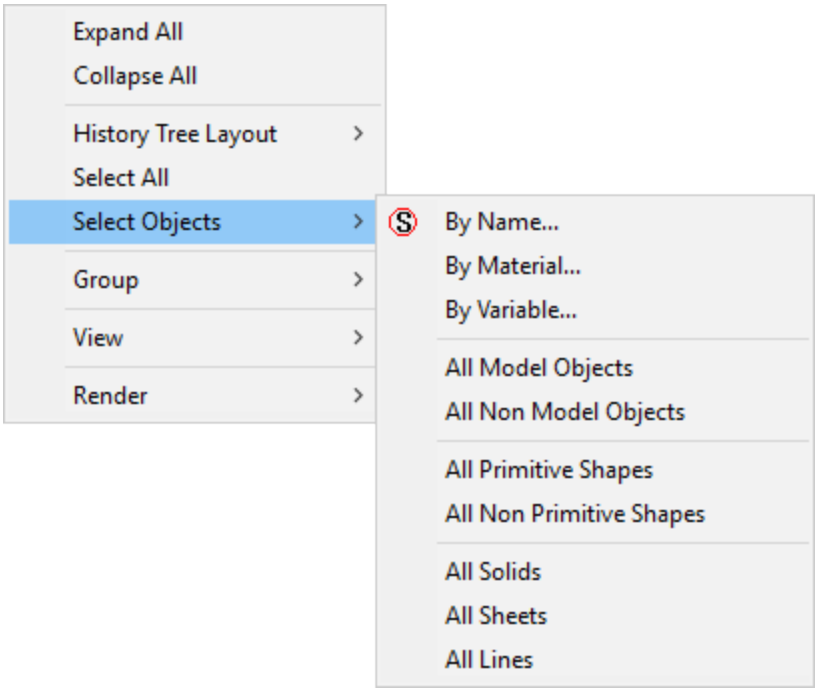
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects in the History Tree

Selecting objects in the History Tree also selects them in the View window. This can be useful for complex objects, when it may be easier to find objects of interest by name or material because the object is inside or behind others.

Ctrl+click to make multiple selections in the History Tree. You can select a range of objects by clicking the first and using **Shift+click** to select all in the range. You can also click and drag the mouse to make rubber band selections. Only visible objects are selected. That is, if the hierarchy is closed under the selection, any operand parts are ignored and do not interfere with cut and paste operations.

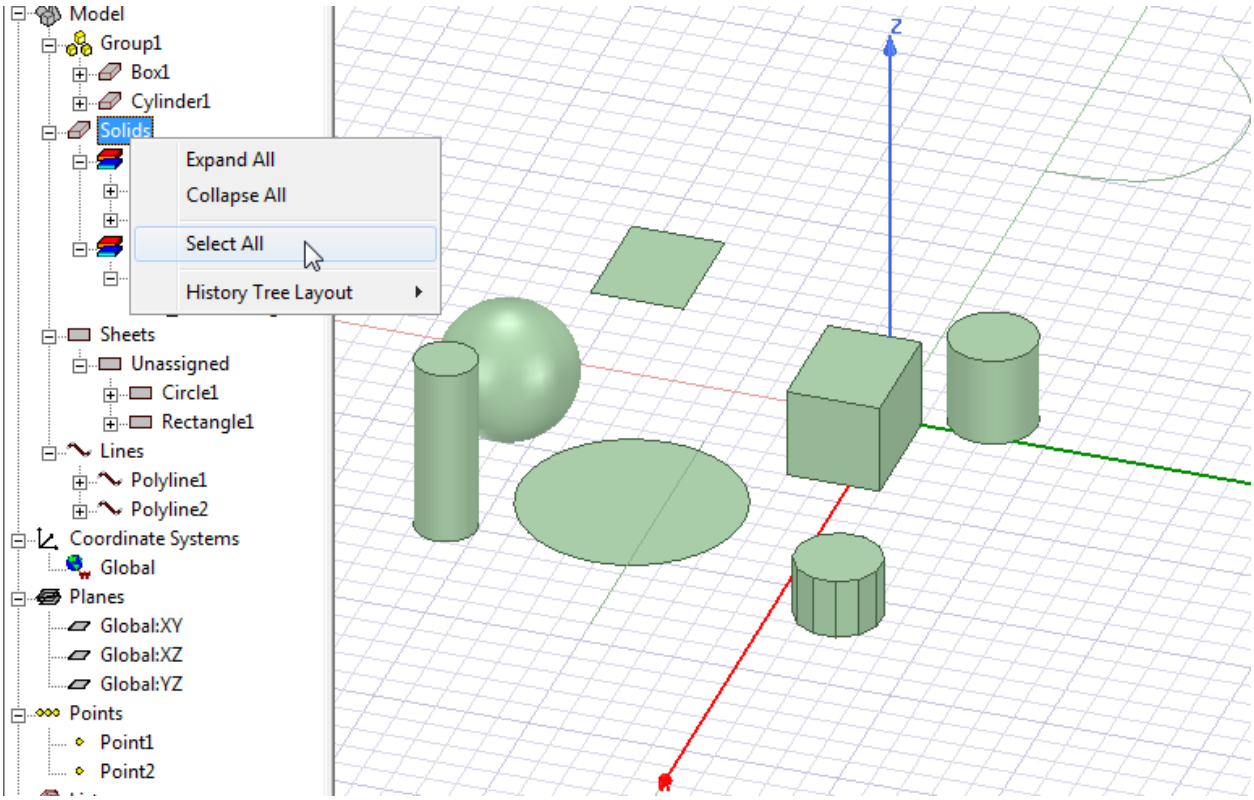
Right-click **Model** in the History Tree to see additional selection options:



You can right-click any History Tree object or grouping to display a shortcut menu with a **Select All** option. This lets you select all non-grouped solids, sheets, lines, non-model objects, or all unclassified objects, as well as all non-grouped objects assigned to specific materials.

For example, right-click Solids and then **Select All**:

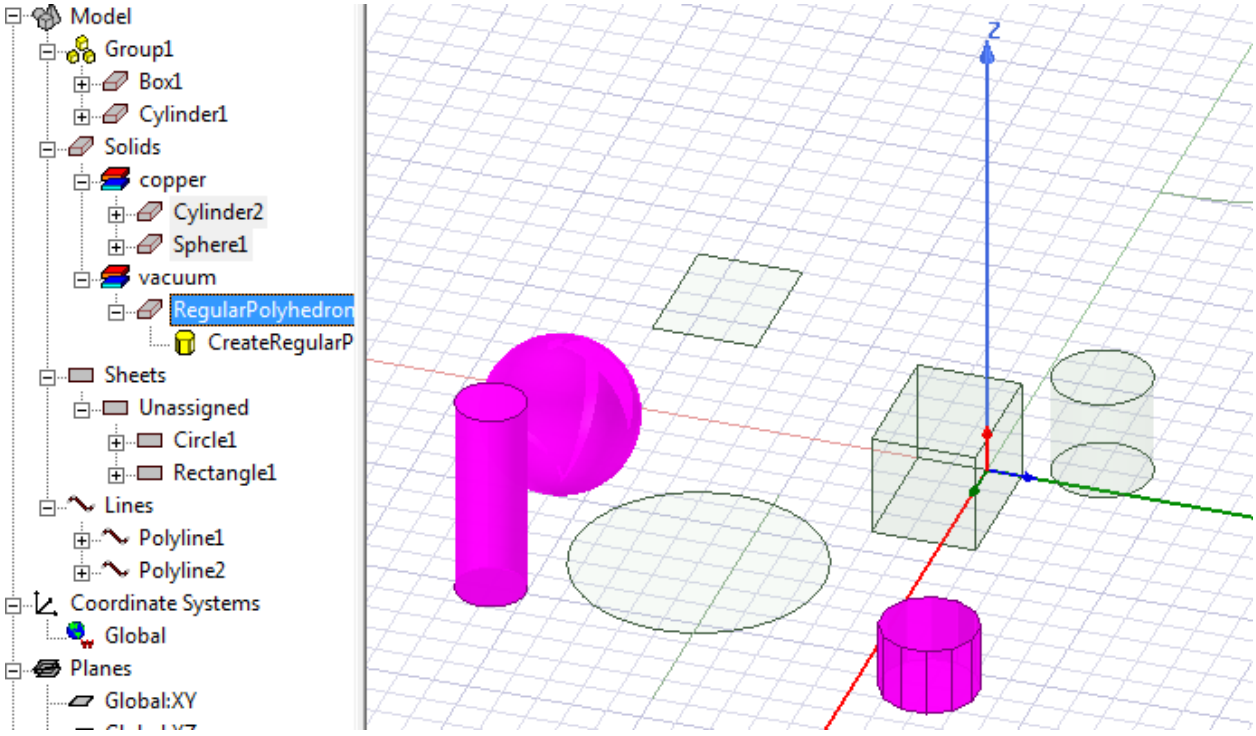
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After clicking **Select All**, the History tree and the **Model** window highlight the selected Solid Objects.

Notice that Box1 and Cylinder1 are not selected because they are in Group1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



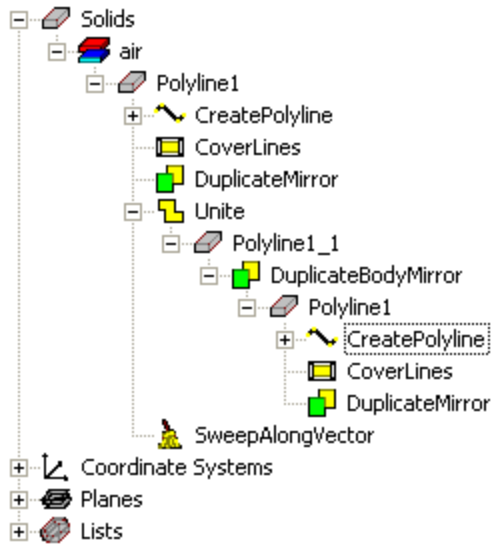
The right-click for Model and **Select All** does select the members of Group1 because they are part of the model.

If you select a geometric object, such as a Box or Cylinder, the shortcut menu **Select** command offers choices to select **All Faces**, **All Edges**, or **Sheet Edges** for that object.

Viewing and Editing Commands on History Tree Objects

The history tree also lists the history of all commands carried out on a model's objects (for example, “CreateBox” or “Subtract”). This history is displayed in the order in which it occurred. Here is a history tree from the waveguide combiner example project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

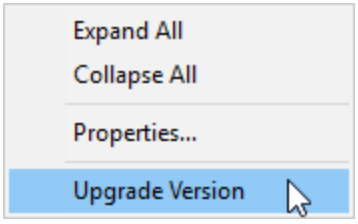


Selecting a command in the history highlights the object in the Modeler window and shows that object's properties (if available) in the docked **Properties** window. You can look at the fields in the **Properties** window to see any editable fields for that command, such as coordinate system, line type, coordinates, or units.

For some commands, such as **Edit** > **Arrange** > **Move** or boolean operations, selecting them in the history tree enables the delete icon (**X**) on the toolbar and the **Edit** > **Delete** menu. In these cases, you can delete those commands from the history tree as a way of undoing those operations. As an alternative to deletion, you can check **Suppress command** in the **Properties** window for that command. This undoes the effect of a command on an object without removing it from the History Tree.

Upgrading Version in the History Tree

Right-click an operation in the History Tree to see the **Upgrade Version** command.



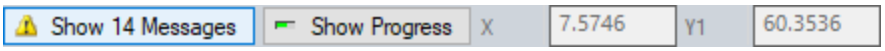
By default, the Modeler opens legacy projects using the modeling kernel version under which the project was saved, so that users don't see side effects from slight topology changes between versions. The **Upgrade Version** operation forces Electronics Desktop to use the latest modeling

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kernel. Typically, this command would only be used when a needed fix is available for the newest version.

Ansys Electronics Desktop Status Bar

The status bar is located at the bottom of the application window. By default, it contains buttons to Show or Hide the Message window and Progress window.

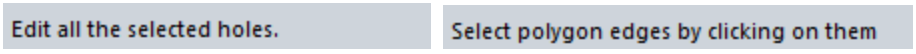


Note that the Show Messages button indicates the number of messages.

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

The status bar also displays helpful information about the current selection or command.

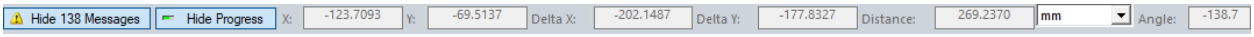
Directions for inputs appear on the left side of the status bar:



Depending on the command being performed, the status bar can display the following:

- **X**, **Y**, and **Z** coordinate boxes
- A drop-down menu for entering absolute, relative, cartesian, cylindrical, or spherical coordinates
- The model's units of measurement

These coordinate boxes and drop-down menus appear on the right side of the status bar:



To toggle status bar display:

- Click **View** > **Status Bar**.

A check mark next to this command indicates that the status bar is visible.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyboard Shortcuts

The following keyboard shortcuts apply to Mechanical in general:

F1	Help (context-sensitive)
Ctrl + F4	Close window
Alt + F4	Close program
Ctrl + C	Copy
Ctrl + N	New Project
Ctrl + O	Open
Ctrl + P	Print
Ctrl + V	Paste
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically

See the subtopics in this Help branch for additional information concerning shortcuts and view navigation.

To customize the shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Desktop Shortcuts

Modifier + key	Hold down the modifier, such as Shift or Ctrl, and press the key.

The following Ansys Electronics Desktop shortcut key combinations are available at any time:

Ctrl + N	New
Ctrl + O	Open
Ctrl + P	Print
Ctrl + S	Save
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete	Delete
F1	Open help

3D Modeler Window Shortcuts

The following shortcut key combinations are available when the Modeler window is active. View navigation shortcuts are also applicable to 3D Plot windows. The table below is organized into four sections, with the shortcuts separated according to the following categories:

- Commands and options used for the selection of entities
- Choosing the entry mode to use when creating geometry
- View navigation (rotating, zooming, panning, choosing a view orientation)
- Choosing the geometry rendering mode (wireframe or shaded)

> **Note:**
>
> In the following table, **LMB** is an abbreviation for *Left Mouse Button*, and **MMB** means *Middle Mouse Button*.
>
> **Wheel** refers to rotation of the mouse wheel that is integrated with the middle button.

Entity Selection Commands and Options:	
B	Select face or object **behind** current selection
F	Select **faces** mode
O	Select **objects** mode
E	Select **edges** mode
V	Select **vertices** mode
M	**Multi-select** mode
Ctrl + A	Select **all** visible objects
Ctrl + Shift + A	Deselect all objects
Geometry Properties Entry Mode:	
F3	Switch to the **point** entry mode (using the mouse or coordinate text boxes to draw objects). Switching takes effect only after the drawing of an object has started.
F4	Switch to the **dialog box** entry mode (draw objects by specifying properties in the Command and Attribute tab of a dialog box). Switching takes effect only after the drawing of an object has started.
View Navigation:	
Ctrl + D	**Fit** view

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + '+'	**Zoom in** (screen center)
Ctrl + '-'	**Zoom out** (screen center)
Alt + double-click	Change the view orientation to one of nine predefined views depending on where you double-click within the Modeler window. See Changing the Model View with Alt+Double-Click Areas for more information.
Alt + *arrow keys*	**Rotate** the view about a vertical or horizontal axis. Each press of an arrow key rotates the view 2.5°.
Ctrl + *arrow keys*	**Pan** the view vertically or horizontally. Each press of an arrow key pans the view 10 screen pixels.
MMB + drag (no modifier)	Freely **rotate** the view orientation (except when clicking and dragging near the window border). When clicking near the window border, rotation is constrained about the screen-normal axis.
MMB + X + drag	Constrained **rotation** of the view orientation about the **X** axis
MMB + Y + drag	Constrained **rotation** of the view orientation about the **Y** axis
MMB + Z + drag	Constrained **rotation** of the view orientation about the **Z** axis
Wheel	Rotate the mouse wheel to **zoom** in and out. Rotate the top of the wheel forward (away from the mouse) to zoom **in** closer, and rotate it rearward to zoom **out**.
Shift + MMB + drag	**Zoom** in or out. Moving the cursor upward zooms in closer. Moving it downward zooms out. (* see note below)
Ctrl + MMB + drag	**Pan** the view (* see note below)
(*) **Note:** The listed actions for **Shift** and **Ctrl** keys in combination with the **MMB** are the view navigation assignments for 2025 R1. Simultaneously, legacy hot keys using **Shift** or **Alt + Shift** in combination with the **LMB** for these functions continue to be supported by default. Optionally, you can choose to disable the legacy shortcuts that were applicable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.	
Geometry Rendering Modes:	
F6	Render the geometry as wireframes (no shading of faces)
F7	Render the geometry with smooth, shaded faces

To customize shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Report Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

The following shortcut key combinations are available when a Report window is open:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + A	Select all traces
Ctrl + C	Copy selected trace
Ctrl + V	Paste
Ctrl + X	Cut selected trace
Ctrl + Y	Redo insert report
Ctrl + Z	Undo insert report
Delete	Delete

Choosing the View Navigation Options

New mouse button and modifier key assignments (hotkeys) have been added for navigating the model view (rotate, pan, and zoom functions). Simultaneously, legacy hotkeys for these functions continue to be supported by default. Optionally, you can choose to disable the legacy view navigation shortcuts that were applicable to version 2023 R2 and earlier, as follows:

1. Access the *Options* dialog box using one of the following two methods:
   - From the menu bar, click **Tools** > **Options** > **General Options**.
   - From the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select **General** > **User Interface**. Then:
   - To use either the legacy or current view navigation buttons and hotkey combinations, select **Enable Legacy View Navigation**. This option is selected by default.
   - To use only the current view navigation behavior and disable legacy button/hotkey support, ensure that **Enable Legacy View Navigation** is cleared.
3. Click **OK**.

   The navigation option change becomes effective immediately (no program restart required).

Current vs. Legacy View Navigation

The following table summarizes changes in mouse-button and hotkey assignments between the current scheme (version 2024 R1 and newer) and the legacy scheme. "*Legacy*" assignments are applicable to version 2023 R2 or earlier and when the *Enable Legacy View Navigation* option is selected in newer versions):

> **Note:**
>
> The *"Current"* assignments are available whether or not the *Enable Legacy View Navigation* option is selected:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

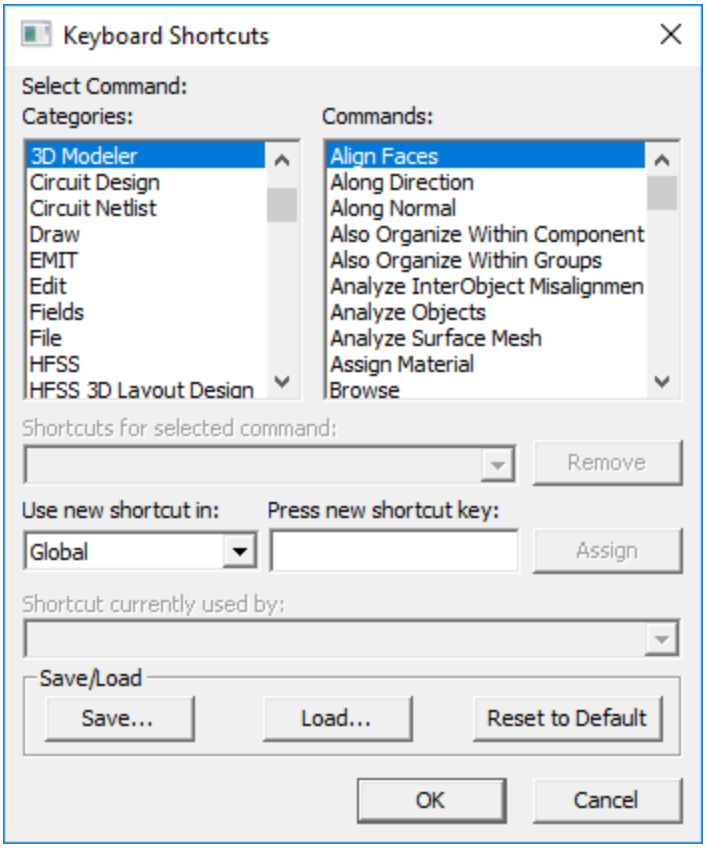
Function	Key / Mouse Button Assignment	
	Current	**Legacy**
Pan (all contexts)	Ctrl + MMB + drag or Ctrl + *arrow keys*	Shift + LMB + drag or Ctrl + *arrow keys*
Rotate (3D contexts)	MMB + drag or Alt + *arrow keys*	MMB + drag or Alt + LMB + drag or Alt + *arrow keys*
Zoom (all contexts)	Wheel or Shift + MMB + drag or Ctrl + "+", Ctrl + "-"	Wheel or Alt + Shift + LMB + drag or Ctrl + "+", Ctrl + "-"

- "**LMB**" = Left Mouse Button
- "**MMB**" = Middle Mouse Button
- "**Wheel**" refers to rotation of the mouse wheel.
- In some 2D contexts, where rotation is not applicable, you can use **MMB + drag** to pan the view for either scheme.
- For **Alt + *arrow keys***, rotation is about a vertical or horizontal axis, regardless of the model view orientation.
- For zooming, if using the "=\+" key, instead of the "+" key on the numeric keypad, do **not** press Shift.

Custom Keyboard Shortcuts

Click **Tools** > **Keyboard Shortcuts** to open the **Keyboard Shortcuts** window. Here, you can view existing assignments, create new shortcuts, and save or load assignment files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting a **Command Category** populates the **Commands** list with the available commands for that category. If the command has an assigned shortcut, it is displayed in the **Shortcuts for the selected command** field. You can use the **Remove** button to disable the shortcut for the selected command. If the selected command does not have an assigned shortcut, the **Shortcuts for selected command** field and the **Remove** button are unavailable.

To create a new shortcut key:

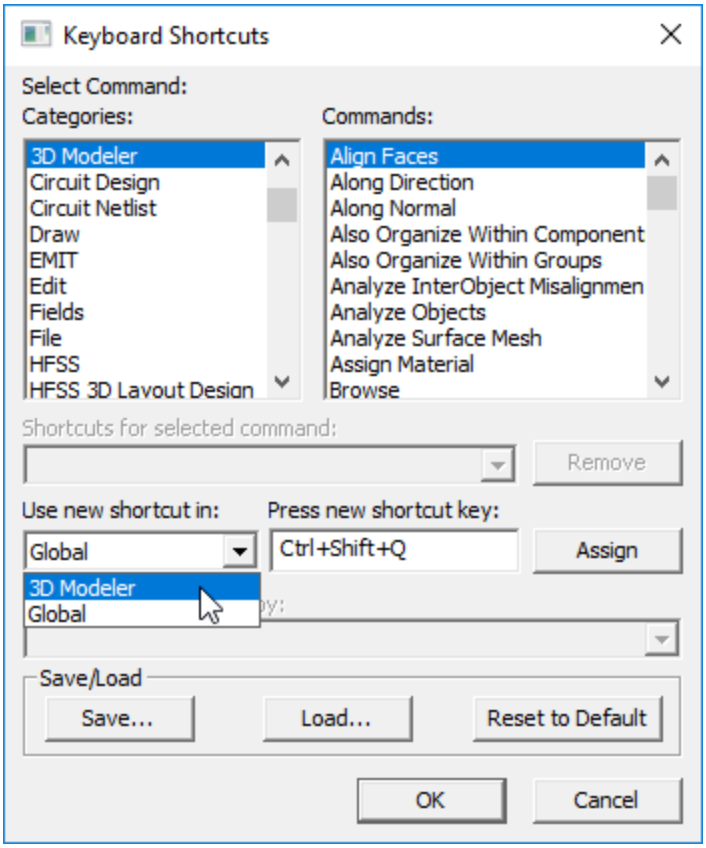
1. Select the **Category** and **Command**.
2. If you want to disable a current assignment for the selected command, click **Remove**.
3. To assign a keyboard shortcut, place the cursor in the **Press new shortcut key** field.

   The field displays the keystrokes you make. When you have made keystrokes, the **Assign** button becomes available. If you combine keystrokes, these are displayed with a plus sign (+) between them. For example, Ctrl+P or Alt+O.

   If the shortcut you select is currently in use, that information displays in the **Shortcut currently used by** field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. The **Use new shortcut in** drop-down list displays **Global** by default, which means that the shortcut will apply to all applicable contexts. If a limited context exists, the menu offers a selection.



5. When you have made your desired assignments, you can save them to a named file.

   Click **Save...** to save the assignments to your desired location in Ansoft Keyboard Shortcut (*.aks) format.

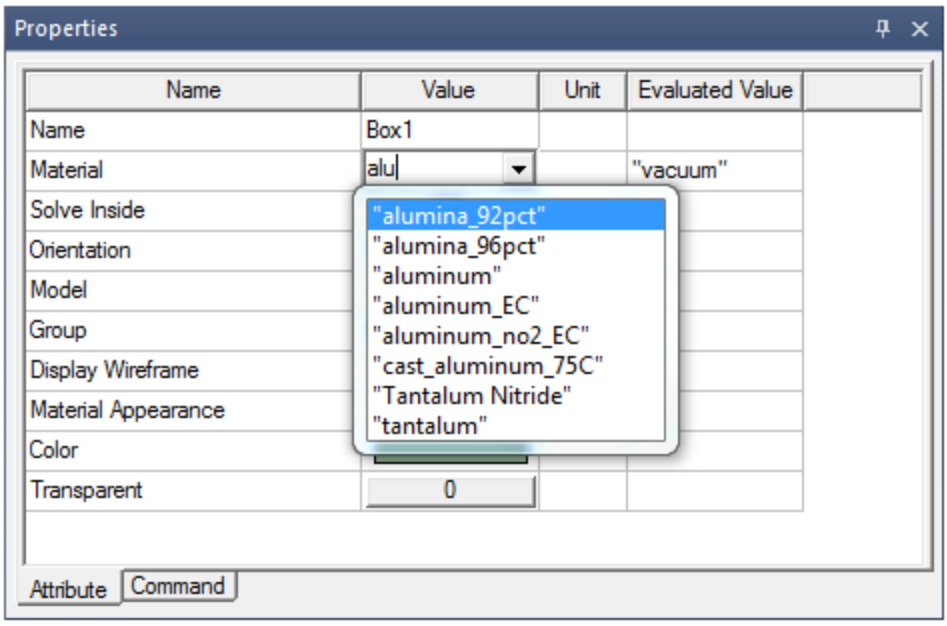
   If you have an existing *.aks file, use the **Load...** button to locate and select the file.

6. Click **OK** to save the current settings, or **Reset to Default**.

Auto-Complete for Variables and Properties in Electronics Desktop

When you edit a properties or variable text field, Electronics Desktop can display possible matches for what you type. This can help if a variable or material name is long. You can save time by selecting a pre-determined match rather than typing out the entire name. The following figure shows an example of auto-complete for material names:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

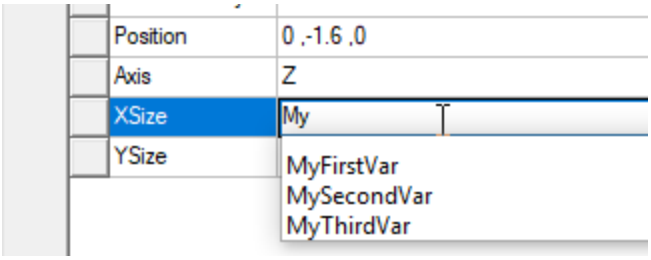


Using Auto-Complete

Certain commonly used text fields have auto-complete configured. When you start typing in these cases, matches display in a list below the text field. If there is no matching text, then no list displays. The list is automatically sized, but you can resize it. Electronics Desktop remembers the new size when doing additional auto-complete matching in the same text field. Switching to a different text field resets the sizing.

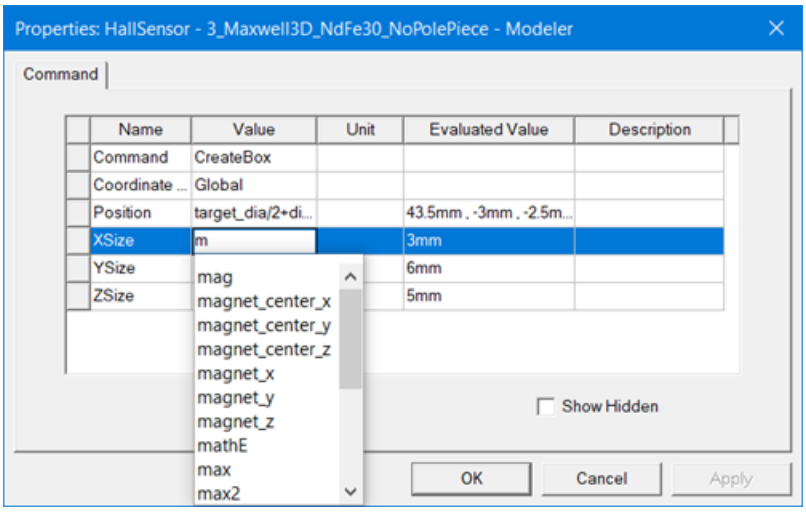
Hitting **Tab** or **Enter** accepts the current selection for auto-complete, which replaces the text typed with the full auto-complete match and hides the list. Otherwise, hitting the escape key **Esc** hides the auto-complete list. Typing more letters causes auto-complete matching to resume.

Where you select design properties, auto-complete works with expressions for the values, providing matches for the names of constants, intrinsic variables, functions, project variables (prefixed with “$”), and design properties. When project variables are displayed (that is, you select the project in the Project tree), auto-complete for value expressions also works, except without matching design properties. The variable auto-complete list does not include Separator Variables or Hidden variables since these could cause invalid formulas.



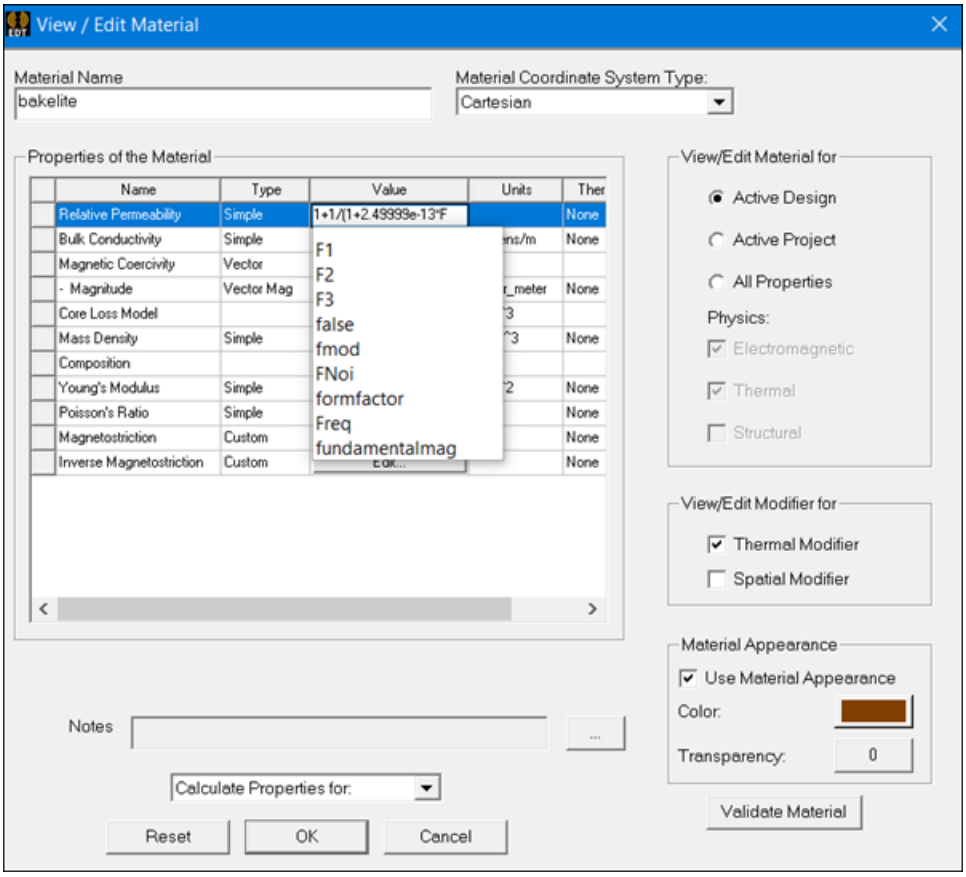
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties of other items may also have auto-complete configured to work with value expressions (for example, properties of a CreateBox command or of a circuit component). The following shows a design property with auto-complete matching:



Auto-complete also works with value expressions for material properties:

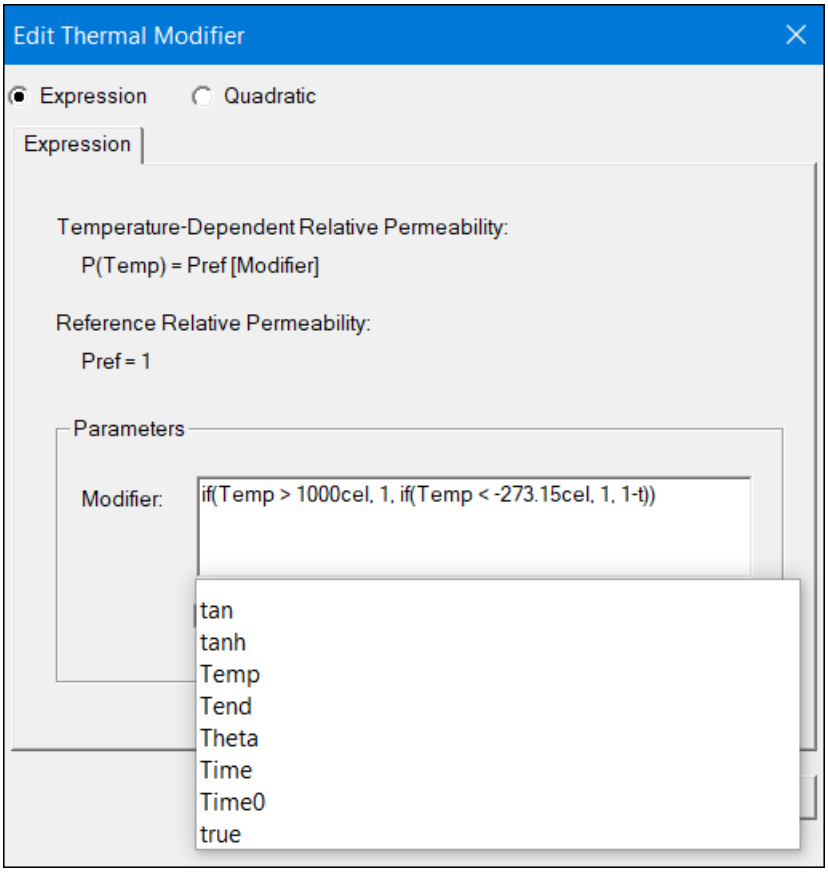
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note the scroll bar, indicating that there are more matches than those currently displayed.

Auto complete also works with thermal modifier expressions for Materials:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using the Password Manager to Control Access to Resources

Ansys Electronics Desktop lets you specify library resources that require password access, and encryption of those resources. For convenience, the same password can apply to multiple resources. To access the **Password Manager,** click **Tools** > **Password Manager**. This displays the **Password Manager** window.

To Specify a New Password Protected Resource

1. Click **Tools** > **Password Manager**.
2. Click **New**.

   This opens the **New Encrypted Resource** dialog box.

3. Specify the name of the resource that you want to protect and click **OK**.

   This displays the **Enter Passwords** window, which contains radio buttons to let you:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter Password and confirm for **Full Access** or for **Execute Only Access**.
- **Use Ansys Password** (for execute only). This option does not require you to enter a password, but still encrypts the library.

4. Once you have selected a radio button and specified any necessary passwords correctly, click **OK**.

   The **Password Manager** updates to show the resource.

To Encrypt a Resource

1. Click **Tools** > **Password Manager**.
2. The **Password Manager** window appears.
3. In the Password Manager, select an existing resource to highlight it.
4. Click **Encrypt File**.

   A file browser window appears.

5. Use the drop-down menu to select the appropriate **Files of Type** filter.

   Any existing resources in the selected directory will appear.

6. When you have selected the appropriate resource, click **OK**.

   This encrypts the resource.

> **Note:**
>
> The **Expire** resource on option lets you select the date when the password expires for the relevant resource.

Running Ansys Electronics Desktop from a Command Line

Ansys Electronics Desktop includes line arguments that can be included when launching from a command line or terminal prompt. All command-line arguments are case-insensitive. The commands associated batch options can also be used with a Job Management Interface for submitting jobs to Ansys or RSM and other supported schedulers.

Command-line Syntax

```
ansysedt <options> <run command(s)> <project file/script file>
```

It is good practice to put quotation marks around the path to the solver executable, and around the full path to the project. This ensures that spaces in the path or project will not be an issue. The same is true of the design name, if there are spaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Commands

The following command line run commands are available for Ansys Electronics Desktop. Of the commands (BatchSave, BatchSolve, BatchExtract, RunScript, RunScriptandExit), one or none must be specified as arguments after the solver executable. When none is specified, you may specify a project or archive to open when Electronics Desktop launches, and can only use the **-Iconic** and **-Help** options. The commands are further described below:

- **-BatchSave** [options] **<***file or folder specifier***>**

  Saves a named project or folder containing one or more project files or folders to the current version. You can run this command with the **-Iconic**, **-Logfile**, and **-ng** options.

  If no migration is necessary (*e.g.* a project containing a circuit design): The original file will be overwritten with the new version, and no backup will be made.

  Special behavior when opening backed up files from the user interface: Files located in the AnsysEM_Backup directory would be the original non-migrated files that were backed up during an earlier –batchsave or open. Opening a file located in the AnsysEM_Backup directory via the open file dialog box will invoke the Parasolid migration, and the migrated file will be saved in the parent folder of the AnsysEM_Backup folder with a unique name so as not to overwrite any existing files.

  If a ***file*** is given, **-batchsave** will run on the specified file.

  If a ***folder*** is given, **-batchsave** will run on all files underneath the specified folder.

  .aedt, .aedtz, and .a3dcomp files will be processed.

  Allowed options:

  norecurse: only process files in top level of input folder

  forcearchive: project files are converted, then saved as archive files.

  previewonly: don't perform actual conversion, instead write preview of filenames to batchlog.

  outputfolder=<outputfolder>: save converted files to <outputfolder>, preserving subfolder structure.

- **-BatchSolve <***project file name***>**

  By default, this run command solves all adaptive setups, Optimetrics setups, and sweeps found in the project file. You can run this command with the **-Iconic**, **-Logfile**, **-ng**, and **-WaitForLicense** options. If parallel solve is possible, you can use the **-Distribute** option.

  If you wish to specify which setups **-BatchSolve** completes, you can use additional parameters:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **[*designName*]** – batch solve all setups for the specified design in the project file.
- **[*designName*]:Nominal** – batch solve all nominal setups for the specified design in the project file.
- **[*designName*]:Optimetrics** – batch solve all Optimetrics setups for the specified design in the project file.
- **[*designName*]:[*Nominal/Optimetrics*]:[*SetupName*]** – batch solve the specified Nominal or Optimetrics setup in the specified design.

If you wish to specify whether **-BatchSolve** setups are completed locally or remotely, you can use the following options:

- **-Local** – performs the **-batchsolve** on the local machine.
- **-Remote -machineList** – performs the **-batchsolve** on a remote machine. The <*machineList*> should provide a single hostname.
- **-Distributed -machineList** – performs a distributed **-batchsolve** using a specified machine list.

  The **-machineList** parameter for a **-Distributed** setup can be formatted three ways:

  - **-MachineList list= "<*machine1*>, <*machine2*>, ..."** – machine names (either by IP address or hostname) are separated by commas. If the list contains any whitespace, it must be enclosed in quotation marks. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the list. That is, if host1 appears in the list once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

    list= accepts the following additional modifiers:

    <
    MachineName
    >:<*TasksOnMachine***>:<***CoresOnMachine***>:<***GPUsOnMachine***>**

    Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

    Example:

    ```
    list="Orion:4:8:90%:1, Aries:3:12, Pluto:6:12"
    ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList file= "<***machineListFilepath***>"** – machine names (either by IP address or hostname) are listed in a file (one per line), and you specify the filepath. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the file. That is, if host1 appears in the file once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

  file= accepts the following additional modifiers, in the file itself:

  <
  MachineName
  >:<*TasksOnMachine***>:<***CoresOnMachine***>:<***GPUsOnMachine***>**

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

  Example:

  ```
  "Orion:4:8:90%:1",
  "Aries:3:12",
  "Pluto:6:12",
  ```

  > **Note:**
  >
  > Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList num= "<***numberofDistributedEngines***>"** – This format is used when a scheduler (such as LSF, PBS, SGE or HPC) is used to manage the jobs sent to a cluster of hosts. In a scheduler environment, you can specify the number tasks for distributed processing. In this case, you do not specify the machine names after the flag because the names are provided by the scheduler. For example, in the Windows HPC environment, you can write the number of tasks as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-MachineList num=4
```

Distributed setups can also take the following optional arguments. When these are not present, the behavior defaults to single-level distributed solutions with no change in order of precedence among possible distribution types.

The arguments are:

- **includeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If included distribution types are specified, only the listed distribution types are enabled. If default is specified, the default set of enabled distribution types is used. To see valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  ```
  "includeTypes=Frequencies,Mesh Assembly"
  ```
- **excludeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If excluded distributed types are specified, all distribution types except those listed will be enabled. If default is specified, the default set of enabled distribution types is used. To see a valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  ```
  "excludeTypes=Frequencies,Mesh Assembly"
  ```
- **maxLevels= <**1 | 2**>** – the maximum number of levels of job distribution (the current maximum is 2). See: Selecting Optimal Configurations for Distributed Analysis.

  > **Note:**
  >
  > Two-level distribution is not currently supported for Mechanical designs. Therefore, set maxLevels = 1 for *Mechanical – Modal* and *Mechanical – Thermal* solutions.

- **numLevel1=** – when two-level distribution is selected (**maxLevels=2**), this specifies the number of level 1 tasks.

-Auto [**NumDistributedVariations=<***num***>**]

This flag enables automatic HPC settings and must be used with one of the following options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-machinelist list=**<*machine list*>, with tasks for each machine set to -1
- **-machinelist numcores=**<*num*>, under a scheduler

All design types being solved must support **-auto** or the solve will be aborted.

The NumDistributedVariations option can be used to specify the number of optimetrics variations to solve simultaneously. The default is to solve optimetrics variations sequentially.

Arguments with **-auto** in a scheduler environment:

- **numcores=**<*total number of cores*>

  Total number of cores, and can be used only with -auto and in a scheduler environment.

- **file="**<*tmachine list file path*>**"**

  The specified file can contain line delimited machine specifiers as described above.

- **num=**<*tnum distributed tasks*>

  This is the total number of tasks and used only in a scheduler environment.

- **numgpus=**<*tnumber of GPUs to use*>

  This is the total number of GPUs and used only in a scheduler environment. numgpus must be combined with either **num=** or **numcores=**.

  You can also specify how a **-BatchSolve** distributes Optimetrics variations:

- **-auto** – Without additional parameters, the batch log file will specify that Optimetrics variations be solved sequentially. If -auto is specified with a machine list, the number of tasks for each machine must be -1.
- **-auto NumDistributedVariations=<***num***>** – You can specify an integer value greater than 1. This is the number of variations that will be solved in parallel.

- **-BatchExtract <***BatchExtract script file name***> <***project file name***>** – allows the following commands to be executed non-graphically via script and without checking out any GUI licenses:
  - ExportProfile
  - ExportConvergence
  - ExportMeshStats
  - ExportNetworkData
  - ExportNMFData
  - ExportEigenmodes
  - ExportTransientData

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Update Reports
- ExportToFile

A project file *must* be specified when **-BatchExtract** is used. Commands in the script file will only be executed in the specified project.

> **Important:**
>
> - **-ng** must be used with **-BatchExtract**, or it will fail.
> - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.

- **-RunScript <***script file name***>** – runs the specified script. You can use the **-ScriptArgs** option to add one or more arguments to this command, and the **-Iconic** option.
- **-RunScriptAndExit <***script file name***>** – runs the specified script and exits Electronics Desktop. You can use the **-ScriptArgs** option to add one or more arguments to this command. You can also use **-Iconic**, **-Logfile**, and **-WaitForLicense**.

> **Note:**
>
> **-BatchSolve <***DesignName***>** is mutually exclusive with **-RunScriptAndExit <***ScriptName***>**.

- **-monitor** – during non-graphical analysis, you can monitor progress and messages. Progress, warning and info messages are logged to the standard output stream. Error and fatal messages are logged to the standard error stream. Schedulers intercept these streams and provide commands for display of this output. See individual scheduler documentation for specifics.
- **-grpcsrv** – For external cpython scripting support. If not specified with **ansysedt.exe**, the Start GRPV server listens on a port in the default range 50051:51051. If specified, Start GRPV server listening on the the specified port number. An error is issued if the port is already used. For example:

```
ansysedt.exe // Start GRPV server listening on a port in
the default range 50051:51051.
ansysedt.exe -grpcsrv portnumber // Start GRPV server
listening on the the port number. error if the port
already used.
ansysedt.exe -grpcsrv 50051:50150 // Start GRPV server
listening on a port range 50051:50150.
ansysedt.exe -grpcsrv 50051:100 // Start GRPV server
listening on a port range 50051:50151.
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Management from the Command Line

- **-showmonitorjob** – Launch the monitor job dialog.
- **-showsubmitjob** – Launch the submit job dialog.
- **-showselectscheduler** – Launch the select scheduler dialog.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchExtract** operation using paths to a script file and a project file:

> **Note:** The example is for an HFSS project. The path, project and design names will vary depending on what the user specifies.

```
ansysedt -ng -batchextract exportToFile.py "C:\Program Files\ANSYS
Inc\v251\AnsysEM\Examples\ElectronicsDesktop\HFSS\RF
Microwave\OptimTee.aedt"
```

Where exportToFile.py contains:

```
oDesktop.RestoreWindow()
oProject = oDesktop.SetActiveProject("OptimTee")
oDesign = oProject.SetActiveDesign("HFSSDesign1")
oModule = oDesign.GetModule("ReportSetup")
oModule.UpdateReports(["XY Plot 1"])
oModule.ExportToFile("XY Plot 1", "exportToFilePy.csv")
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedt -Iconic -LogFile "H:\Logs\mylog.log" -BatchSolve
myDesign:Nominal "H:\Projects\MyProject.aedt"
```

Specifying Project Files

Specifying a project file opens that project when Electronics Desktop launches. If **-BatchSolve** is set, the project will also be solved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify an archive file instead of a project file. If **-Batchsolve** is set, the project will be automatically restored and solved. Otherwise, you are prompted for a restore location, and the project will be restored and opened.

When a **-Batchsolve** is being performed on an archive file, you may also specify **-archiveoptions**:

- **overwritefiles** – allows non-project/results-extracted files to overwrite existing files.
- **path= <***projectFilepath***>** – extracts the project file and associated files to the specified path. If not specified, the archive will be extracted into the same directory as the archive file.
- **repackageresults** – Add batchsolve results back to archive file.
- **winpath= <***windowsProjectFilepath***>** – specifies the Windows-specific path to the extracted project file. This is used when a batch job is to be run on a Linux system, but monitored on Windows.

Options

The following options can be associated with one or more of the run commands:

- **-autoextract** – exports profile (as text), convergence (as text), and report data (as CSV) for the requested project/design/setup in a batch job. Once the solve is complete, an export directory is created (for example, "Project1.aedtexport" for a project named "Project1.aedt") that contains a sub-directory for each design name. You can also specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file (for import and display in Ensight) for all valid field plots. Export files reside within each design-name directory, and include setup name, design variation, job ID, and problem type, as applicable.

  > **Note:**
  >
  > - The -autoextract option is only valid when used with -BatchSolve
  > - The -autoextract option is automatically added for all Ansys Cloud Direct jobs submitted from Electronics Desktop. There is also an additional "reports" and/or "fieldplots" option that immediately follow "-autoextract". This causes all reports to be exported as CSV files at the end of the batch solve, after the profile and convergence have been exported.
  > - For example, you can specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file for all valid field plots.

- **-batchoptions** – for batch jobs, specifies any of the options in **Tools** > **Options**. See additional information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-batchoptionhelp** – opens a window showing -batchoptions help. The paths shown in this window can be used with batchoptions and the Update Registry Get and Set commands. See: Setting or Removing Option Values in Configuration Files: UpdateRegistry Command.
- **-distribute** – distributes a batch solve to multiple machines. This option must be combined with the **-BatchSolve** run command. See: Distributed Analysis.
- **-help** – opens a window displaying command line options. This can only be used without a run command.
- **-iconic** – runs Electronics Desktop with the window iconified (minimized).
- **-logfile <***filePath***>** – specifies a log file. If none is specified, <project_name>.log will be written to the <project_name>.batchinfo directory.
- **-monitor** – enables batch job output to standard output and standard error streams.
- **-ng** – runs Electronics Desktop in non-graphical mode. This *must* be used with the **-BatchExtract** command.
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.
- **-scriptargs** – used in conjunction with **-RunScript** or **-RunScriptAndExit**, adds arguments to the specified script. You can pass multiple arguments to **-scriptargs** by surrounding the arguments in quotation marks. For example:

  ```
  ansysedt.exe -scriptargs "Design1 Setup1" -RunScriptAndExit
  C:\temp\test.py
  ```

  In Python, the command line parameter following **-scriptargs** is passed without modification as a single string in the ScriptArgument python variable.

  In VBscript, the command line parameter following **-scriptargs** is split into multiple strings and converted to a VBscript collection which is accessible via the **AnsoftScript.Arguments** collection. To access these arguments, for example:

  ```
  msgbox AnsoftScript.Arguments(0) // Returns Design1
  msgbox AnsoftScript.Arguments(1) // Returns Setup1
  ```

  In either case, **Design1** is taken into Electronics Desktop as the first argument, and **Setup1** as the second argument. If you failed to use quotation marks, **Design1** would be taken as the first argument and **Setup1** would not be understood by Electronics Desktop.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software.
- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

You can override the option registry entries via the **-batchoptions** command line. These overrides apply only to the current Desktop session. The registry setting overrides may be specified on the command line, or may be in a file with the file pathname specified on the command line. Batch jobs can be submitted from the command line or through Electronics Desktop's job submission window.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

- LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.
- LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

Note:

-batchoptions is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**, **-BatchSave**, or **-BatchExtract**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, HFSS-3DLayout, Icepak, Q3D, Q2D, Maxwell3D, Maxwell2D, and Mechanical.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> No adaptive meshing occurs for any Icepak or Mechanical solutions. Therefore, after solving, the final mesh will be identical to the starting mesh for these two design types.

This example enables CreateStartingMesh for HFSS, and runs a batch solution of the specified project:

```
ansysedt -batchoptions "'HFSS/CreateStartingMesh'=1" -batchsolve
"D:\projects\MyProject.aedt"
```

See: Additional Examples of -Batchoptions Use.

Export Options Files

The **Tools** > **Options** > **Export Options Files** command writes XML files containing the options settings at all levels to the specified directory. This feature is intended to make it easier for different users to use Ansys Electromagnetics Suite 2025 R1 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Examples and Further Explanations of -batchoptions Use

This section provides further examples and explanations of -batchoptions.

- Example with registry settings specified on the command line
- Example with registry settings specified in a file
- When to Use the -batchoptions Desktop Command Line Option

The following examples use general Desktop and HFSS-specific settings. This feature is available for all desktop products.

- The registry path separator is the slash (/) character.
- Each complete registry key (that is, a registry path and option name) is enclosed in single quotes.
- Registry string values are enclosed in single quotes.
- After the -batchoptions switch, the set of registry keys and values that follows it must be enclosed in double quotes. However, if a batchoptions file is referenced (instead of listing the options directly on the command line), the double quotes are not used around the filename.
- Backslashes in registry key string values must be escaped with another backslash (\\ ), since a backslash by itself is an escape code within strings.

Example with registry settings specified on the command line

```
ansysedt.exe -batchsolve -batchoptions
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=2   (* This option is not
currently available from the Add Batchoption dialog box.)
'Desktop/ProjectDirectory'='C:\\projects\test'" projectname.aedt
```

This command line overrides registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key) and ProjectDirectory (Desktop key) options.

> **Note:**
>
> - Multiple registry settings may appear in a single -batchoptions value, separated by whitespace.
> - The -batchoptions value must be enclosed in double quotes if it contains any whitespace.

Example with registry settings specified in a file

```
ansysedt.exe -batchsolve -batchoptions <filename>
projectname.aedt
```

where the referenced file, <*filename*>, contains:

```
$begin 'Config'
  'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
  'HFSS/NumCoresPerDistributedTask'=2
  'Desktop/ProjectDirectory'='C:/projects/test'
$end 'Config'
```

This command overrides the registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key), NumCoresPerDistributedTask (HFSS key), and ProjectDirectory (Desktop key) options. These overrides apply only to the current Electronics Desktop session.

> **Note:**
>
> - The -batchoptions **<*filename*>** value must be enclosed in double quotes if it contains whitespace
> - The **$begin 'Config'** and **$end 'Config'** lines are required

For additional options you can override from the command line with **-batchoptions**, see:

- For -batchoptions Use: Project Directory and Lib Paths
- For -batchoptions Use: TempDirectory
- For -batchoptions Use: Various Desktop Settings

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When to Use the -batchoptions Desktop Command Line Option

You can set analysis parameters for batch mode jobs using the graphical user interface (GUI). For example, you can set all options for a batch job using the **Add Batch Option** dialog box, which is accessed through the **Submit Job To** dialog box. These parameter settings include the following solver options (several examples, not a complete list):

- HPCLicenseType
- tempdirectory
- Desktop/AutoExtractReports
- Desktop/Settings/ProjectOptions/MumberOfProcessors
- <*design_type*>/DefaultProcessPriority
- <*design_type*>/EnableGPU and/or EnableGPUForEye – Applicable to Circuit Design, EMIT, HFSS, HFSS 3DLayout, and Maxwell 3D (also EnableGPUForSBR for HFSS only)
- <*design_type*>/MPIVendor (either “Intel” or “Microsoft”) – Applicable to HFSS, HFSS 3DLayout, Icepak, Maxwell 2D, Maxwell 3D), and Q3D

For graphical analyses that do **not** use batch mode, you specify the analysis parameters using the GUI (**Tools** > **Options** > **General Options** or **HPC and Analysis Options**). These settings are written to the registry when you exit the Electronics Desktop program. The settings are read from the registry when the application is started. Therefore, when you start the Electronics Desktop application, all settings retain the values from the previous session of the same user on the same machine. If there was not a previous session of the same user on the same machine, then the values are obtained from other registry configuration files or from a default value.

When running a batch analysis, any setting that is not specified using the **-batchoptions** command line option is taken from the registry. This value is typically the setting from the last session of the same user on the same machine. However, the **-batchoptions** command line option allows you to override the parameter with values specified on the command line or in a batchoptions file. The values specified using the **-batchoptions** command line option only apply to the batch job, and do not affect the parameter values stored in the registry.

If important **-batchoptions** values are not specified when running a batch job, the paramaters could be affected by an interactive session running on the same host by the same user. Parameter changes can occur if the user sets an option in the GUI and then exits the program, or if another process that accesses the registry exits. To be sure of the desired batch job outcome, avoid changing options in concurrent interactive sessions or include the desired -batchoptions in the command line.

For -batchoptions Use: Project Directory and Lib Paths

The PersonalLib, syslib and userlib settings are a little different from other settings. If the final directory name is different from what is expected, then PersonalLib, syslib or userlib is appended

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as a final directory. In addition, these settings may come from a different registry value if the registry values shown above are not set.

Registry Key	Default Value	Units or Values	Description
Desktop/ProjectDirectory	subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory where new projects are created
Desktop/PersonalLib	PersonalLib subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory PersonalLib is appended if final directory is not PersonalLib
Desktop/syslib	syslib subdirectory of installation directory	Directory pathname	Directory syslib is appended if final directory is not syslib
Desktop/userlib	userlib subdirectory of installation directory	Directory pathname	Directory userlib is appended if final directory is not userlib

For -batchoptions Use: TempDirectory

Registry Key	Default Value	Units or Values	Description
TempDirectory	Set by installer	-	Directory for temporary files

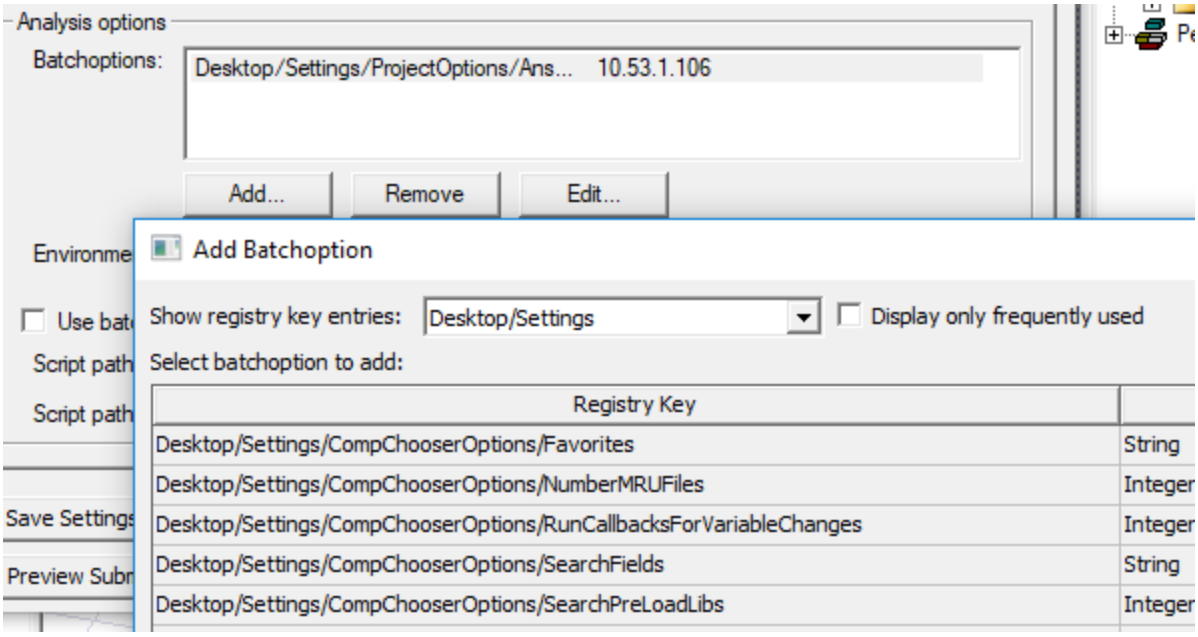
For -batchoptions Use: Various Desktop Settings

Note that most of these options only affect the GUI. To view these options from the *Submit Job To* window:

1. In the *Analysis options* area of the *Submit Job To* window, click **Add...**.
2. In the *Add Batchoption* window that appears, select **Desktop/Settings** from the **Show**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

registry key entries drop-down menu.



Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/AnimationMemory	200	Megabytes (MB)	Stop animations when available memory falls below this value
Desktop/Settings/ProjectOptions/AnsoftCOMPreferredIPAddress (see note below table)	“” (empty string)	IP address (as a string)	IP address used to connect from COM engines to ansysedt.exe

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/AnsysEMPreferredSubnetAddress (see note below table)	“” (empty string)	IP address (as a string)	Subnet used to connect COM engines to ansysedt.exe – allowed formats are: • IPv4 network prefix in CIDR notation Example: 123.123.123.0/24 • IPv4 network prefix with /subnet mask appended Example: 123.123.123.0/255.255.255.0
Desktop/Settings/ProjectOptions/AutoSaveInterval	10	edits	Number of edits to allow between autosaves
Desktop/Settings/ProjectOptions/AutoShowMessageWindow	1 (true)	0 (false) or 1 (true)	Show message window on new messages
Desktop/Settings/ProjectOptions/AutoShowProgressWindow	0 (false)	0 (false) or 1 (true)	Show progress window when starting a simulation
Desktop/Settings/ProjectOptions/DiskLimitForAbort	0	Megabytes (MB)	A warning is issued when available disk space falls below this value
Desktop/Settings/ProjectOptions/DoAutoSave	1 (true)	0 (false) or 1 (true)	Enables autosaves if true
Desktop/Settings/ProjectOptions/DrawStateIconsInProjectTree3	1 (true)	0 (false) or 1 (true)	Change icon when selection does not match active window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/ExpandMessageTreeOnInsert	1 (true)	0 (false) or 1 (true)	Ensure that new messages are visible in the message window tree
Desktop/Settings/ProjectOptions/ExpandOnInsert	0 (false)	0 (false) or 1 (true)	Expand project tree on insert
Desktop/Settings/ProjectOptions/HighlightActiveContextInProjectTree2	1 (true)	0 (false) or 1 (true)	Emphasize active command context (menu and toolbars)
Desktop/Settings/ProjectOptions/SavePreviewImagesInProject File	1 (true)	0 (false) or 1 (true)	Save preview images in project file
Desktop/Settings/ProjectOptions/UpdateReportOnFileOpen	0 (false)	0 (false) or 1 (true)	Update reports on file open

Note:

The preferredIP address and preferred subnet address settings are mutually exclusive. If both are specified to be non-empty strings, then the preferred IP address takes precedence, and the preferred subnet address is ignored. This feature is typically used for cluster environments using batch solves. The setting can be made via batchoptions but can also be done via UpdateRegistry.

Running Ansys Electronics Desktop from a Windows Remote Terminal

When running Mechanical from a remote terminal, there are some performance and behavior issues to consider. These issues are due to the interaction of bandwidth/opengl drivers/remote-terminal-protocol

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Showing axes when interactively drawing objects will slow the performance.
- Remote OpenGL performance will be slower in general. Graphics card and driver quality helps.
- All 3D windows will be closed when you switch from remote PC to a console or from a console to remote. This is to avoid display/opengl instability during the switch.
- Grid will not be turned off while viewing a plot from a remote desktop. The mouse over highlights on 2D plots may appear as not totally overlapping the line color or as thin dotted lines.

Using Windows HPC Commands

HPC Integration allows you to submit jobs directly using Ansys Electromagnetics command line arguments for batch solves. Ansys Electromagnetics products must be accessible from the same directory on all machines. The Ansys Electromagnetics command line syntax is documented here. You must pass in a -distributed flag as part of the Ansys Electromagnetics command line arguments if you want to run a distributed simulation.

For information concerning the supported HPC software, go the the Platform Support webpage and click the links to the following topics:

- *Ansys 2025 R1 - Job Schedulers and Queuing Systems Support*
- *Ansys 2025 R1 - Message Passing Interface Support for Parallel Computing*

Before running a job you must click **Tools** > **Job Management** > **Select Scheduler** and use the dialog box to designate the head node of a cluster. You can then click **Tools** > **Job Management** > **Submit Job** to submit the batch commands for the job.

Customizing Ansys Electronics Desktop with Ansys ACT

With Ansys ACT, you can create custom applications or extensions to customize supported Ansys products, including Electronics Desktop.

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

An ACT guided process extension enables you to leverage both the functionality of Electronics Desktop and the scripting capabilities of the Workbench/AIM framework API. You can manipulate existing features and simulation components, organizing them as needed to produce a custom automated process. A guided process extension is exposed in Electronics Desktop as a wizard that provides step-by-step simulation guidance within the application workflow.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To access ACT functionality in Electronics Desktop, open the ACT Home page by clicking **View** > **ACT Extensions**.

For more information, see ACT Simulation Wizards and Electronics Desktop Wizard in the *Ansys ACT Developer's Guide*.

Installing PyAEDT (Beta)

PyAEDT is a Python library that interacts with the AEDT API to make scripting simpler for the end user. It supports all AEDT 3D products (HFSS, Icepak, Maxwell 3D, and Q3D Extractor), 2D tools, Ansys Mechanical, EMIT, Circuit tools like Nexxim, system simulation tools like Twin Builder, and layout tools like HFSS 3D Layout and EDB. Additionally, it enables the end user to have a CPython interface with AEDT. Its class and method structures simplify operation for the end user, enabling more Pythonic code while reusing information as much as possible across the various APIs.

Documentation for PyAEDT can be found online at: https://aedt.docs.pyansys.com/version/stable/

Installing PyAEDT adds three items to the **Tools** > **Toolkit** > **PersonalLib** menu and to the **Automation** tab:

- **Console** – launches the PyAEDT console.
- **Jupyter Notebook** – launches Jupyter Notebook (a computational notebook) in an internet browser.
- **Run PyAEDT Script** – launches a file browser allowing you to select a Python script to run via PyAEDT.

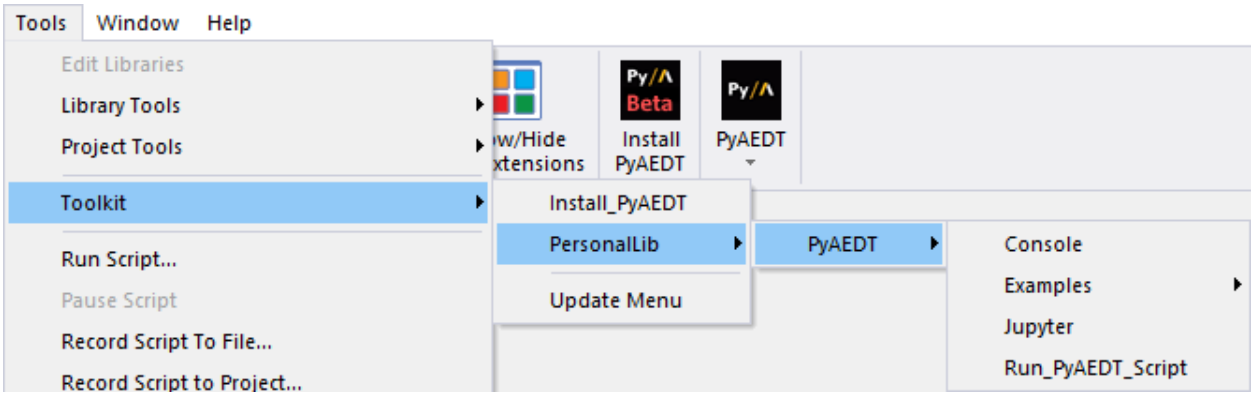
To install PyAEDT:

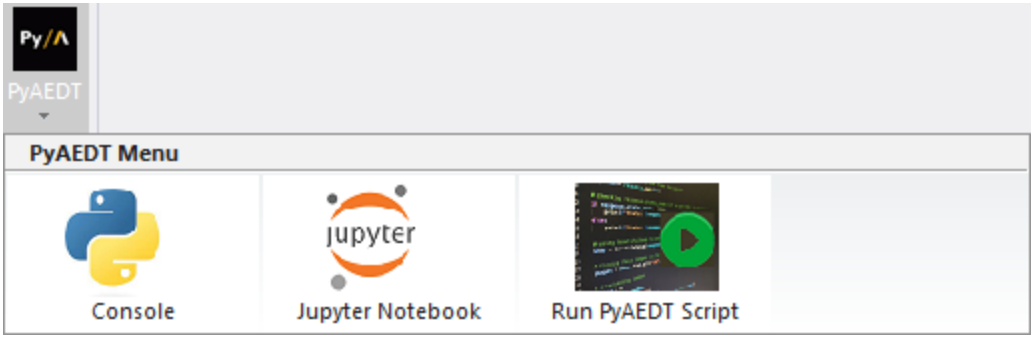
- From the **Automation** tab, click **Install PyAEDT**.

A web browser launches, and takes you to detailed installation instructions.

When installation is complete, the **Tools** > **Toolkit** > **PersonalLib** menu and the **Automation** tab update to display PyAEDT menu options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





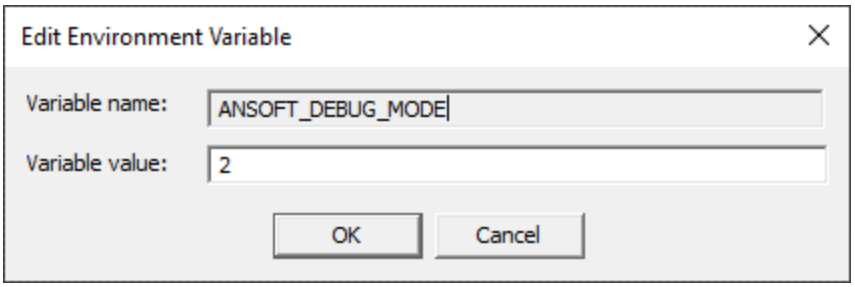
Debug Logging

Electronics Desktop provides error logging capabilities.

To begin logging errors:

1. Click **Tools** > **Debug Logging**.

   The **Edit Environment Variable** window appears, showing the ANSOFT_DEBUG_ MODE variable.



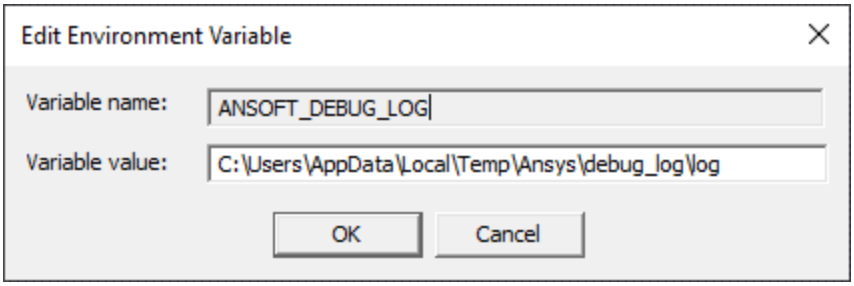
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter a **Variable value**.

   The default value is 2, and should not be changed unless directed by technical support.

3. Click **OK**.

   The **Edit Environment Variable** window updates to display the ANSOFT_DEBUG_LOG variable.



4. Enter a folder path where log files (*.log) will be stored.
5. Click **OK**.

   Errors are logged to the specified folder until you disable logging.

To stop logging errors:

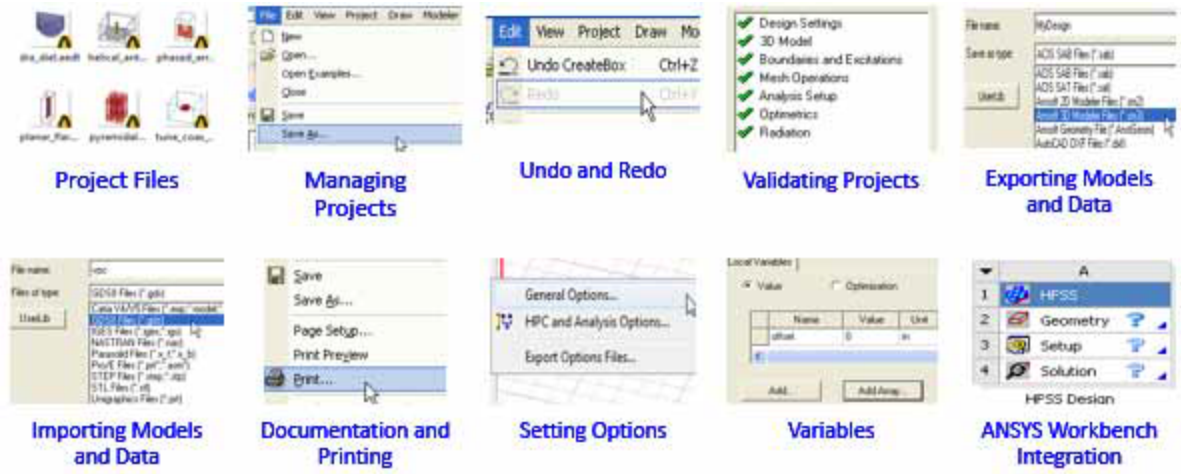
- Click **Tools** > **Stop Debug Logging**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Working with Ansys Electronics Desktop Projects

An Ansys Electronics Desktop project is a folder that includes one or more models, or *designs*. Each design ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information, or a schematic or netlist.



A new project called *Projectn* is automatically created when the software is launched. By option, a design named Design*n* is automatically created for a new project. You can also open a new project by clicking **File** > **New**. In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

> **Note:**
>
> Not all options and capabilities documented in this section apply to all design types.

Ansys Electronics Desktop Files

When you create any project in Ansys Electronics Desktop, it is given an .aedt file extension and stored in the directory you specify. For Legacy projects, the transition is handled as follows:

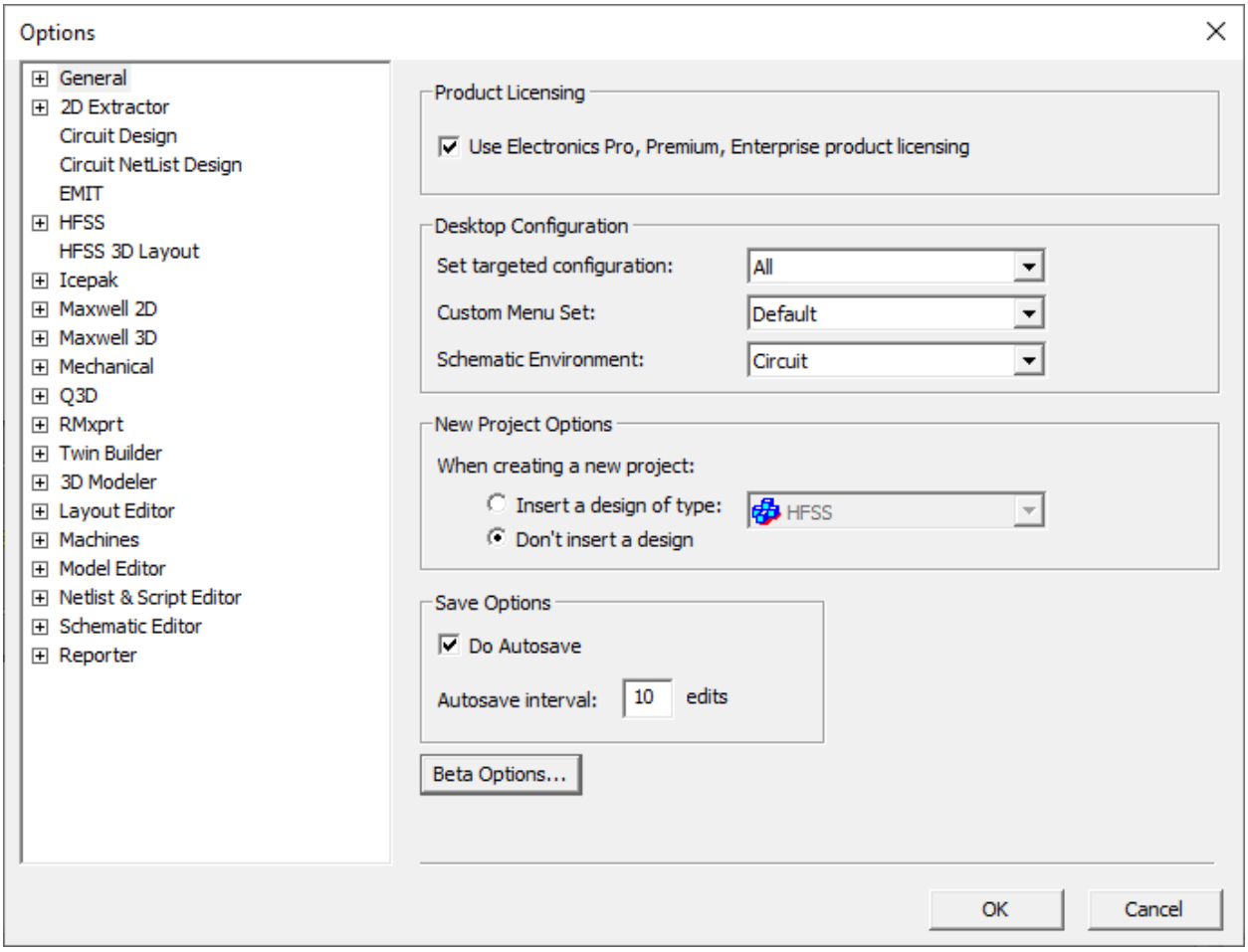
- When opening a legacy project in interactive mode, a dialog box appears, informing you that the legacy file extension is no longer supported and that if you continue, the project will be converted to the new .aedt extension. If you agree, the project and results directory are renamed/moved to the new file extensions immediately. The read continues with the standard code for reading previous version projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Ansys Electronics Desktop 2025R1 version cannot open projects created with 2022R2 or earlier that were using ACIS modeler.
- Attempts to run batch solve or non-graphical with a legacy project return an error. There is no automatic/hidden conversion of file extensions. Note that the existing BatchSave command can be used to convert many projects to the new extension and version.
- Workbench integration has not been modified. It continues to open legacy projects without a warning and copies results from legacy to the new extension.

Setting Project Options

Tools > **Options** > **General Options** opens the **Options** window.



The left pane provides access to the following sets of options:

- General
- 2D Extractor
- Circuit Design

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Circuit Netlist Design
- HFSS
- HFSS 3D Layout
- Icepak
- Maxwell 2D
- Maxwell 3D
- Mechanical
- Q3D
- RMxprt
- Twin Builder
- 3D Modeler
- Layout Editor
- Machines
- Model Editor
- Netlist and Script Editor
- Schematic Editor
- Reporter

Use **Tools** > **HPC and Analysis Options** to specify active configuration per design type, queuing, distributed memory vendor, HPC licensing, select a MPI vendor, and whether to enable GPU for transient solves.

The **Tools** > **Options** > **Export Options Files** command writes xml files containing the Options settings at all levels to the specified directory. The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. The Example Uses for Export Options Features section outlines some use cases enabled by this feature.

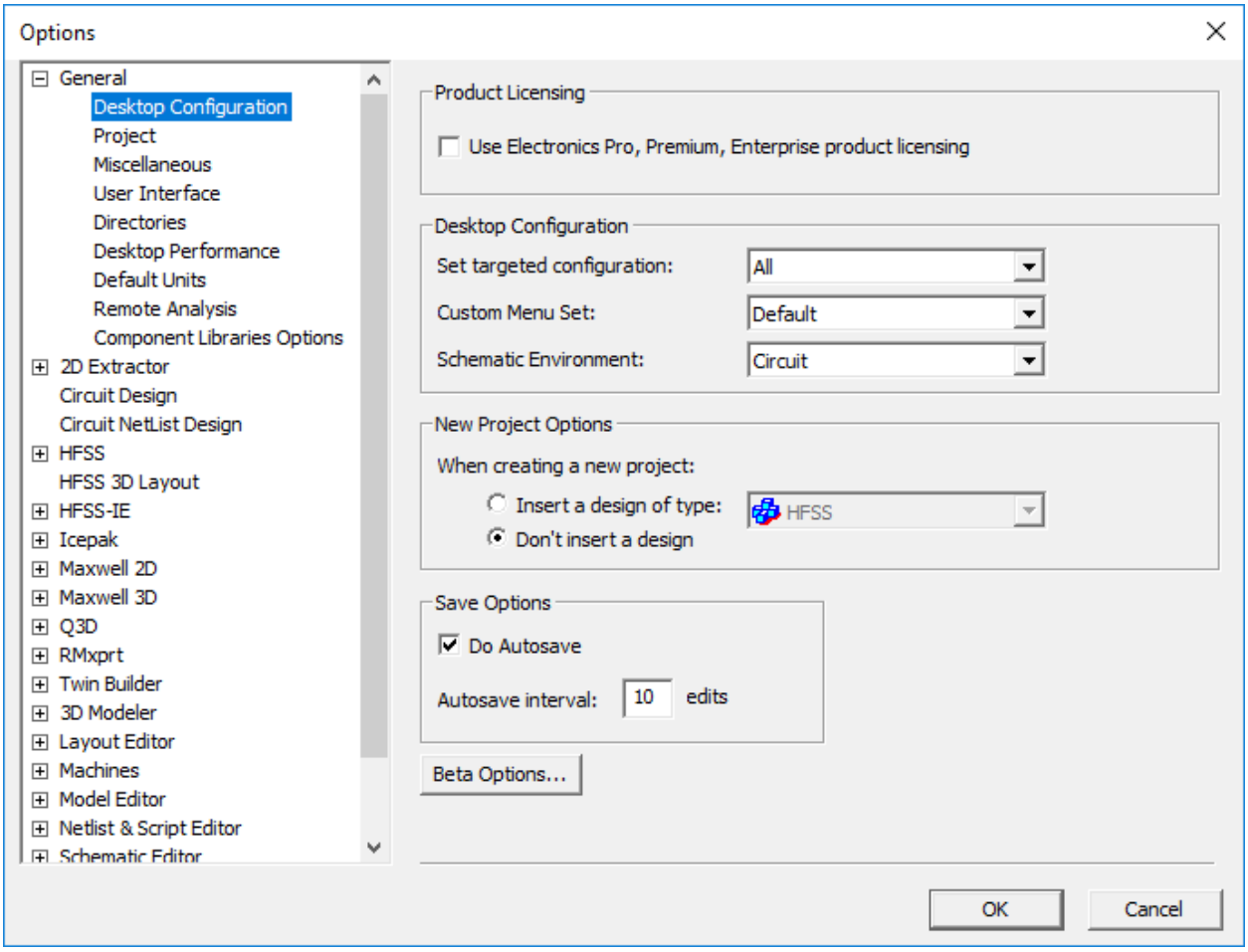
Setting General Options

To set general options in Ansys Electronics Desktop:

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

   The **Options** window opens with the Desktop Configuration options selected by default.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the left pane, select the entries below **General** to display the associated options:

- Desktop Configuration
- Project
- Miscellaneous
- User Interface
- Directories
- Desktop Performance
- Default Units
- Remote Analysis
- Component Libraries Options

2. Click each entry and make the desired selections.

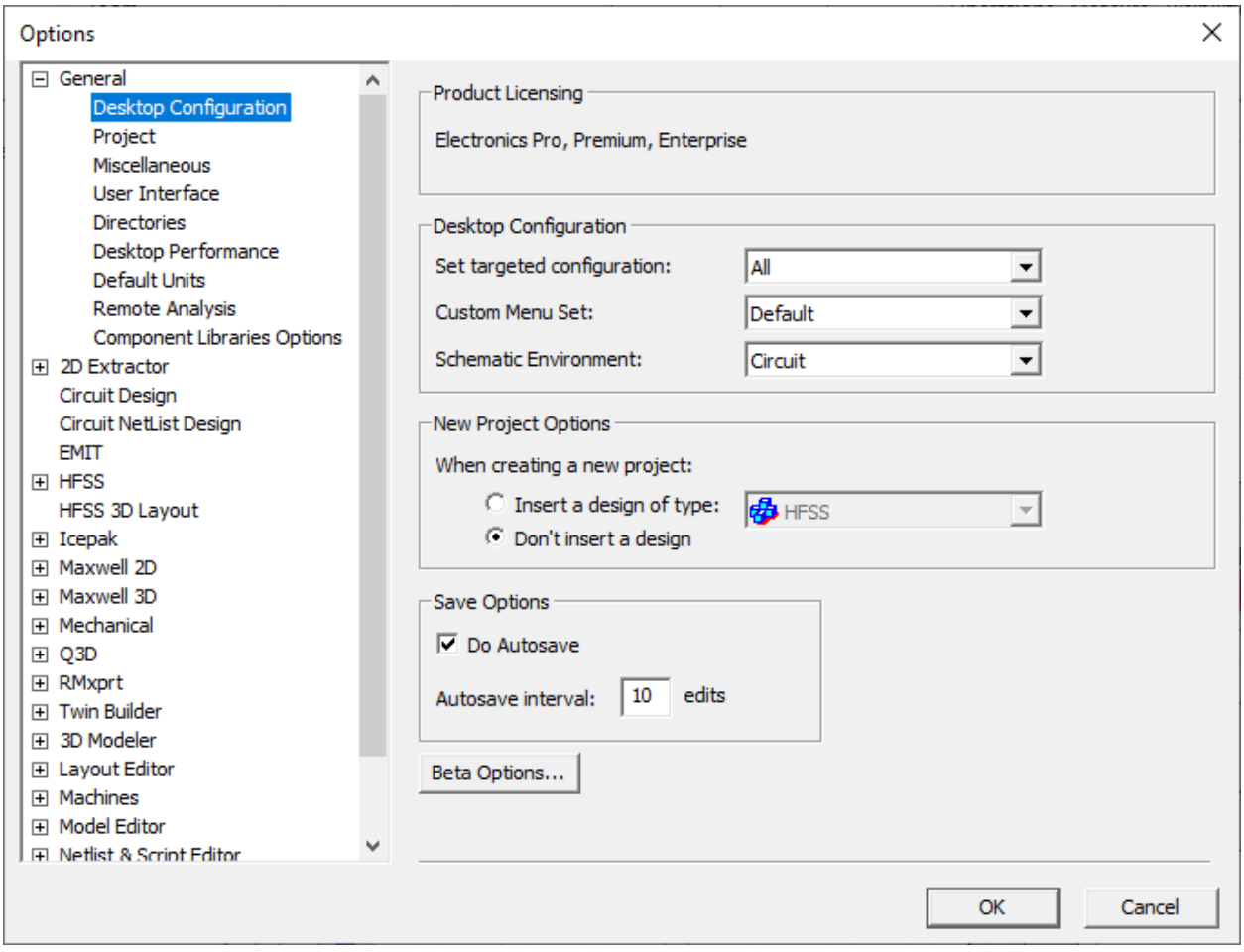
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

> **Note:**
>
> Not all option settings are applied immediately in the current design. Some settings take effect in newly created designs.

General Options: Desktop Configuration

Under General Options, **Desktop Configuration** options allow you to customize Electronics Desktop in a way that suits your work priorities. These include options for menus, new projects, save intervals, and beta features.



Ansys Electronics Desktop now uses Electronics Pro, Premium, Enterprise (PPE) product licensing. Legacy product licensing and DSO are no longer supported.

With PPE, HPC licensing is used to enable all cores, GPUs, and distributed tasks.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more details, see Setting HPC and Analysis Options.

In the **Desktop Configuration** area, options include:

- **Set Targeted Configuration** – Choosing a Targeted Configuration changes the Custom Menu Set and Schematic Environment for your area of focus.

  Choices are:

  - **All** – Default option. Sets the **Custom Menu Set** to **Default**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **EM** – Electromagnetic focus. Sets the **Custom Menu Set** to **EM**, the **Schematic Environment** to **Twin Builder**, and the default design type to **Maxwell 3D**.
  - **RF** – Radio Frequency focus. Sets the **Custom Menu Set to** RF, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **SI** – Signal Integrity focus. Sets the **Custom Menu Set** to **SI**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS 3D Layout**.
  - **Twin Builder** – Sets everything to default to **Twin Builder**.
- **Custom Menu Set** – Changes which menu options are available in Electronics Desktop.

  Choices are:

  - **Default** – All solvers appear on the **Project** menu.
  - **EM** – Only electromagnetics solvers appear on the **Project** menu.
  - **RF** – Only radio frequency solvers appear on the **Project** menu.
  - **RF.0** – All solvers appear on the **Project** menu. The **HFSS RF Setup** menu appears.
  - **SI** – Only signal integrity solvers appear on the **Project** menu.
  - **SI1.0** – Only signal integrity solvers appear on the **Project** menu. The **Import**, **Solution Setup**, **Automation**, and **Definitions** menus appear.
  - **SI2.0** – All solvers appear on the **Project** menu. The **HFSS SI Setup** menu appears.
  - **Twin Builder** – Only Twin Builder appears on the **Project** menu.
- **Schematic Environment** – Select **Circuit**, **Twin Builder**, or **Maxwell**.

In the **New Project Options** area, options include:

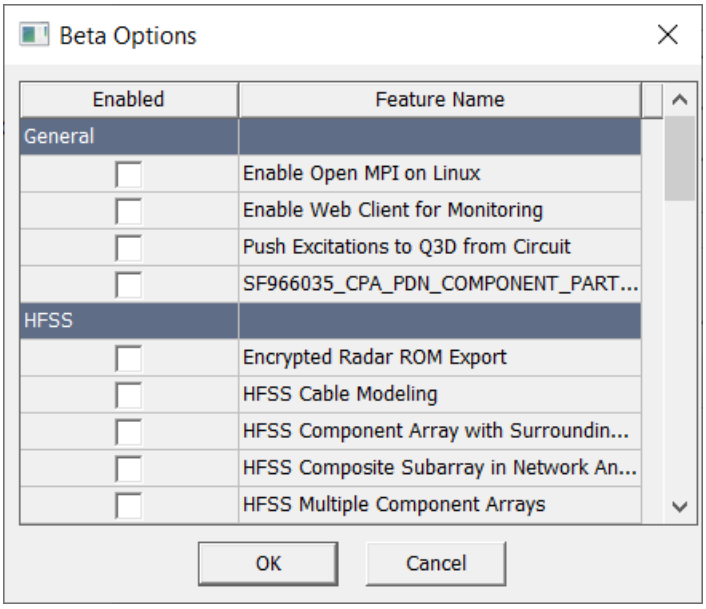
- **When Creating a New Project** – Select whether to insert a design when creating a new project. If you choose to insert a design by default, use the drop-down menu to select the default design type.

In the **Save Options** area, options include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Do Autosave** – Electronics Desktop has autosave enabled by default. Deselect this option to disable it.
- **Autosave Interval** – When **Do Autosave** is enabled, select the number of edits at which Electronics Desktop autosaves.

Click the **Beta Options** button to open a window listing beta options. You may need to scroll or size the window to view all options.

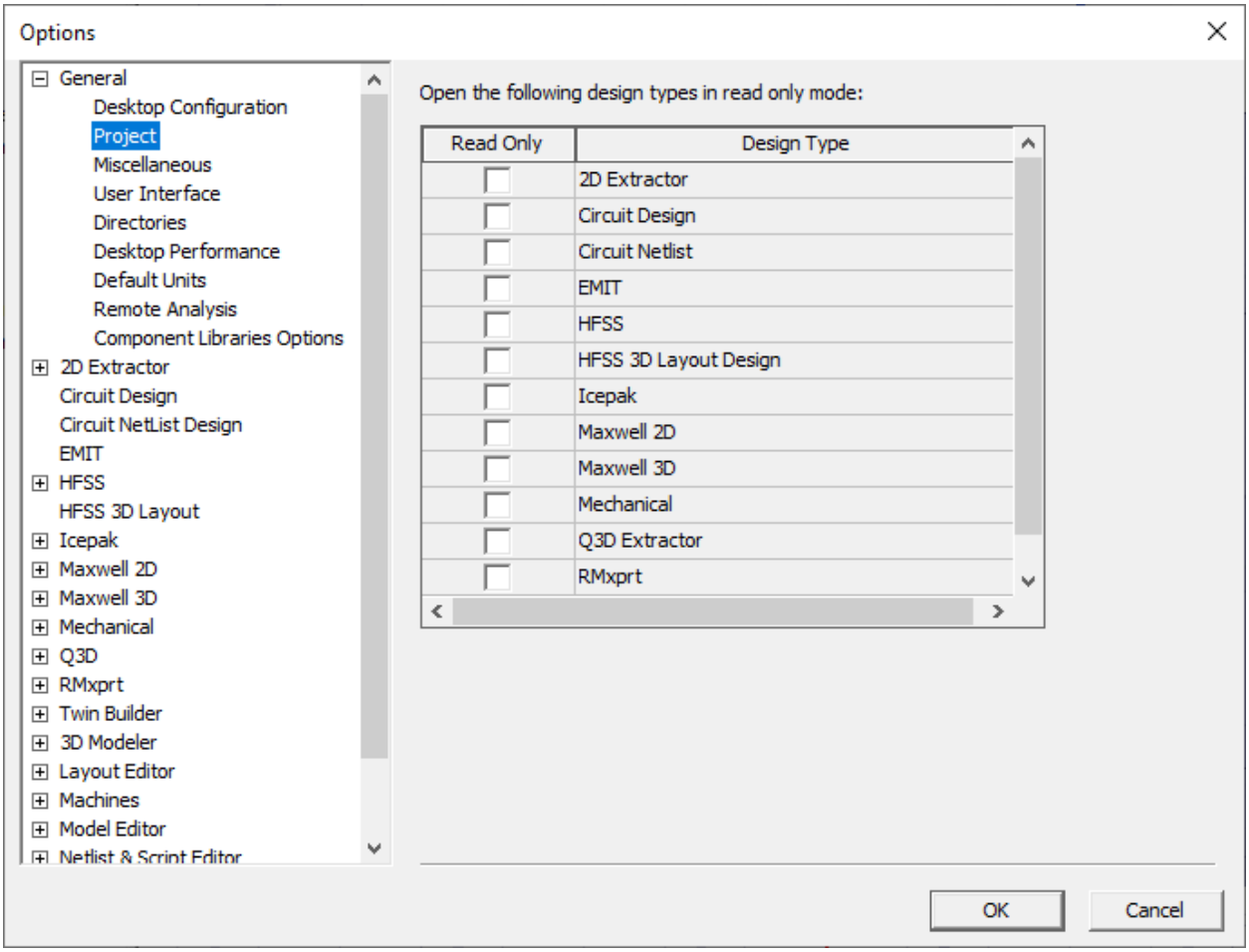


From there, enable or disable options, and click **OK**. You will need to restart to enable your selections.

General Options: Project Options

Under General Options, **Project** options allow you to select design types to open in read-only mode. This can help prevent accidental changes.

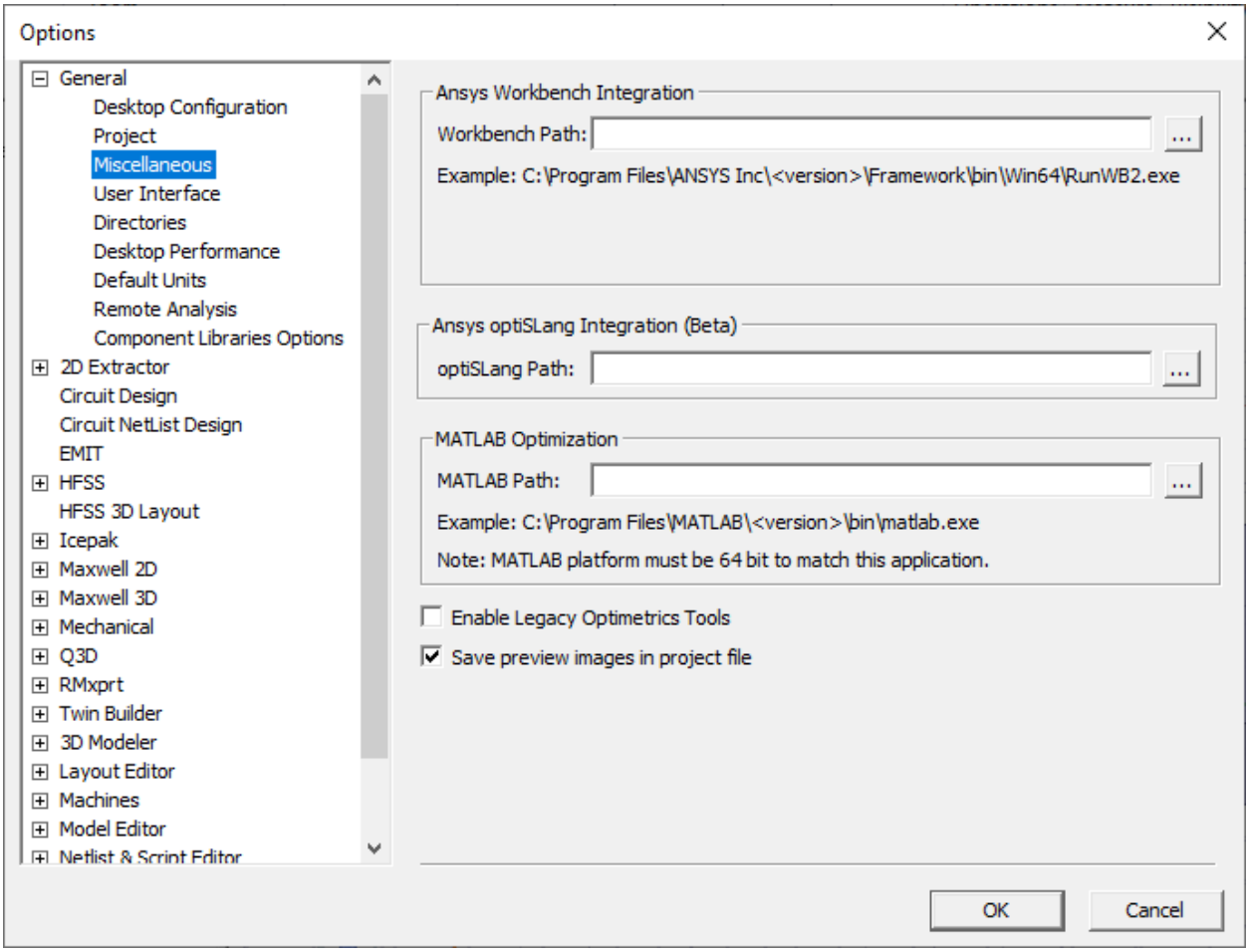
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



General Options: Miscellaneous Options

Under General Options, **Miscellaneous** options allow you to specify paths for Ansys Workbench Integration, optiSLang path, MATLAB Optimization and to enable Legacy Optimetrics Tools.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Workbench Integration allows you to specify the path to your Ansys Workbench installation. This path can be used in Optimetrics for connecting to the Design Xplorer.

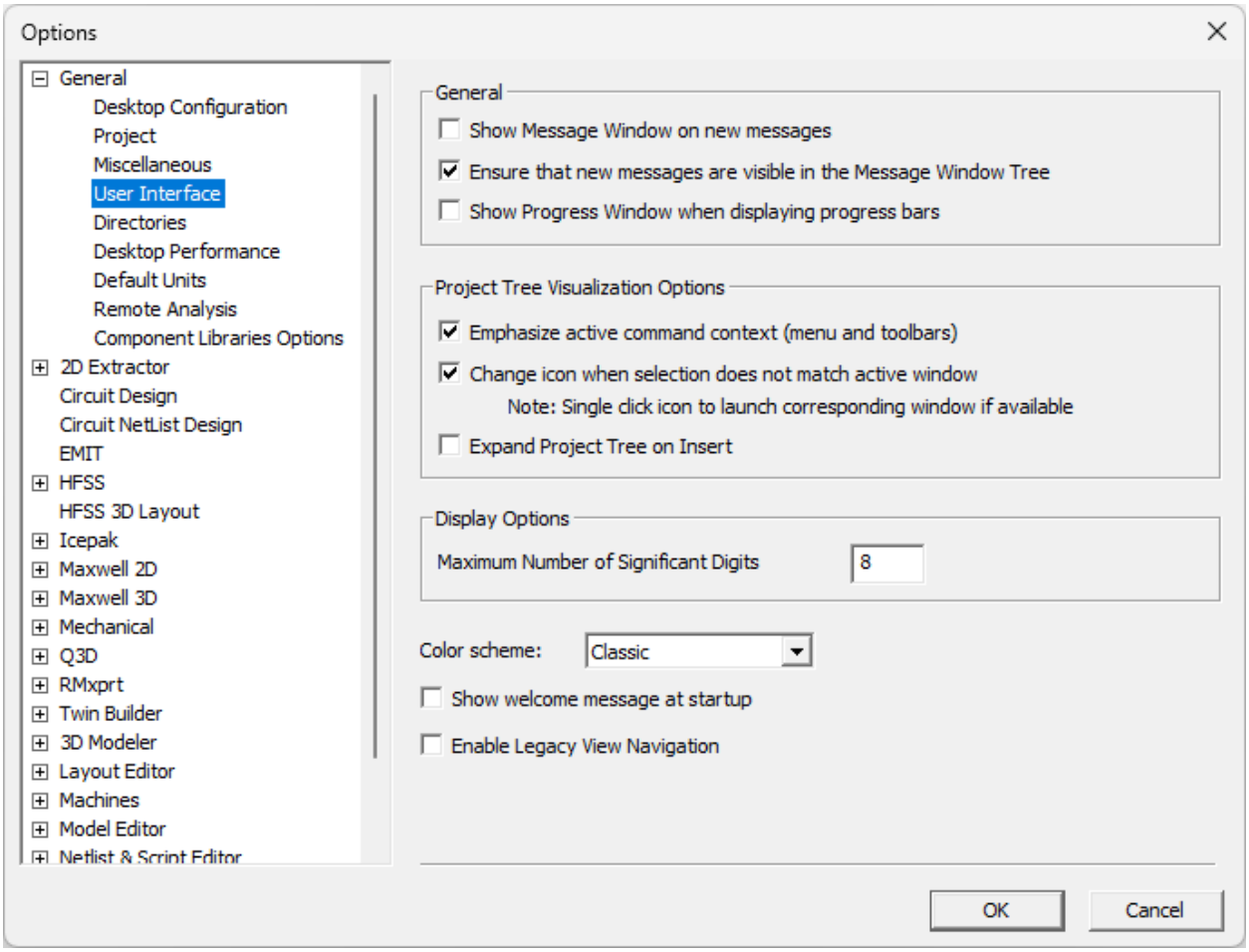
MATLAB Optimization allows you to specify the path to your installation of MATLAB. MATLAB can be used as an Optimizer.

> **Note:**
>
> The platform for MATLAB and Ansys Electronics Desktop must match (for example, a 64 bit version of MATLAB).

General Options: User Interface Options

Under General Options, **User Interface** options allow you to change how Electronics Desktop displays messages, command text, the project tree, welcome messages, and more.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **General** area, options include:

- **Show Message Window on new messages** – when selected, the message window automatically opens if a message arrives.
- **Ensure that new messages are visible in the Message Window Tree** – when selected, the message window expands as needed to display messages.
- **Show Progress Window when displaying progress bars** – when selected, the progress window automatically opens while simulations are in progress.

In the **Project Tree Visualization Options** area, options include:

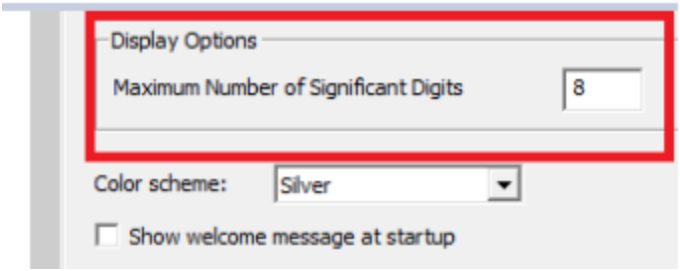
- **Emphasize active command context** – when selected, active elements in the Project Tree display in bold text.
- **Change icon when selection does not match active window** – when selected, a small, window-shaped overlay icon displays in the corner of the selected Project Tree element. This icon changes when the data in the active window is unrelated to the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

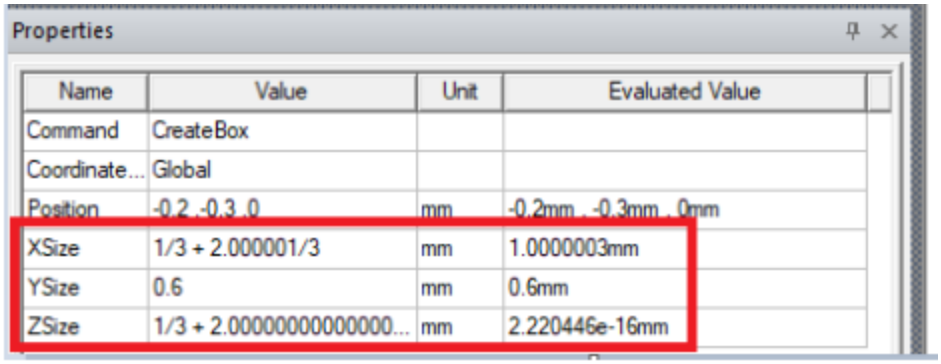
selected project item (data affecting the same model is considered to be related). Clicking the icon opens the window and brings it into focus.

- **Expand Project Tree on Insert** – when selected, the Project Tree automatically expands when you insert a new design.

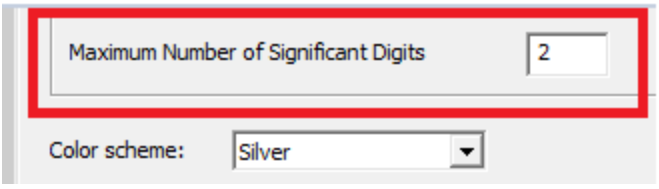
In the **Display Options** area, specify the **Maximum Number of Significant Digits** to display. The default is 8 and the maximum is 20. This affects the digits displayed in the **Solutions** dialog box, evaluated variable values, Animation dialog boxes, Optimetrics, Reports, and so forth. For example, here is the default.



So some evaluated variable values with significant digits appear as follows.

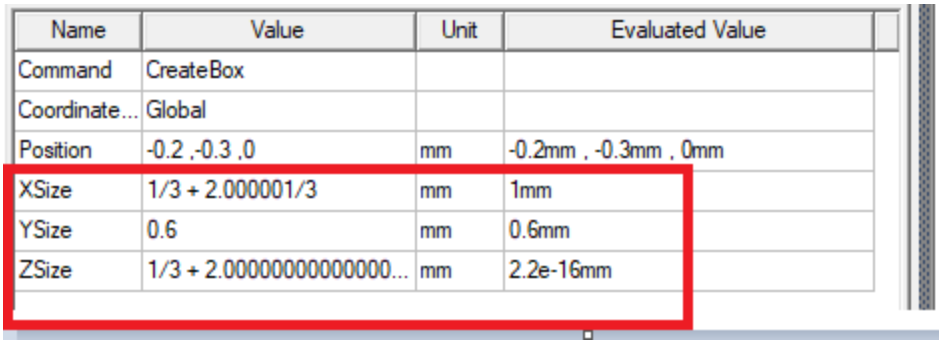


Change Precision = 2. This lets you display near-zero values in the Properties window as 0.



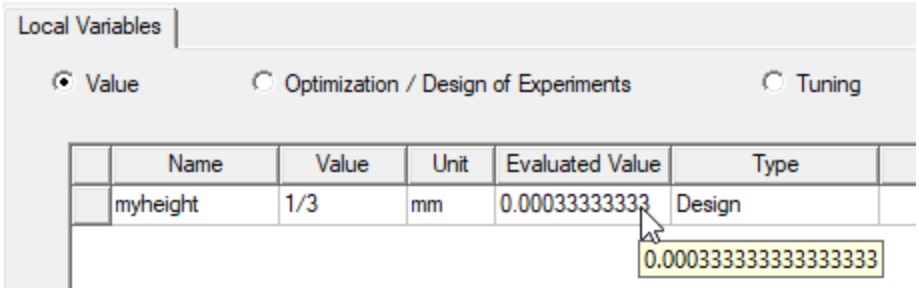
You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

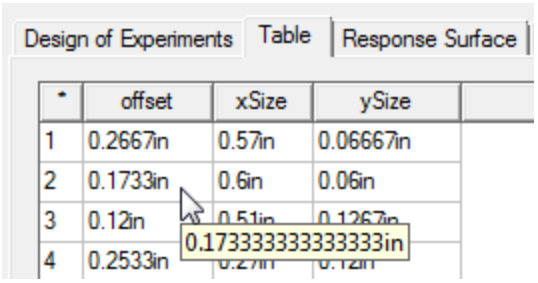


Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.2 ,-0.3 ,0	mm	-0.2mm , -0.3mm , 0mm
XSize	1/3 + 2.000001/3	mm	1mm
YSize	0.6	mm	0.6mm
ZSize	1/3 + 2.00000000000000...	mm	2.2e-16mm

You can still see the full precision values in tooltips by holding a cursor over the displayed value.



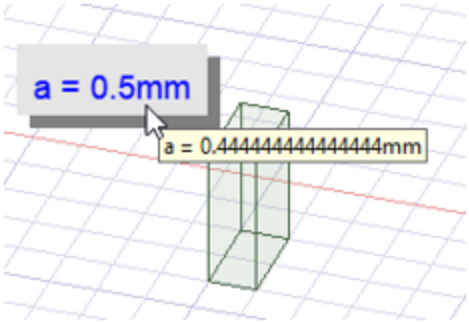
In the case of variable values, if you have assigned more significant digits, you will see these when editing the variable value. In the case of table displays of values, the tooltip display shows all available digits when the mouse pointer is over a result:

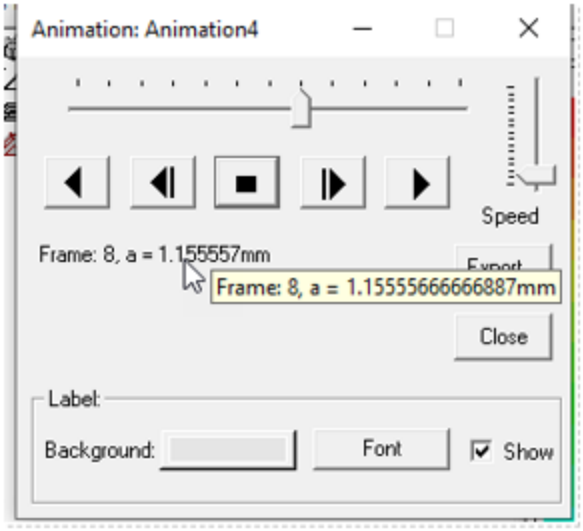


If you set the **Maximum Number of Significant Digits** to a lower value, Change Precision = 2. You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

The tooltip functions to show internal digits in throughout the Ansys Electronics Desktop interface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Select a **Color Scheme**. The choices are **Light (Beta)**, **Dark (Beta)**, or **Classic**: See Choosing a Color Scheme for more information and samples of each scheme.

Select **Show welcome message at startup** if you want to see a welcome message when Electronics Desktop opens.

Enable legacy view navigation enables you to choose between two schemes for view navigation keyboard shortcuts and mouse button assignments, as follows:

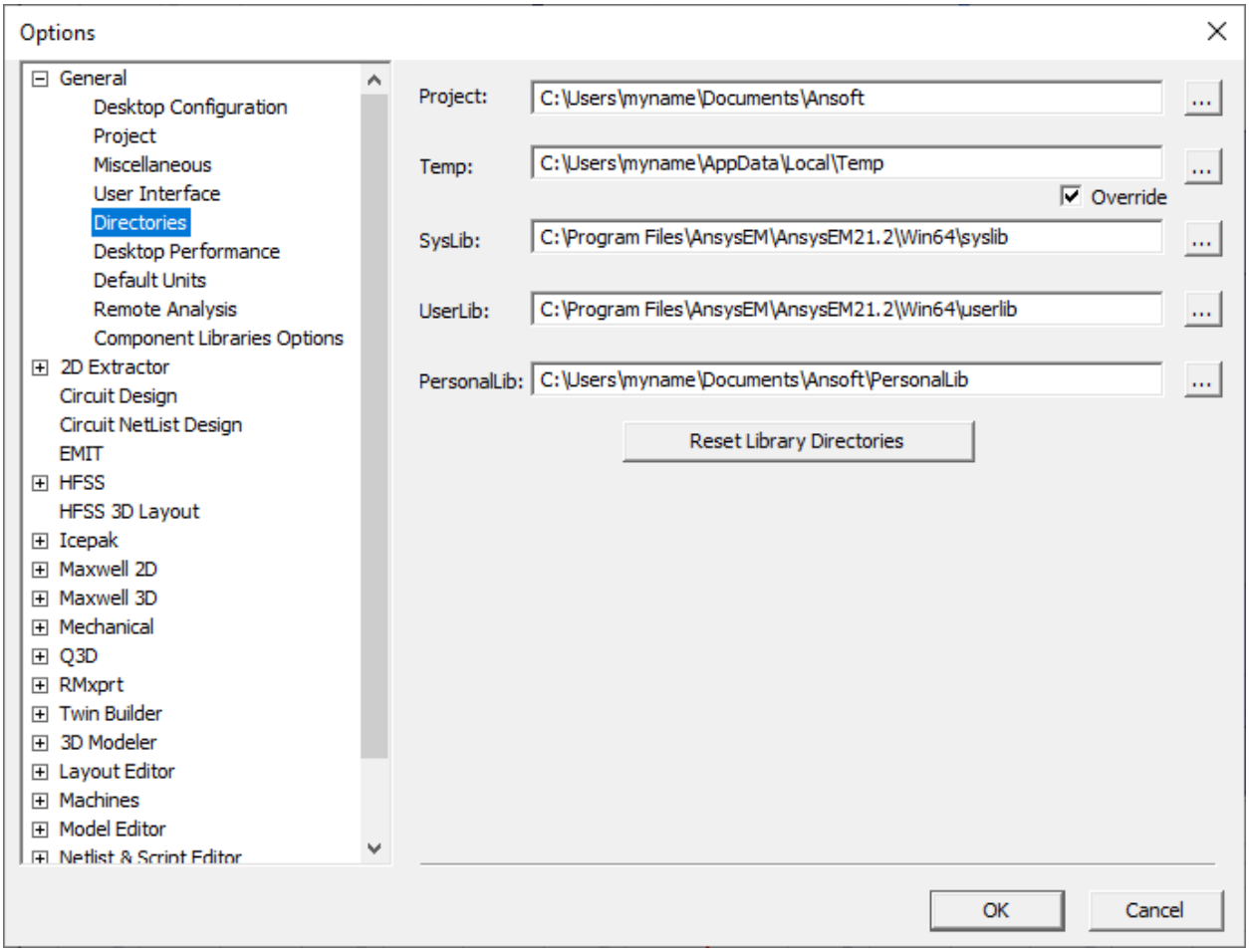
- **Cleared:** Use only the view navigation mouse-button and hotkey assignments introduced in Ansys Electronics Desktop version 2024 R1 and applicable to subsequent versions too. Legacy (2023 R2 and earlier) mouse-button and hotkey assignments will *not* be recognized when this option is cleared.
- **Selected:** Both the legacy (2023 R2 and earlier) and current (2024 R1 and newer) mouse-button and hotkey assignments are supported, and either can be used for view navigation.

See Choosing the View Navigation Options for more information, including a detailed comparison of the two schemes.

General Options: Directories Options

Under General Options, **Directories** options allow you to get the paths for Project, Temp, SysLib, UserLib, and PersonalLib directories.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the **[...]** buttons to browse to paths and click **OK**. You will need to select **Override** to change the Temp directory.

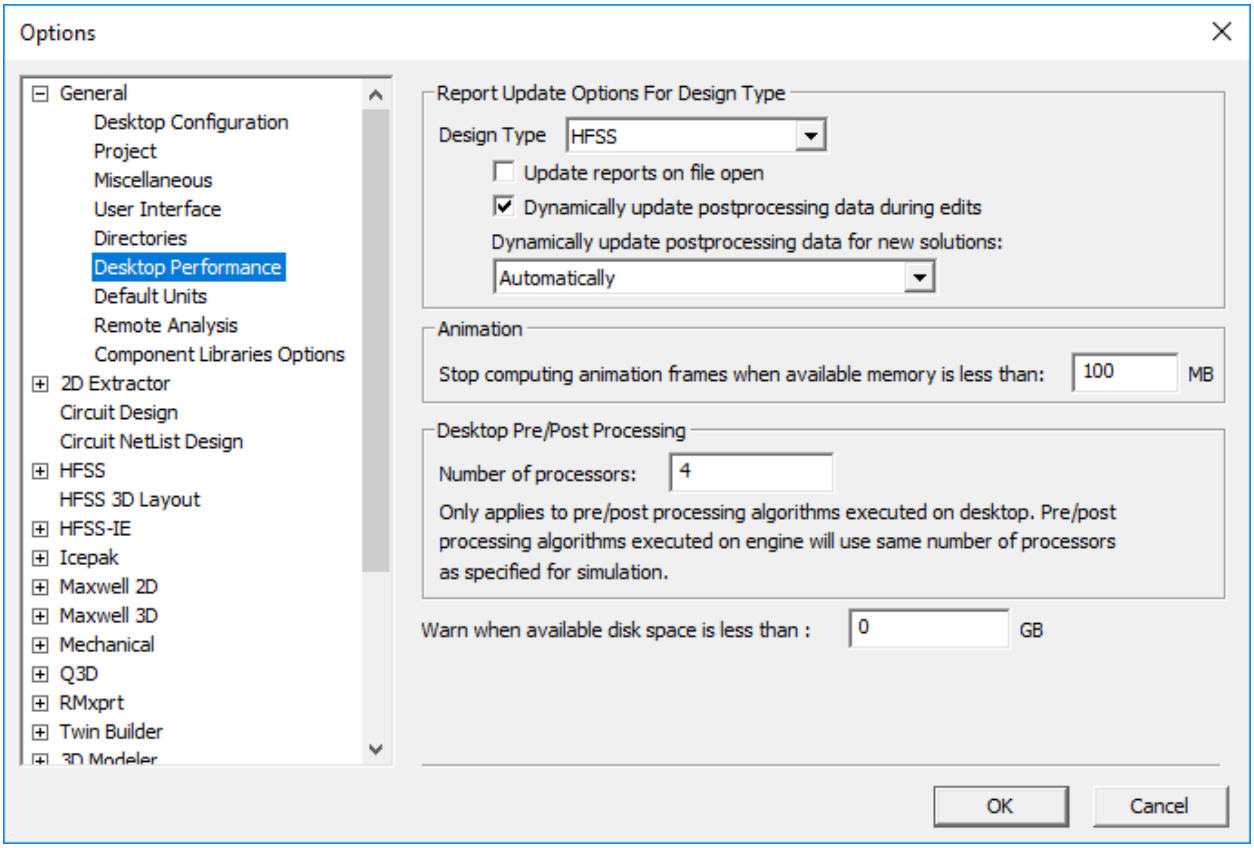
If you need to restore defaults, click **Reset Library Directories**.

> **Note:**
>
> Your changes will be reflected in the User Defined Primitives menu on next startup.

General Options: Desktop Performance

Under General Options, **Desktop Performance** options allow you to change settings for animation, pre/post-processing, disk space warnings, and updating reports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options are set on the **Desktop Performance** panel under **General** in the **Options** window.

In the **Report Update Options for Design Type** area, select a **Design Type**. For each type, you can set the following options:

- **Update Reports on File Open** – when selected, reports are automatically updated whenever an existing file with solution data is opened for viewing/editing.
- **Dynamically Update Postprocessing Data During Edits** – when selected, report plots and overlays update as you edit their parameters.
- **Dynamically Update Postprocessing Data for New Solutions** – Because updating reports during analysis can impact solution time, you can specify how often your reports are updated:
    - **After Each Variation** – updates reports after analysis of each variation has been completed. Used for an Optimetric or parametric analysis.
    - **Automatically** – balances report and field plot updating with solution time.

      For Adaptive Passes, plots update at the end of each solution pass. For Last Adaptive or Transient, plots update at the end of the transient or adaptive solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, reports may be updated after each adaptive pass but field plots will not be updated until the solution is complete.

- **Immediately** – updates reports and plots as soon as data comes from the solver.

  This option will have the greatest impact on overall solution time, but affords the fastest updates to reports and field plots. Caution should be used in selecting this option. Some types of reports and field plots may take a long time to update, especially as mesh size increases.

- **Never** – updates reports only upon manual intervention. This prevents updates from impacting solution time.
- **On Completion** – updates reports once, when the solution completes.

> **Note:**
>
> Updates done on completion are done after the solve has been completed, and the time for that update is not included in the simulation profile.

In the **Animation** area, you can elect to **Stop computing animation frames when available memory is less than** a specified value, in MB (the default is 100MB). This setting is used to prevent problems related to low memory should an animation require large memory allocation.

In the **Desktop Pre/Post Processing** area, you can specify the **Number of Processors**. This option only affects pre- and post-processing (not solve or simulation). Pre-processing algorithms can take advantage of multiple processors for visualization and faceting of 3D models, model validation for 3D products, auto net identification for Q3D, and more. The default value is determined by the number of logical processors on the machine running Electronics Desktop:

- The default core usage per desktop session (UI + solve) is set to 2/3 of the logical processors on the machine.
- The cores for default local config is max{4, 1/3 of the logical processors on the machine}
- The default number of processors for pre/post is min{1, 2/3 of the logical processors - default_cores_for_local_config}
- When the core usage per desktop session cannot be evenly distributed between solve and pre/post, more cores will be assigned to solve.

Example 1:

- logical processors = 24
- cores usage per desktop session = 16
- local hpc set to 4 cores
- processors for pre/post set to 12

Example 2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- logical processors = 20
- cores usage per desktop session = 13
- local hpc set to 4 cores
- processors for pre/post set to 9

Example 3:

- logical processors = 5
- cores usage per desktop session = 3
- local hpc set to 2
- processors for pre/post set to 1

You can also elect to **Warn when available disk space is less than** a specified value, in GB.

General Options: Default Units

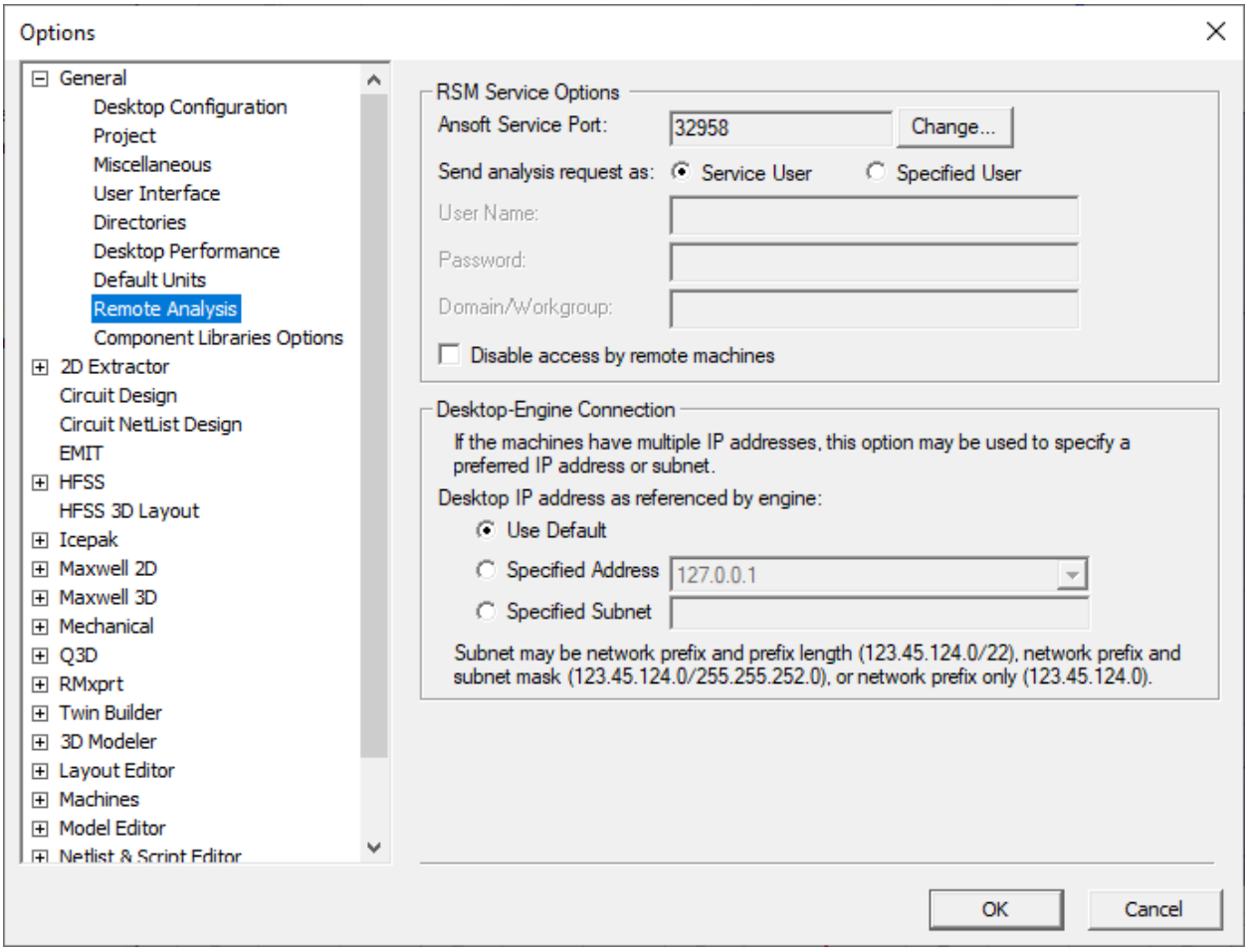
Under General Options, **Default Units** options allow you to set default units for the following:

- Length
- Angle
- Time
- Temperature
- Torque
- Magnetic Induction
- Pressure
- Frequency
- Power
- Voltage
- Current
- Speed
- Mass
- Conductance
- Resistance
- Inductance
- Capacitance
- Force
- Angular Speed
- Magnetic Field Strength

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Options: Remote Analysis Options

Under General Options, **Remote Analysis** options allow you to launch all analyses as a service or specified user rather than as the current user.



In the **RSM Service Options** area, options include:

- **Ansoft Service Port** – Click **Change** to update the port number. Ansys Electromagnetics RSM Service should be running on this port for all distributed machines.
- **Send Analysis Request As** – Select either **Service User** or **Specified User**. Selecting **Specified User** enables the **User Name**, **Password**, and **Domain/Workgroup** fields.

> **Note:**
>
> If any of the remote machines is Linux-based, you must specify the current user.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Disable Access By Remote Machines** – If desired, select to disable access for remote machines.

When multiple IP addresses are available, the **Desktop-Engine Connection** area allows you to specify the preferred IP address for communication:

- **Use Default** – your system's default IP address.
- **Specified Address** – an IP address you specify.
- **Specified Subnet** – a subnet you specify. Subnet may be network prefix and prefix length (123.12.123.0/22), network prefix and subnet mask (123.12.123.0/255.255.252.0), or network prefix only (123.23.123.0).

Changing the Listening Port used by Ansys RSM Service

To change the listening port used by the RSM Service, you must change the ansoftrsmservice.cfg file, as follows:

Specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file (the 'AnsoftCOMDaemon' block). The following example changes the listen port from 32958 to 32957, with these blocks at the beginning of the file:

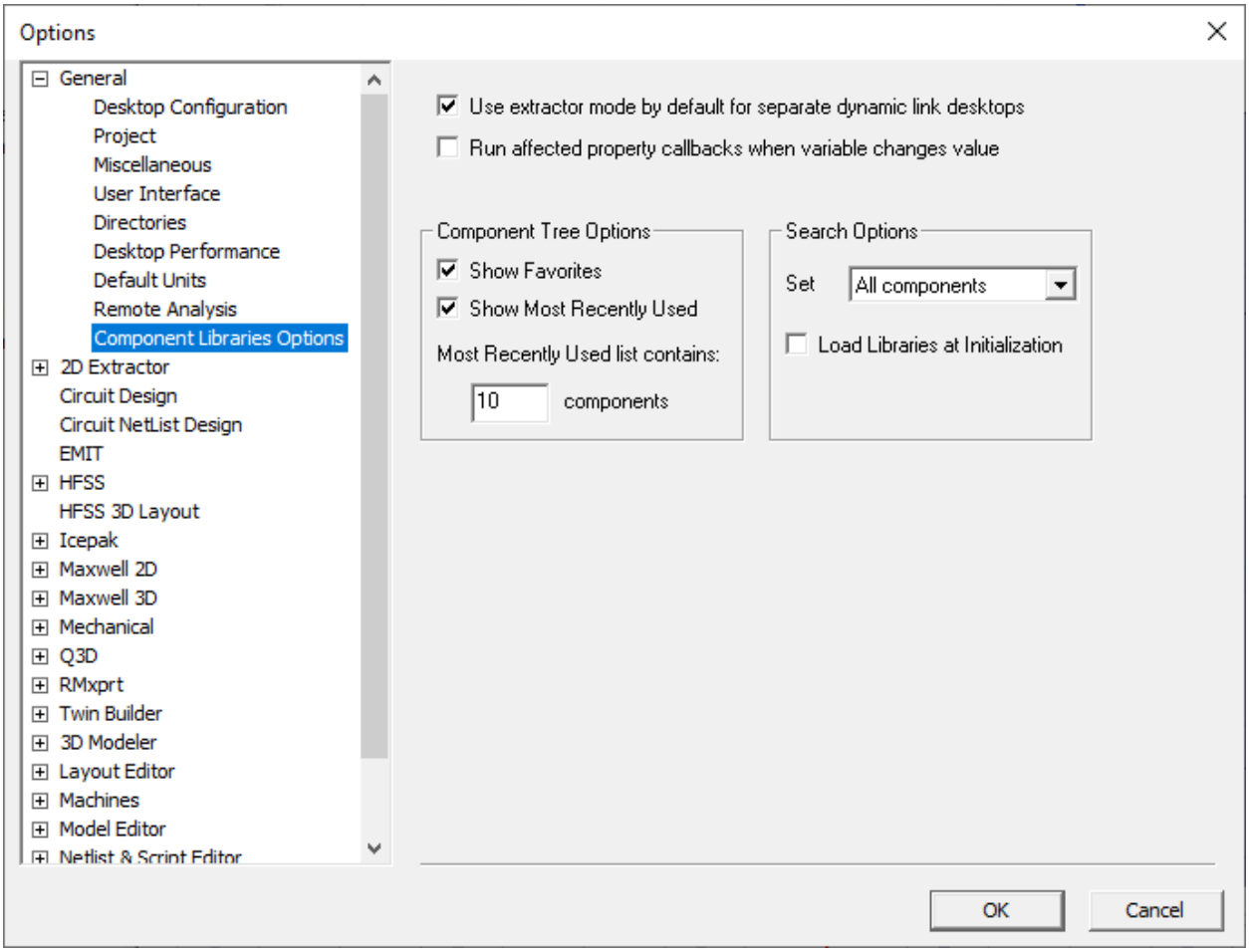
```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

General Options: Component Libraries Options

Under General Options, **Component Libraries** options allow you to change how Electronics Desktop handles components.

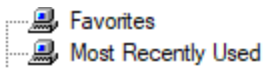
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can elect to **Use extractor mode by default for separate dynamic link desktops**, or to **Run affected property callbacks when variable changes value**.

In the **Component Tree Options** area, options include:

- **Show Favorites** – enables the **Favorites** folder in the **Component Libraries** tree.
- **Show Most Recently Used** – enables the **Most Recently Used** folder in the **Component Libraries** tree. Use the field to specify the number of components shown in the **Most Recently Used** folder. The default is 10.



In the **Search Options** area, options include:

- **Set** – specifies the search set when searching the Component Libraries. Choose to search **All components**, **Current list only**, or **Append to current list**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

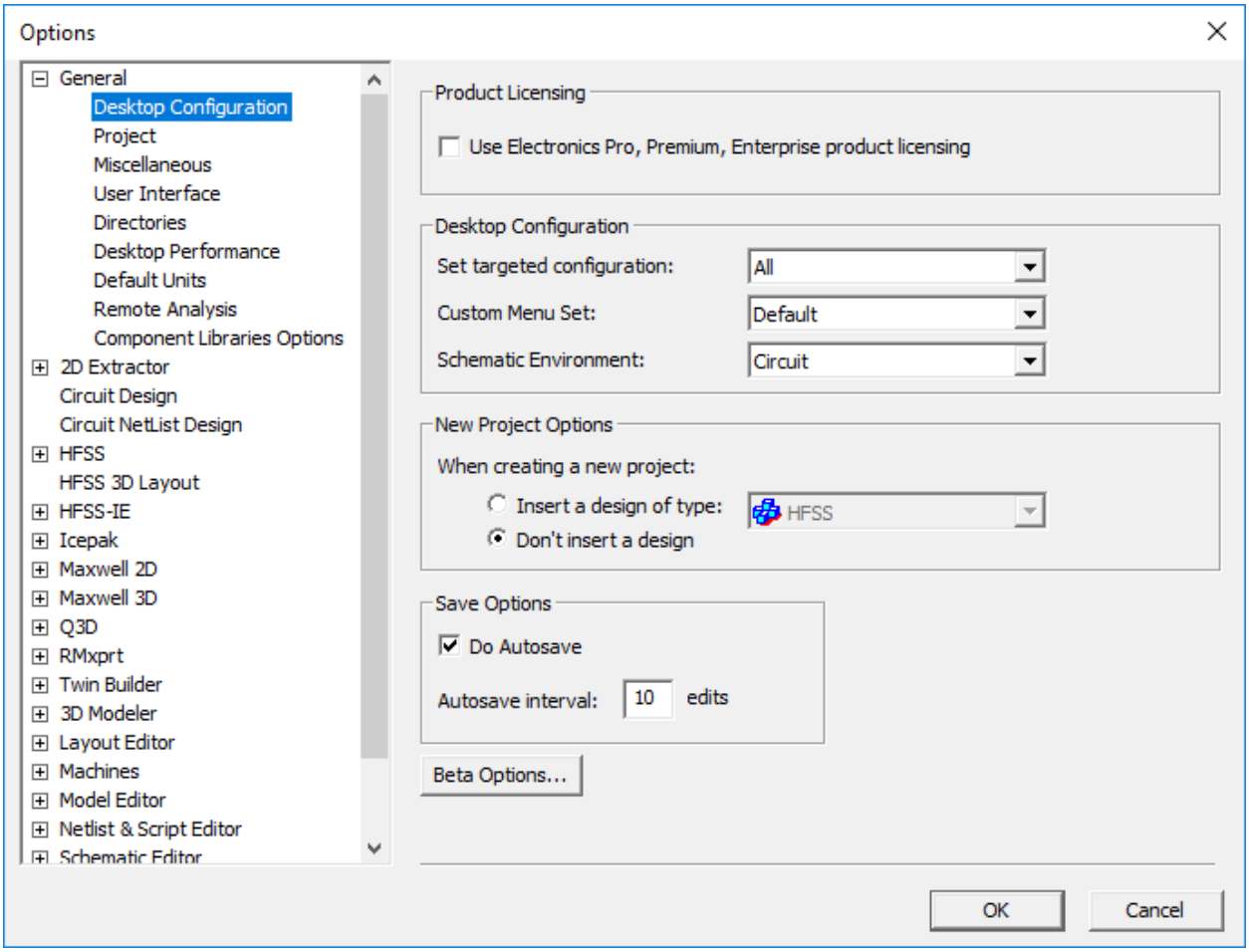
- **Load Libraries at Initialization** – this option can slow initialization.

Setting Mechanical Options

To set Mechanical options in Electronics Desktop:

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

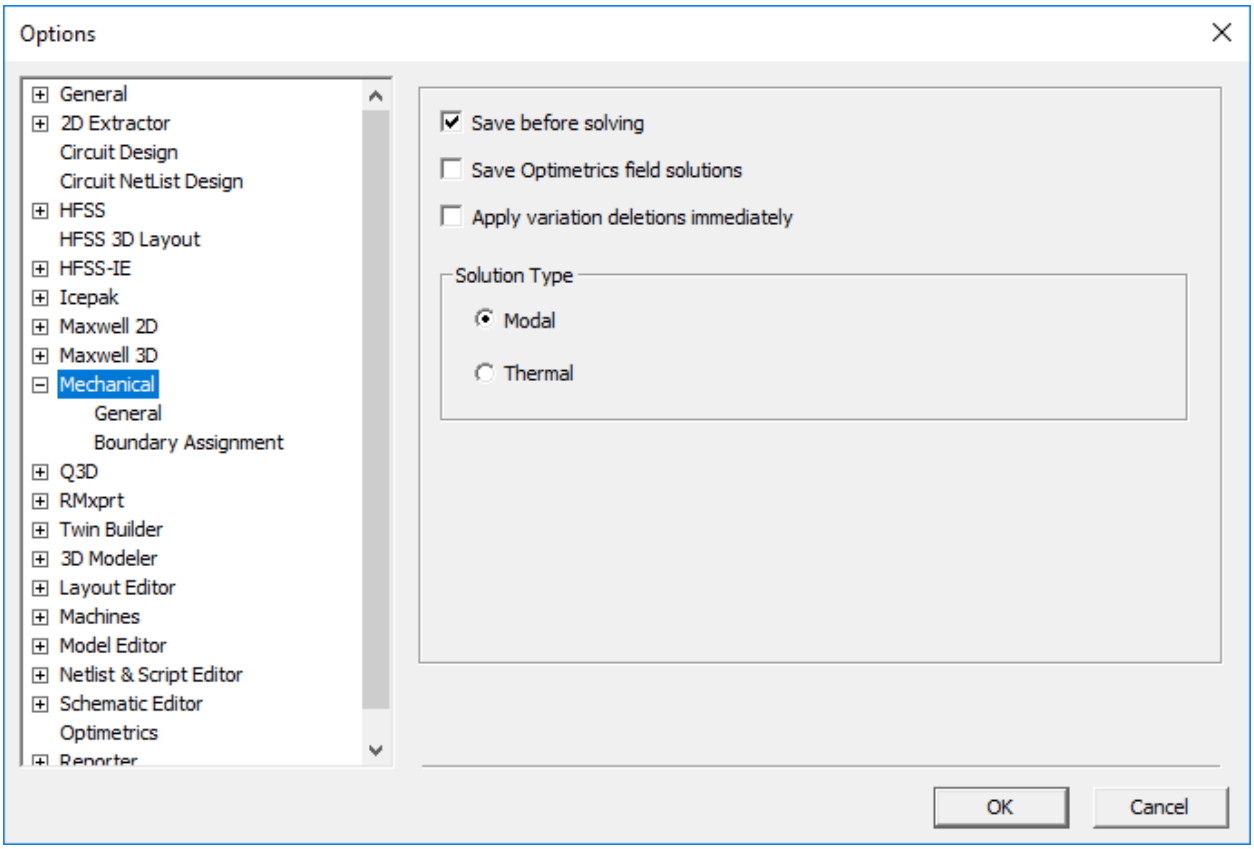
   The **Options** window appears, displaying **General** options.



2. In the left pane, expand and select **Mechanical**.

   The **Mechanical** options branch expands, and the Mechanical **General** options display by default:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the tree in the left pane to select a group of options.

Under Mechanical **General** options, you can set the following:

- **Save before solving** – forces a full save before running the solver. If a simulation is started while another is running, you will be asked if you want to solve without saving first. You can run multiple simulations, and if you have not edited the project in between solutions, crash recovery will work. In any case, you can start a new solution while running another without having to abort the current simulation.
- **Save Optimetrics field solutions** – when enabled, saves the field solution data for every solved design variation.
- **Apply variation deletions immediately** – select to save disk space.
- **Solution Type** – Two solution types are available for Mechanical projects. Choose one of the following two options:
  - **Modal:** Determine natural frequencies and vibration mode shapes.
  - **Thermal:** Perform a steady-state heat transfer analysis to determine temperature and heat flux distributions.

Under Mechanical **Boundary Assignment** options, you can set the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Use wizards for data input when creating new boundaries:** When applicable a step-by-step wizard walks you through the assignment of new boundary conditions. Existing boundaries are edited through the conventional boundary dialog box.
- **Duplicate boundaries/mesh operations with geometry** – when enabled, boundaries and mesh operations assigned to geometry are copied along with the geometry. That is, duplicating the geometry also duplicates the boundary and mesh operation assignments.
- **Visualize boundaries on geometry** – when enabled, boundaries are visualized on the model geometry when selected in the Project Manager.

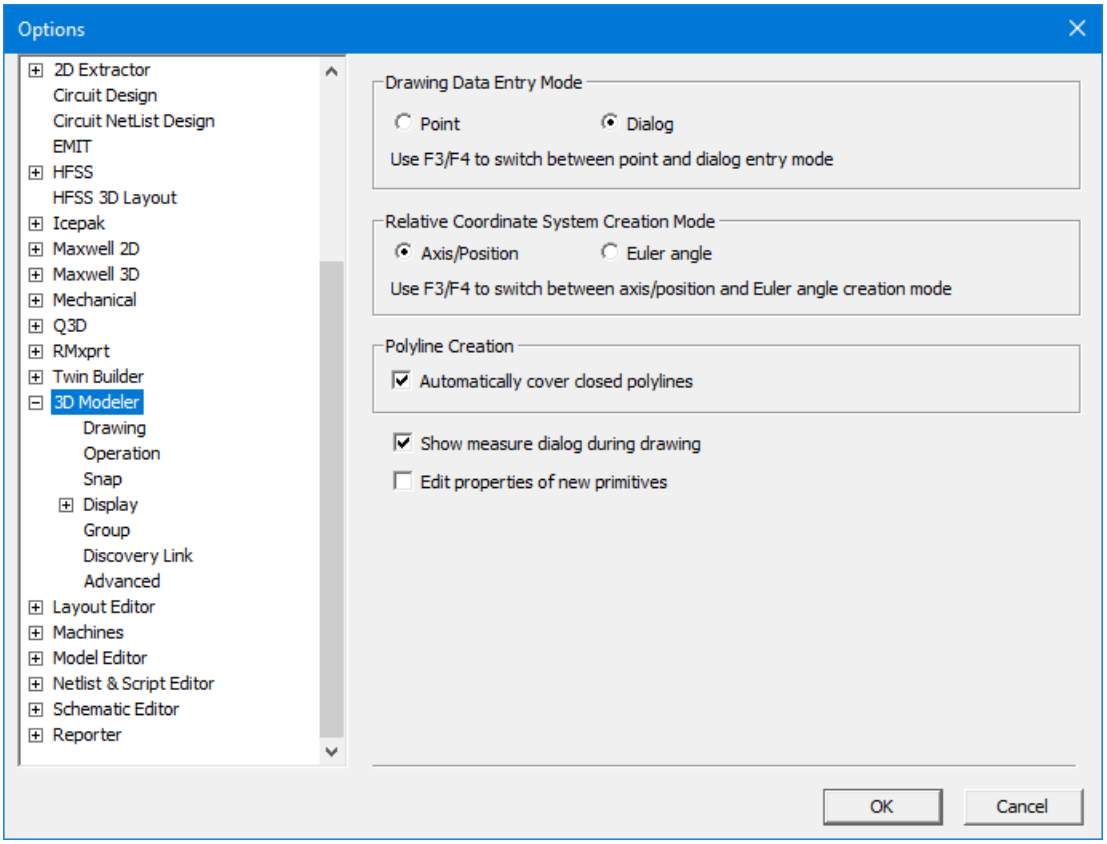
Setting 3D Modeler Options

To set 3D Modeler options in Ansys Electronics Desktop:

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

   The **Options** window opens with the Desktop Configuration options selected by default.

   In the left pane, select 3D Modeler to view modeler options:



   In the left pane, select the entries below **General** to display the associated options:

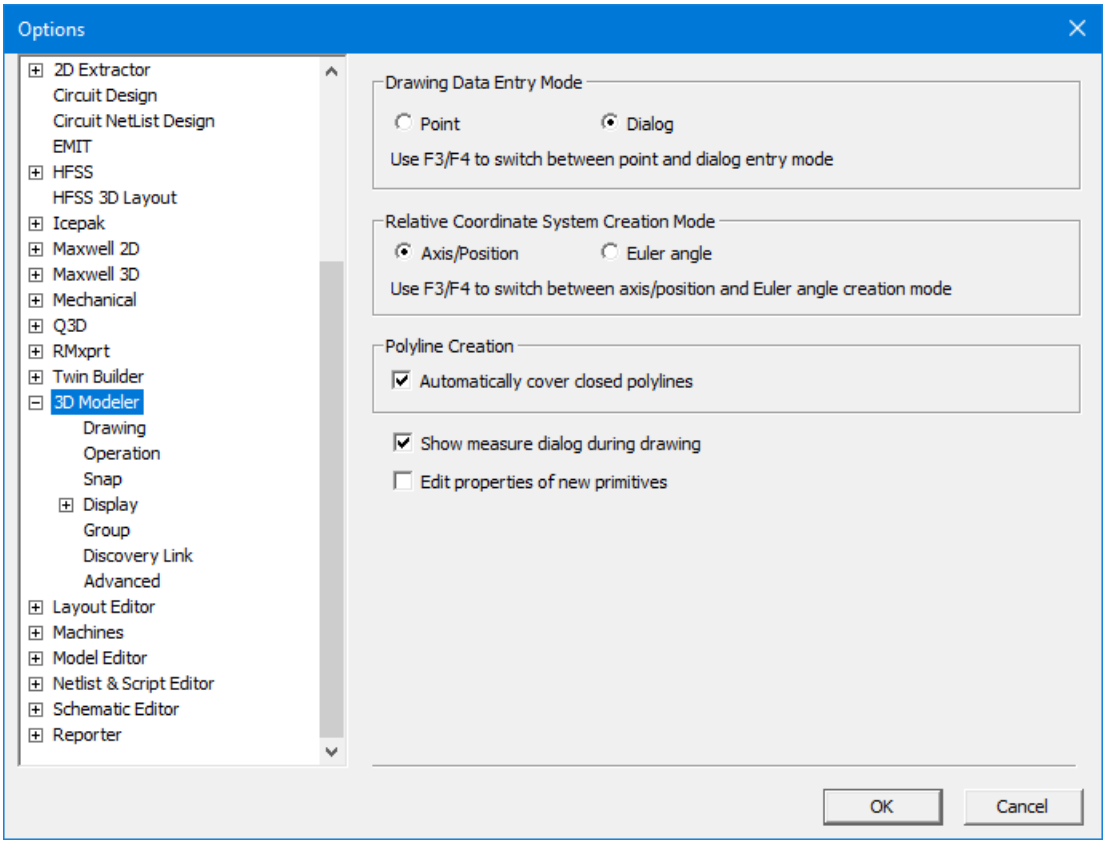
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Drawing
- Operation
- Snap
- Display
- Group
- Discovery Link
- Advanced

2. Click each entry and make the desired selections.
3. Click **OK** to apply your preferences.

3D Modeler Options: Drawing Options

Under 3D Modeler options, **Drawing** options allow you to decide how Electronics Desktop draws objects.



In the **Drawing Data Entry Mode** area, select **Point** to draw new objects directly via the mouse, or select **Dialog** to prompt a **Properties** dialog box that allows you to enter object dimensions. Dialog mode works with the equation-based line and all two- and three-dimensional objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Relative Coordinate System Creation Mode** area, select **Axis/Position** or **Euler angle**. This determines how the relative coordinate system is created from the **Modeler** > **Coordinate System** > **Create** > **Relative CS** menu. When **Axis/Position** is selected, relative coordinate systems are created in Draw mode. When **Euler angle** is selected, they are created from values entered in a dialog box.

In the **Polyline Creation** area, enable or disable **Automatically cover closed polylines**. When enabled, polylines become sheet objects and are listed as such in the History tree. When deselected, closed polylines are left as uncovered objects and listed as lines in the History tree.

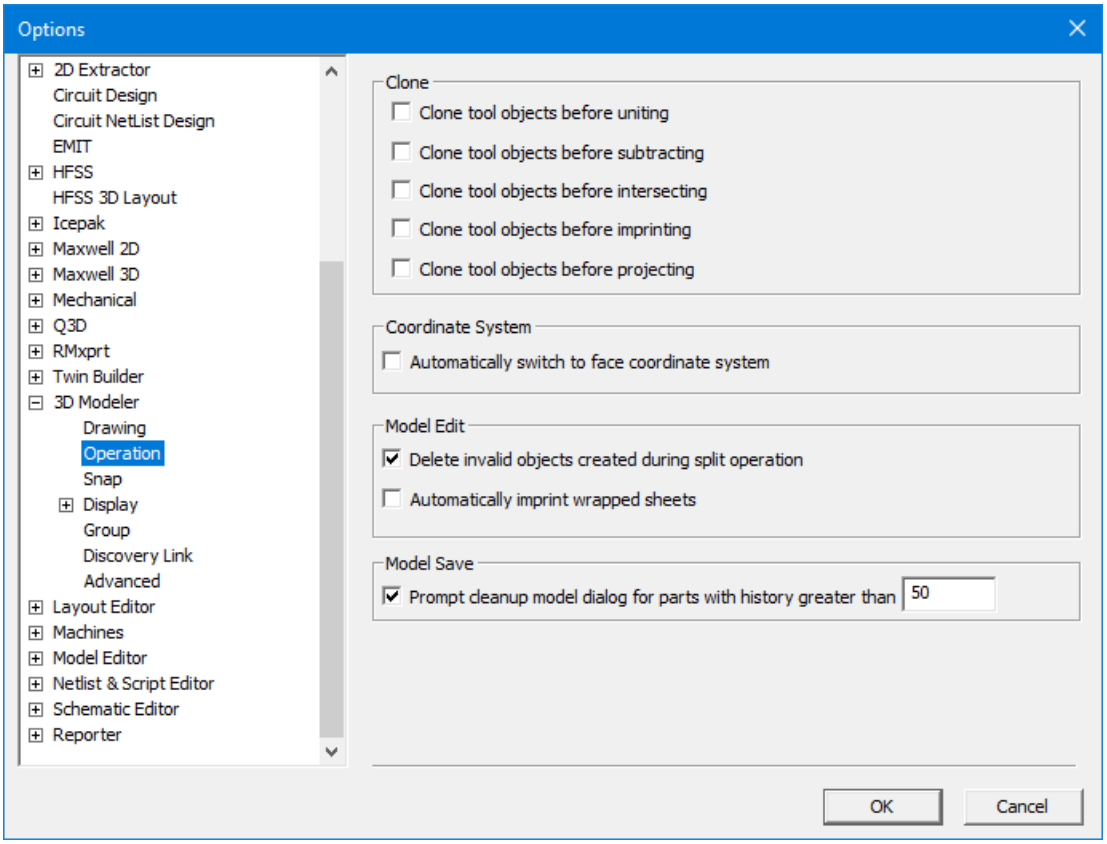
Other options are:

- **Show measure dialog during drawing** – specifies whether a **Measure** dialog box appears upon creation of a new primitive. The dialog box shows the coordinates of the current cursor position.
- **Edit properties of new primitives** – specifies whether a **Properties** dialog box display when you create a new object in the modeling workspace.

3D Modeler Options: Operation Options

Under 3D Modeler options, **Operation** options allow you to select options for clone, coordinate system, model editing, and model save operations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Clone** area, enable or disable options. By default, the modeler deletes tool objects when performing operations on objects. You may specify that the modeler make a clone (copy) of the tool object before these operations:

- **Clone tool objects before uniting**
- **Clone tool objects before subtracting**
- **Clone tool objects before intersecting**
- **Clone tool objects before imprinting**
- **Clone tool objects before projecting**

In the **Coordinate System** area, decide whether or not to **Automatically switch to face coordinate system**.

By default, the modeler operates within the user-selected coordinate system. If this option is enabled, when you select a face and create a new object, the modeler first creates a face coordinate system consistent with the selected face and the new object is created within the face coordinate system. When this option is disabled, you must manually create a Face Coordinate System before creating an object related to it.

In the **Model Edit** area, enable or disable the following options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

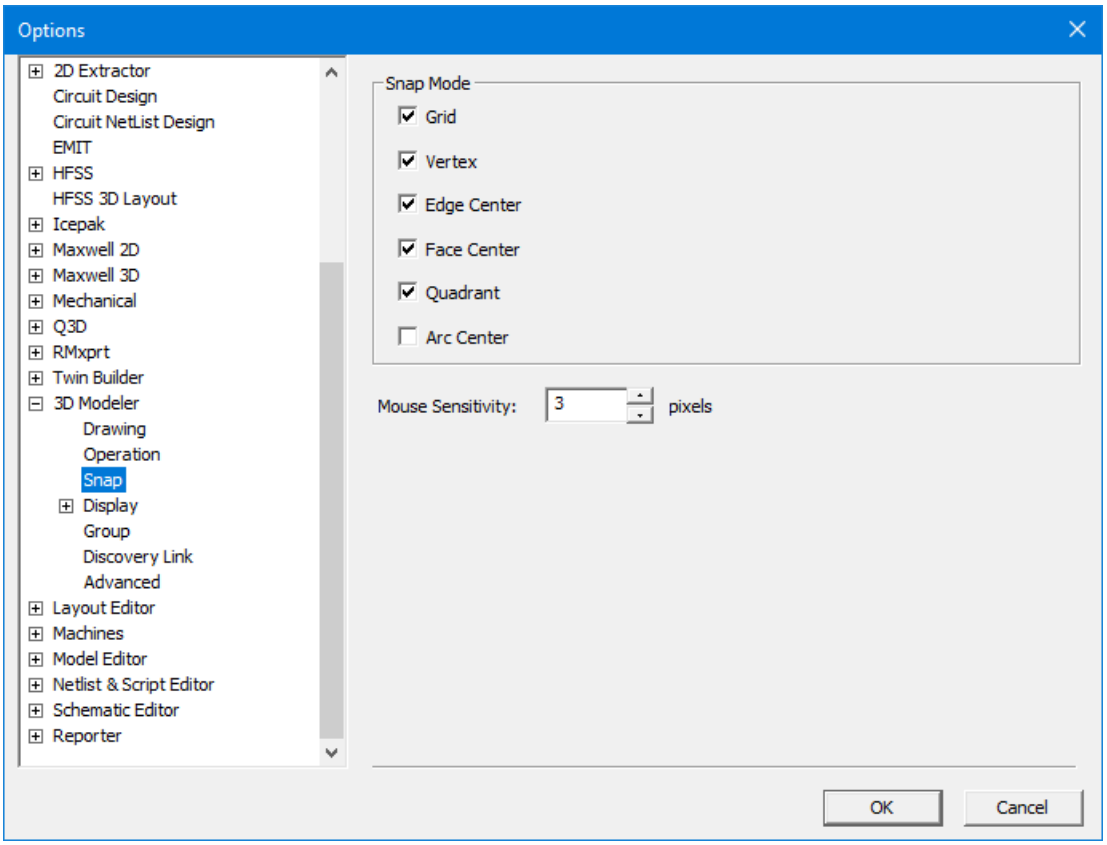
- **Delete invalid objects created during split operation** – When selected, the modeler deletes invalid objects created during split operation. When deselected, invalid objects can be freely created.
- **Automatically imprint wrapped sheets** – Imprinting is a boolean operation, whereas wrapping is not.

  See: Imprinting an Object and Wrap Sheet Command.

In the **Model Save** area, you can enable/disable being prompted for model history cleanup. You can also specify the number of actions on a part that will cause a prompt. The default value is 50.

3D Modeler Options: Snap Options

Under 3D Modeler options, **Snap** options allow you to decide how Electronics Desktop snaps objects. By default, most objects are set to snap (adhere) to the nearest point on the grid. The coordinates of this point are used, rather than the exact location of the mouse.



In the **Snap Mode** area, select the points you would like the cursor to snap to:

- Grid
- Vertex

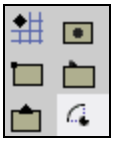
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edge Center
- Face Center
- Quadrant
- Arc Center

Use the **Mouse Sensitivity** option to select how many pixels you must move the mouse before a different snap point is selected. Values between 1 and 20 are accepted, with 1 being the most sensitive and 20 being the least sensitive. The default is 3.

You can also access snap settings:

- **From the Modeling Workspace** – While in draw mode, right-click and select **Snap Mode**.
- **From the Toolbar** – Select **Modeler** > **Snap Mode**.
- **From the Draw Ribbon** – Select and deselect the snap icons.



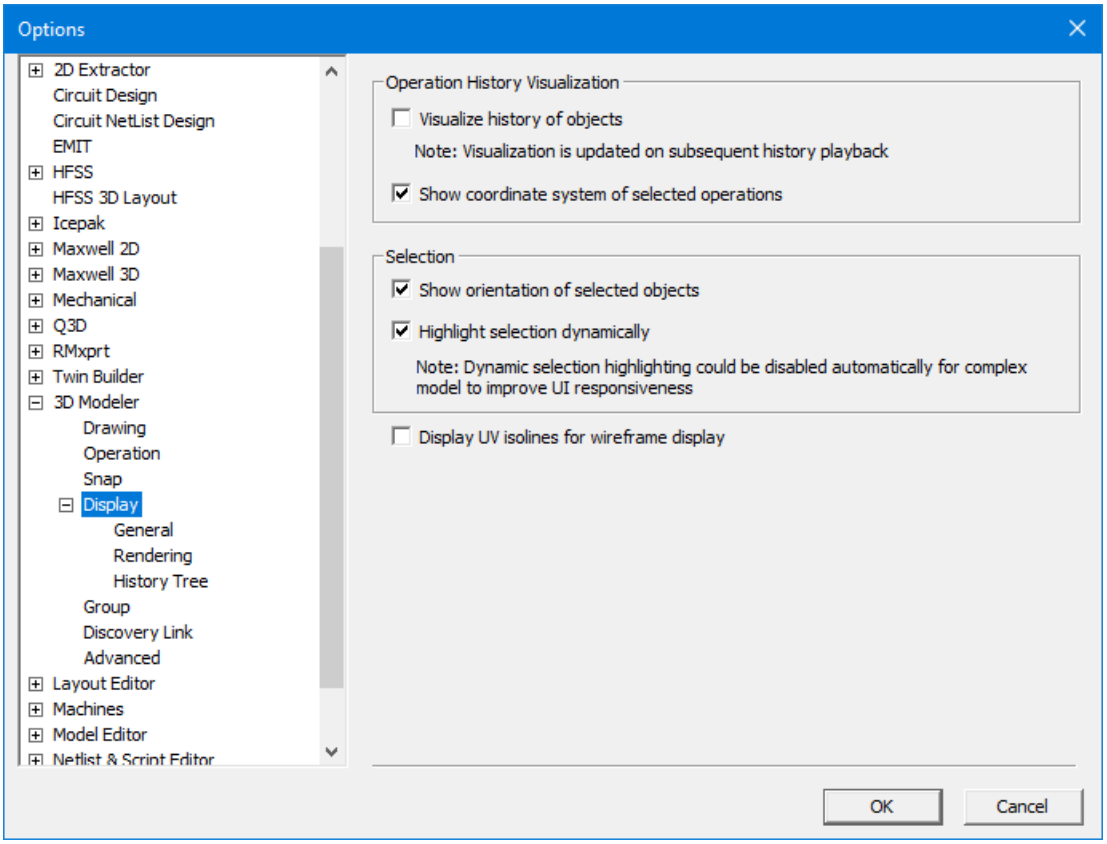
Hover over each icon for a tooltip indicating its equivalent menu option.

> **Important:**
>
> In general, you should select at least one of the snap options . If none is selected, the software is in "free mode" and selects whatever point you click, regardless of its coordinates. This can cause problems creating closed objects because the point you select may appear to be the vertex point of an open object, but may not be the exact coordinates.

3D Modeler Options: Display Options

Under 3D Modeler options, **Display** options allow you to change how Electronics Desktop displays your project tree, history, and objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



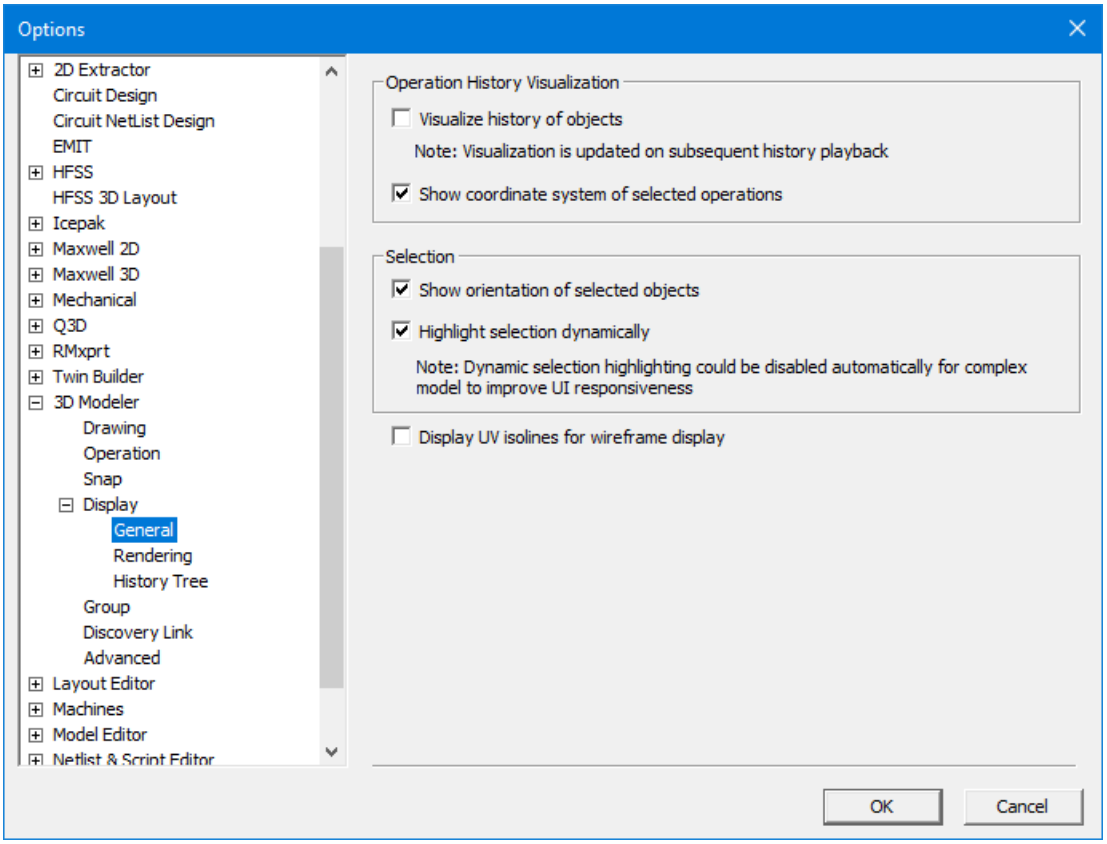
In the left pane, select the entries below **Display** to display the associated options:

- General
- Rendering
- History Tree

Display Options: General Options

Under Display Options, **General** options allow you to change how objects are selected and how their history is visualized.

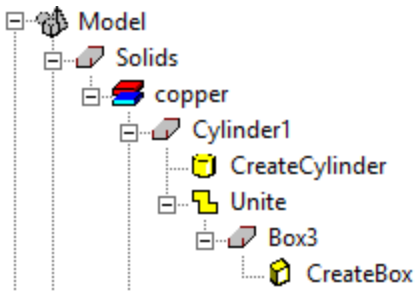
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Operation History Visualization** area, set the following options:

- **Visualize history of objects** – When selecting an object in the history tree, this option lets you view an outline of each part that comprises the object. This can help you visualize an object that has been merged with another object. Clearing this selection removes visualization of objects that are part of the model history. For large models, this is faster and uses less memory.

  Visualized history:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You must restart Electronics Desktop for this option to take effect.

- **Show coordinate system of selected operations** – Select or clear.

In the **Selection** area, set the following options:

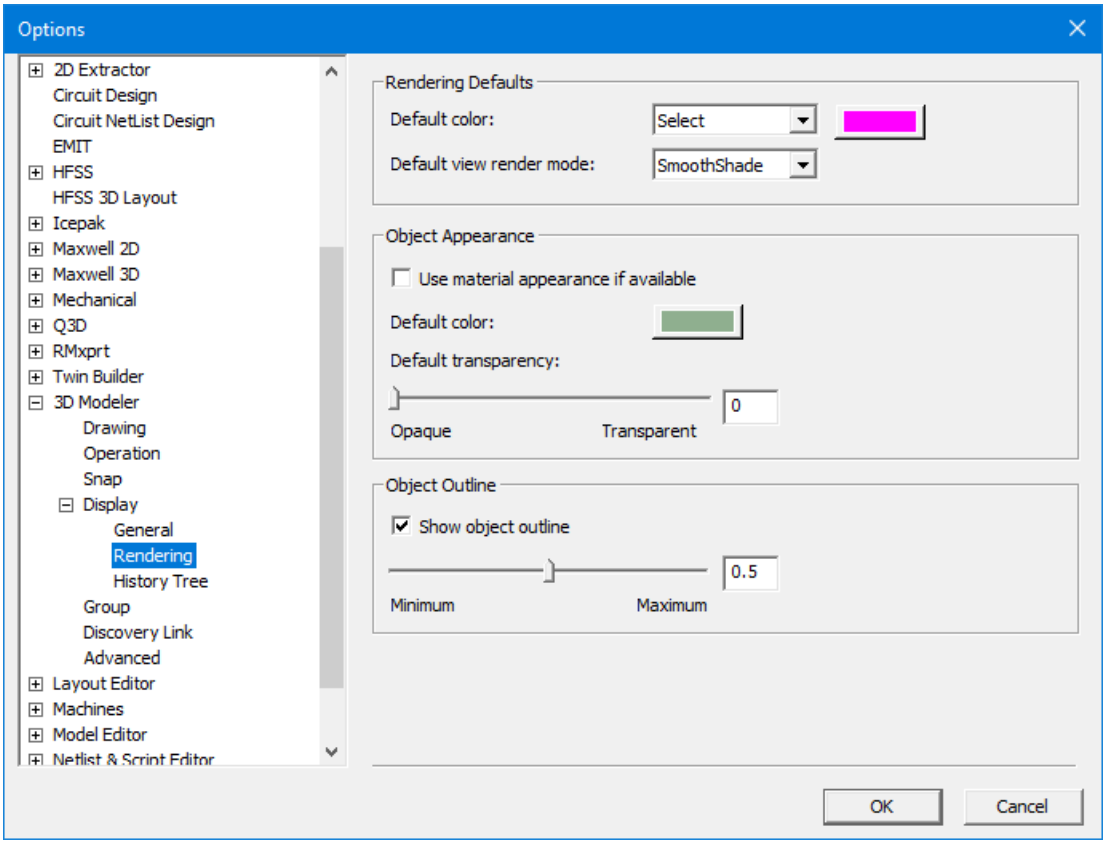
- **Show orientation of selected objects** – Select or clear.
- **Highlight selection dynamically** – This option causes objects or faces (depending on the selection mode) to be highlighted when you pass the mouse pointer over them. You may want to turn this off for complicated models as it can reduce responsiveness. If Electronics Desktop identifies the model to be large enough, dynamic highlighting is automatically ignored. In this case you will see a message about this.

You can also **Display UV isolines for wireframe display**. For models with curved faces, you may prefer to clear this option to simplify the wire-frame display and speed up rendering.

Display Options: Rendering Options

Under Display Options, **Rendering** options allow you to change rendering and object appearance defaults.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Rendering Defaults** area, set the following options:

- **Default color** – Use the drop-down menu to choose the **Select**, **Highlight**, or **Rubberband** action. Click the color button to select a color for each action.
- **Default view render mode** – Select either **WireFrame** or **SmoothShade**. When dealing with complicated geometries, choose **WireFrame** rendering for speed.

In the **Object Appearance** area, set the following options:

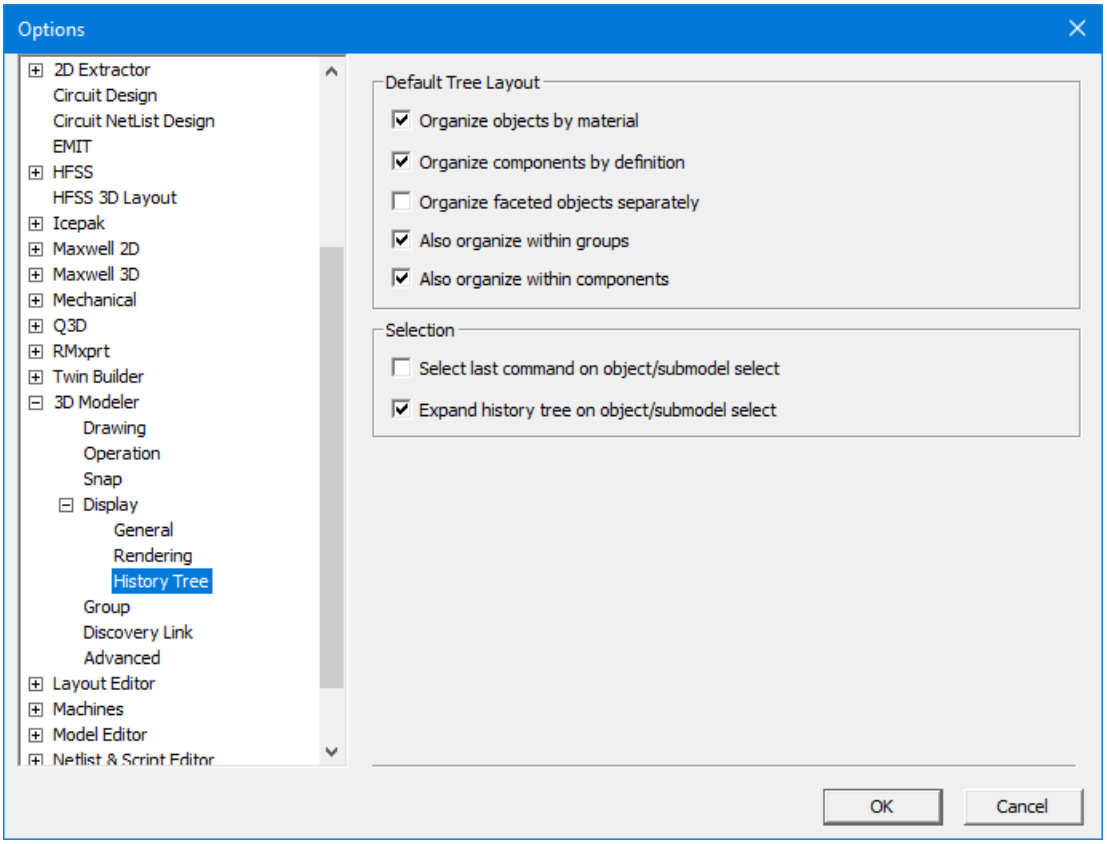
- **Use material appearance if available** – When this option is selected, default color and transparency used for newly created objects come from material settings. Some materials may not have default appearance specifications; in those cases, default color and transparency specified under **Rendering Defaults** are used.
- **Default color** – Click to select the default color.
- **Default transparency** – Use the slider to select the default transparency.

In the **Object Outline** area, select or clear **Show object outline**. If you elect to show the object outline, use the slider to determine the outline thickness. Value must be between 0 and 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Options: History Tree Options

Under Display Options, **History Tree** options allow you to change the history tree display and selection response.



In the **History Tree Layout** area, change the organization of the history tree as desired:

- **Organize objects by material**
- **Organize components by definition**
- **Organize faceted objects separately**
- **Also organize within groups**
- **Also organize within components**

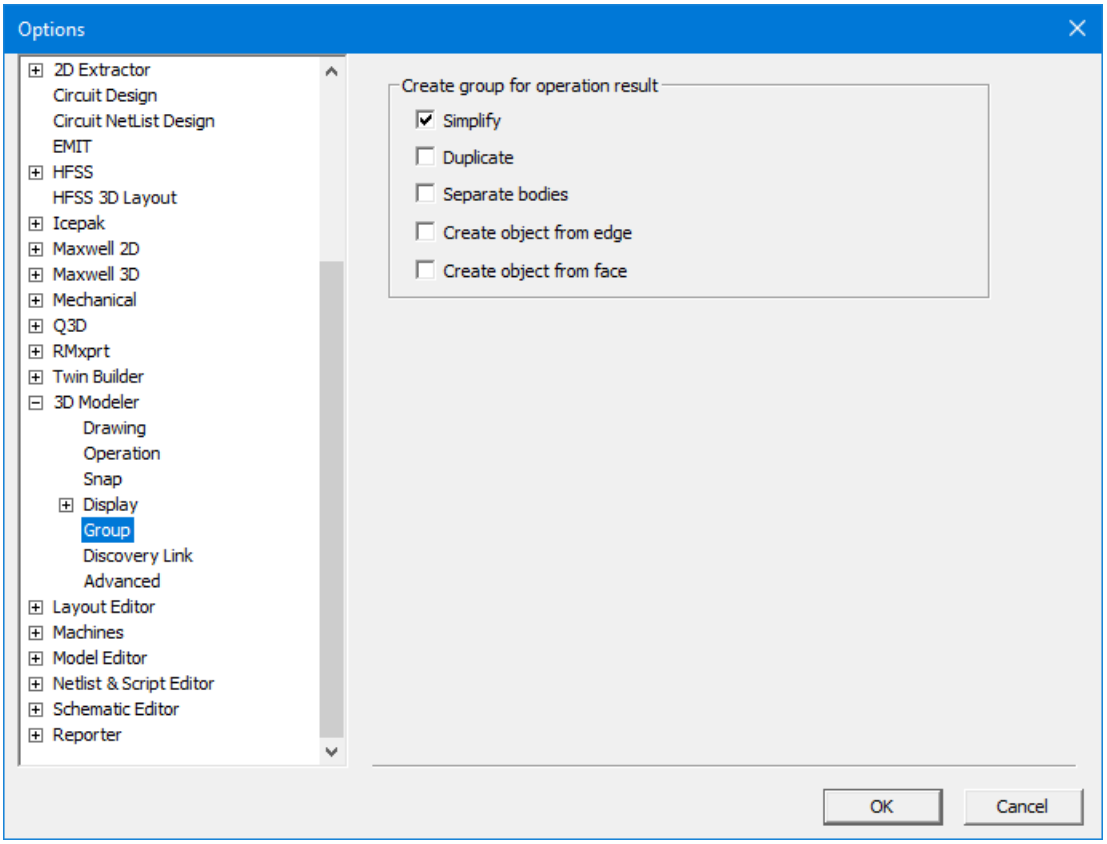
In the **Selection** area, set the following options:

- **Select last command on object/submodel select** – When selected, the history tree is expanded after operations on object properties, even if the tree is collapsed for the item. The command is selected.
- **Expand history tree on object/submodel select** – When selected, selecting an object automatically opens the history tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Options: Group Options

Under 3D Modeler options, **Group** options determine how groups are created based on operations.



In the **Create group for operation result** area, select or clear the following actions:

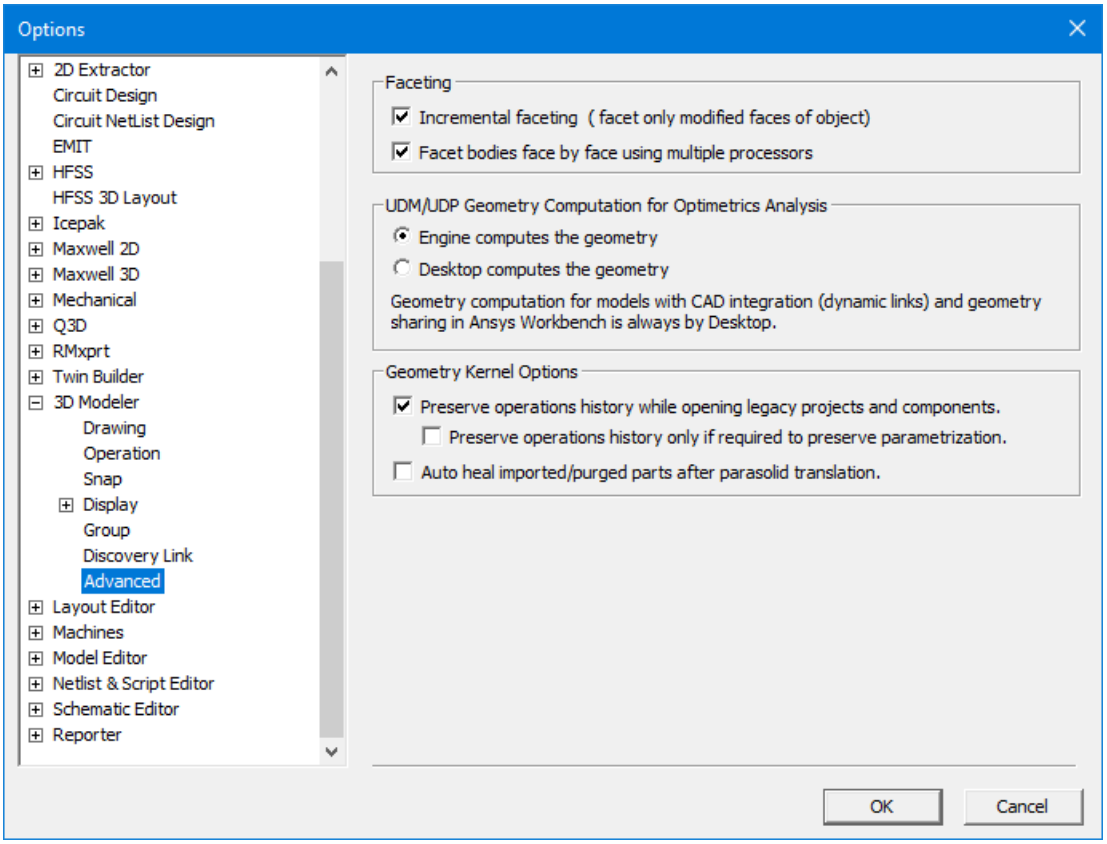
- **Simplify**
- **Duplicate**
- **Separate bodies**
- **Create object from edge**
- **Create object from face**

When any selected action is performed, a group will be created in the History Tree.

3D Modeler Options: Advanced Options

Under 3D Modeler options, **Advanced** options help configure faceting and UDM/UDP.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Faceting** area, select **Incremental faceting** or **Facet bodies face by face using multiple processors**. Face by face will be slower.

In the **UDM/UDP Geometry Computation for Optimetrics Analysis** area, select whether **Engine** or **Desktop** computes the geometry.

In the **Geometry Kernel Options**, enable **Preserve operations history while opening legacy projects** to preserve operations history while opening legacy projects that used the ACIS kernel. When opening legacy projects, the conversion is a one-time event. A renamed copy of the original file is preserved. The default is to preserve operations history. If the option is disabled, the operations history is purged. The option to purge history is available because:

- Some operations cannot be replayed in Parasolid.
- Purging provides backup to translate geometry.
- Parameterization is lost.
- Expect a better conversion success rate with purged models.

Setting Reporter Options

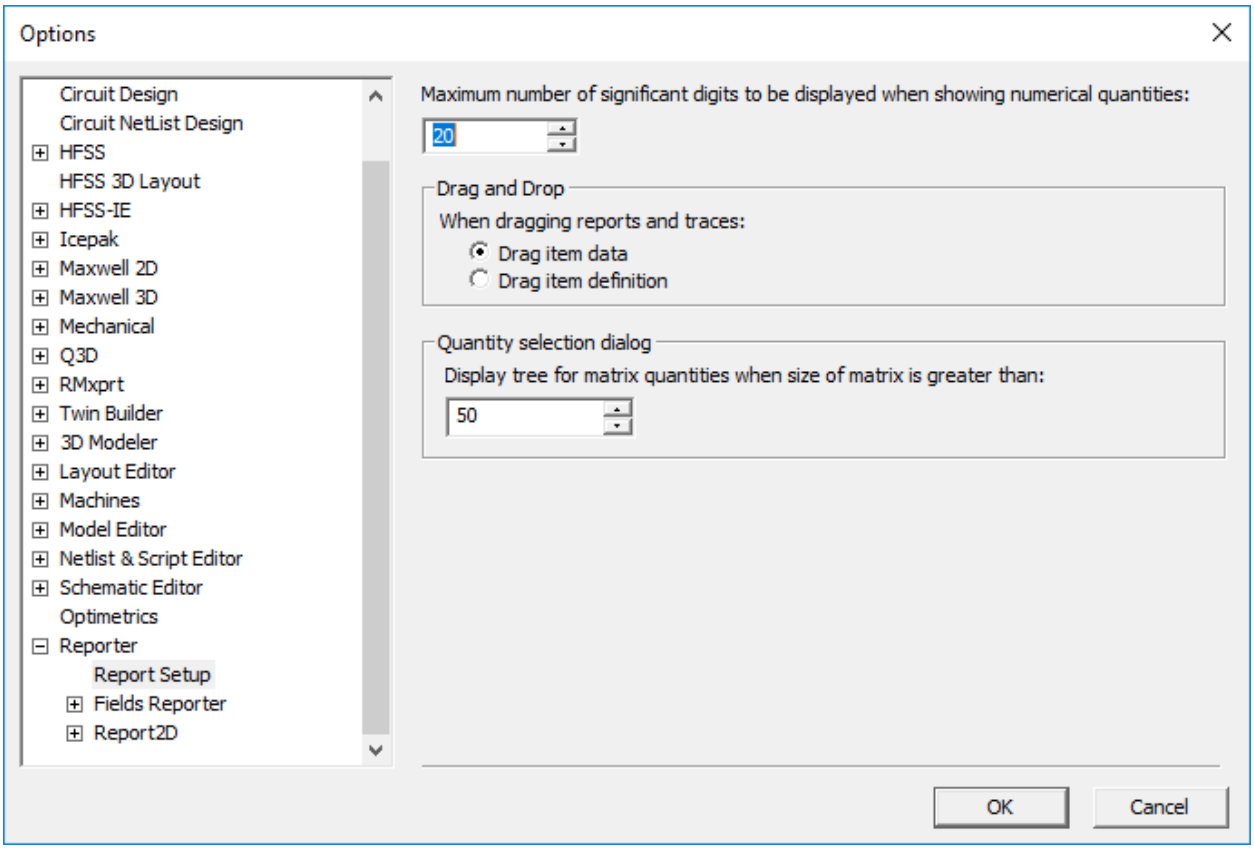
To set Reporter options in Ansys Electronics Desktop:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Options** > **General Options**.
2. In the left pane, select **Reporter**.
3. Select the entries below **Reporter** to display the associated options:
   - Report Setup
   - Fields Reporter
   - Report2D
4. Click each entry and make the desired selections.
5. Click **OK** to apply your preferences.

Reporter Options: Report Setup Options

Under Reporter Options, **Report Setup** options allow you to customize significant digits, drag-and-drop, and quantity selection options for reports.



Set the **Maximum number of significant digits to be displayed when showing numerical quantities**. The value must be between 0 and 20.

In the **Drag and Drop** area, select how Electronics Desktop behaves when you drag reports and traces. You can either **Drag item data** or **Drag item definition**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Quantity Selection Dialog** area, select a **Display tree for matrix quantities when size of matrix is greater than** value. When the number of matrix elements is larger than the selected number, the Quantities field uses a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. This is useful when dealing with large matrices.

Reporter Options: Fields Reporter Options

Under Reporter Options, **Fields Reporter** options fall into three categories:

- Animation Options
- Mesh Plot Options
- Streamline Plot Options

In the left pane, select a category to view those options.

From the **Animation** options, you can set the following:

- **Group Field Overlays by Type** – If selected, field overlays are grouped in the Project Tree. If deselected, they are not.
- **Default Phase Animation Settings** – Select the default start and stop angles as well as the number of steps for Scalar and Vector plots.

From the **Mesh Plot** options, you can set the **Clipping of volume mesh plot**. When dragging a clip plane, the plot can update automatically. Select **Never** to disable this feature, **Always** to enable it at all times, or set a maximum number of mesh elements for automatic update. This option allows you to update plots for smaller meshes but avoid automatically updating larger plots that may consume too much memory.

From the **Streamline Plot** options, set **Streamline drawing stopping criteria** and **Streamline marker spacing**.

Reporter Options: Report2D Options

Under Reporter Options, **Report2D** options fall into several categories:

- Curve
- Axis
- Grid
- Header
- Note
- Legend
- Marker
- Marker Table
- X/Y Markers

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Stacked
- Digital
- General
- Table

In the left pane, select a category to view those options.

Report2D Options: Curve

Curve settings change how curves are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Curve**:

- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Width** – set the line width by editing the real value in the text field.
- **Arrows** – select the check box to use arrows on curve ends.
- **Show Symbol** – select the check box to have symbols mark the locations of data points on the curve.
- **Sym Freq** – set the symbol frequency by editing the integer value in the text field.
- **Sym Style** – use the drop-down menu to select a symbol from: Box, Circle, Vertical Ellipse, Horizontal Ellipse, Vertical Up Triangle, Vertical Down Triangle, Horizontal Left Triangle, and Horizontal Right Triangle.
- **Fill Sym** – select the check box to set the symbol display as solid. Otherwise it will display as hollow.
- **Sym Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Axis

Axis settings change how axes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Axis**:

- **Axis Name** – this describes the axis to which a row's settings apply. You cannot change this field.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Edit Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Min Gutter %** – sets the amount of empty space (gutter) around the axis.
- **Font Description** – describes the settings applied from the **Font** window. You cannot change this field. To change the font, use **Edit Font**.

Report2D Options: Grid

Grid settings change how grids are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Grid**:

- **Grid Name** – this describes the grid to which a row's settings apply. You cannot change this field.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Header

Header settings change how headers are displayed in reports. You can separately style the Title and Subtitle.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Header**:

- **Color** – click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font** – click **Edit Title Font** or **Edit Subtitle Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Company Name** – Enter a company name to appear on all reports.
- **Show Ansys Logo** – Remove the check from the box to hide the Ansys logo that appears on all reports (checked by default).

Report2D Options: Note

Note settings change how notes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Note**:

- **Note Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Note Font** – click **Edit Note Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Visibility** – select the check box to make the background color visible. Deselect to make it transparent.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Visibility** – select the check box make the note border visible. Deselect to make it transparent.
- **Border Line Width** – set the line width by editing the real value in the text field.

Report2D Options: Legend

Legend settings change how legends are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Legend**:

- **Legend Name** – Default is no name. When non-empty, a header row for the Legend in plot shows up with that string.
- **Show Trace Name** – select the check box to show the trace name; deselect it to hide the trace name.
- **Show Solution Name** – select the check box to show the solution name; deselect it to hide the solution name.
- **Show Variation Key** – select the check box to show the variation key; deselect it to hide the variation key.
- **Highlight Curve on Hover** – select the check box to highlight a curve when you hover the cursor over it; deselect it to leave the curve as-is when you hover the cursor.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

You can set different **Text Color** and **Text Font** settings for the Header Row.

Report2D Options: Marker

Marker settings change how markers are displayed in reports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker**:

- **Marker Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Arrow Direction** – use the drop-down menu to set the arrow direction to Up, Down, Left, or Right.

Report2D Options: Marker Table

Marker Table settings change how marker tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker Table**:

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Line Width** – set the line width by editing the real value in the text field.

You can set a different **Text Color** and **Text Font** for the header row.

Report2D Options: X-Y Markers

X-Y Marker settings change how X-Y markers are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **X-Y Markers**:

Background Colors

- **Marker [#] Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Properties

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **On-screen intersection** – select the check box to enable on-screen intersection; deselect to disable.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.
- **Show Name** – select the check box to show X-Y marker names; deselect to hide.
- **Snap to Vertex** – select the check box to snap markers to the vertex; deselect to disable snapping.

Inter Marker Deltas

- **Show Delta** – select the check box to show inter-marker deltas; deselect to hide.
- **Delta Font** – click **Edit Delta Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Delta Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.

Report2D Options: Stacked

Stacked settings change how stacked plots are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Stacked**:

- **Auto Fit Mode** – select the check box to enable auto fit; deselect to turn off autofit.
- **Stack Height in Pixels** – set the default stack height.
- **Curve Grouping Strategy** – can be **Single**, **By Trace**, or **By Units**. **Single** means that a new stacked plot shows a single curve per stack. **By Trace** means that all curves are grouped by their trace. **By Units** means that all curves are grouped by their unit type.

If you change the **Curve Grouping Strategy** set by default, existing stacked plots remain unaffected. The new default will apply only to new stacked plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report2D Options: Digital

Digital settings change how digital stack heights are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Digital**:

- **Digital Literal Foreground** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Expand Arrays/Records** – select the check box to automatically expand arrays and records; deselect it to disable.
- **Digital Stack Height in Pixels** – enter the stack height for Analog, Digital, Enum, Event, and Literal stacks.

Report2D Options: General

These settings contain general report options.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **General**:

- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Plot Area Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Highlight Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Accumulate Depth** – enter a value; the default is 4.
- **Enable Y Axis Stripes** – select the check box to add stripes to the Y axis; deselect to remove stripes.
- **Auto Scale Fonts** -on by default and when enabled scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.

Curve Tooltip

- **Show Trace Name** – select the check box to display trace names in the tooltip when hovering the cursor over a curve.
- **Show Variation Key** – select the check box to display the variation key in the tooltip when hovering the cursor over a curve.
- **Show Solution Name** – select the check box to display the solution name in the tooltip when hovering the cursor over a curve.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clipboard Option

- **Capture Aspect Size Ratio** – select either **As Shown** or **Full Screen**.
- **Capture Background Color** – select either **As Shown** or **White**.

Format

- **Field Width** – enter a field width value.
- **Precision** – enter a precision value.
- **Use Scientific Notation** – select the check box to use scientific notation; deselect to disable scientific notation.

Report2D Options: Table

Table settings change how tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Table**:

- **Rows Per Page** – enter the number of rows you would like to display per page; the default is 2500.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Border Width** – enter a value for the border width.
- **Border Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Width** – enter a value for the grid width.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Page Link Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Arrow Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Header Row

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Format

- **Field Width** – enter a value for the field width.
- **Precision** – enter a value for the precision.
- **Use Scientific Notation** – select the check box to enable scientific notation; deselect it to disable scientific notation.

Copy to Clipboard

- **With Header** – select to include header when copying the table to a clipboard; deselect to remove it.
- **With Tab Separator** – select to include tab separator when copying the table to a clipboard; deselect to remove it.

Setting HPC and Analysis Options

All analysis parameters are accessed via a single window. The machine list and options settings have been integrated into analysis configurations. The default configuration is for solving on a single, local machine. You can create many analysis configurations for remote and distributed solutions, and switch between them depending on the job being solved. Multiprocessing has been integrated into the machine lists.

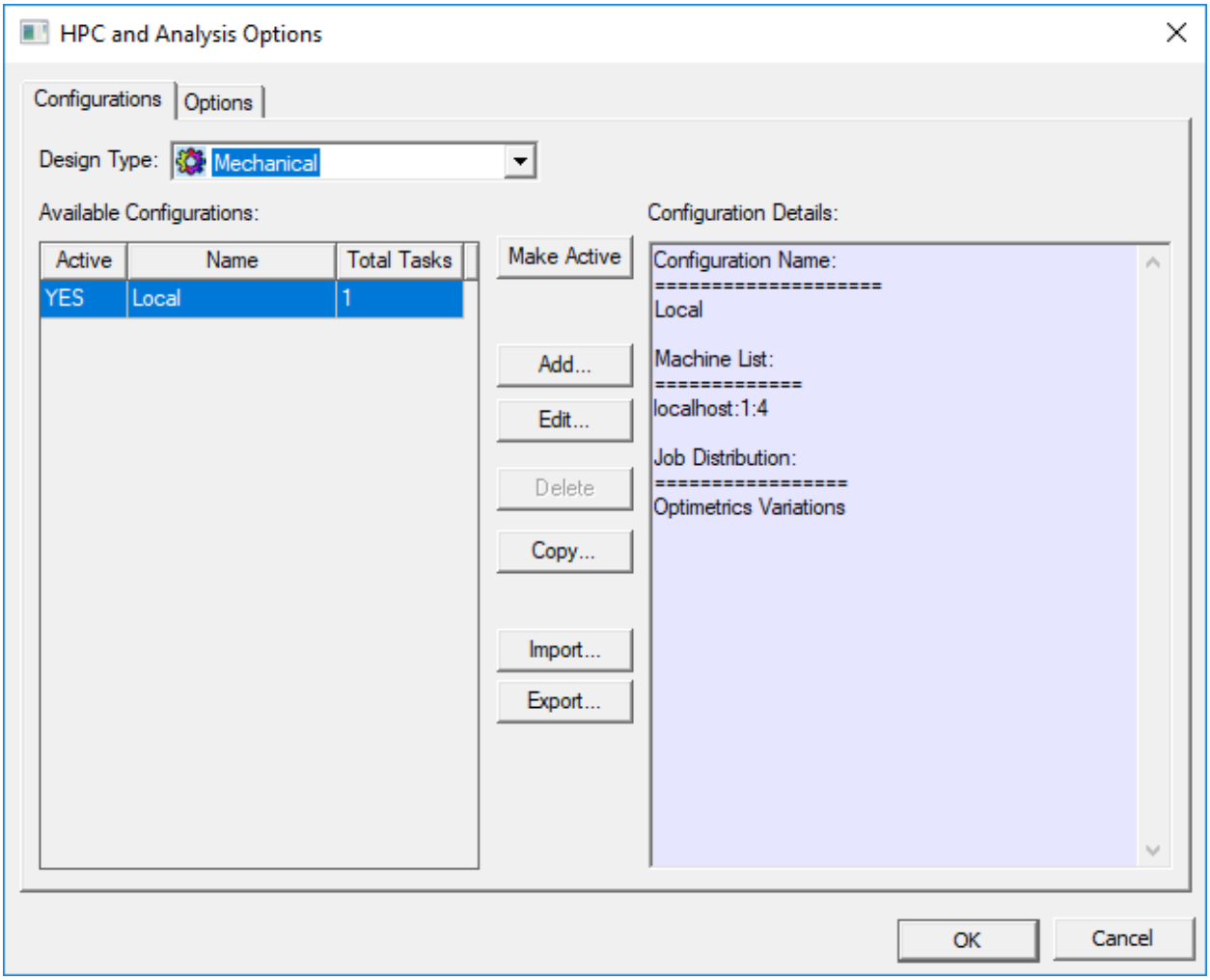
To set HPC and Analysis Options:

1. Click **Tools** > **Options** > **HPC and Analysis Options**.

   You can also access HPC and Analysis Options using the **HPC Options** icon on the **Simulation** ribbon.

   The **HPC and Analysis Options** window appears, displaying the **Configurations** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



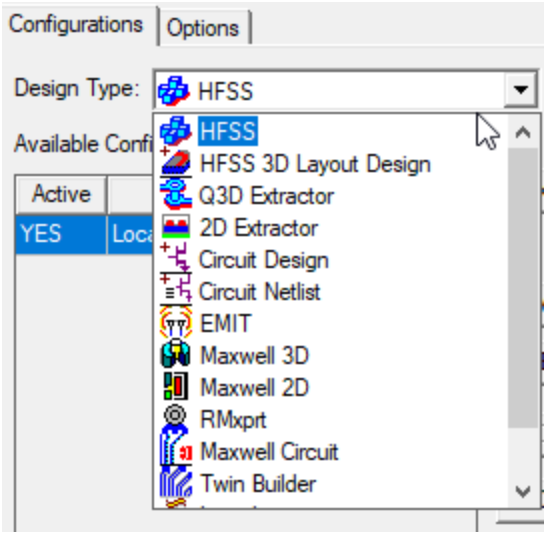
Configurations Tab

Available Configurations are described by **Name**, whether or not they are **Active**, and the **Total Tasks** the configuration can execute.Selecting a configuration from the list displays the details of that configuration in the **Configuration Details** panel.

From the **Configurations** tab, select the **Design Type** to display a list of available configurations for that type.

Configurations must be defined for all design types separately. To use similar analysis parameters for different design types, create separate analysis configurations for each design type. The active configuration is used when solving an analysis for that design type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting an Available Configuration

To activate a configuration, select it from the **Available Configurations** list and click **Make Active**. The active configuration will be indicated by a **YES** in the **Active** column.

Additional options include:

- **Add** – launches a dialog box to create a new analysis configuration.
- **Edit** – launches a dialog box to edit the currently selected analysis configuration.
- **Delete** – deletes the currently selected analysis configuration(s).

  > **Note:** You cannot delete the Local configuration.

- **Copy** – creates a new analysis configuration, and launches a dialog box to edit it. If the dialog box is canceled, the new analysis configuration is not created.
- **Import** – allows you to import an *.acf file to create an analysis configuration.

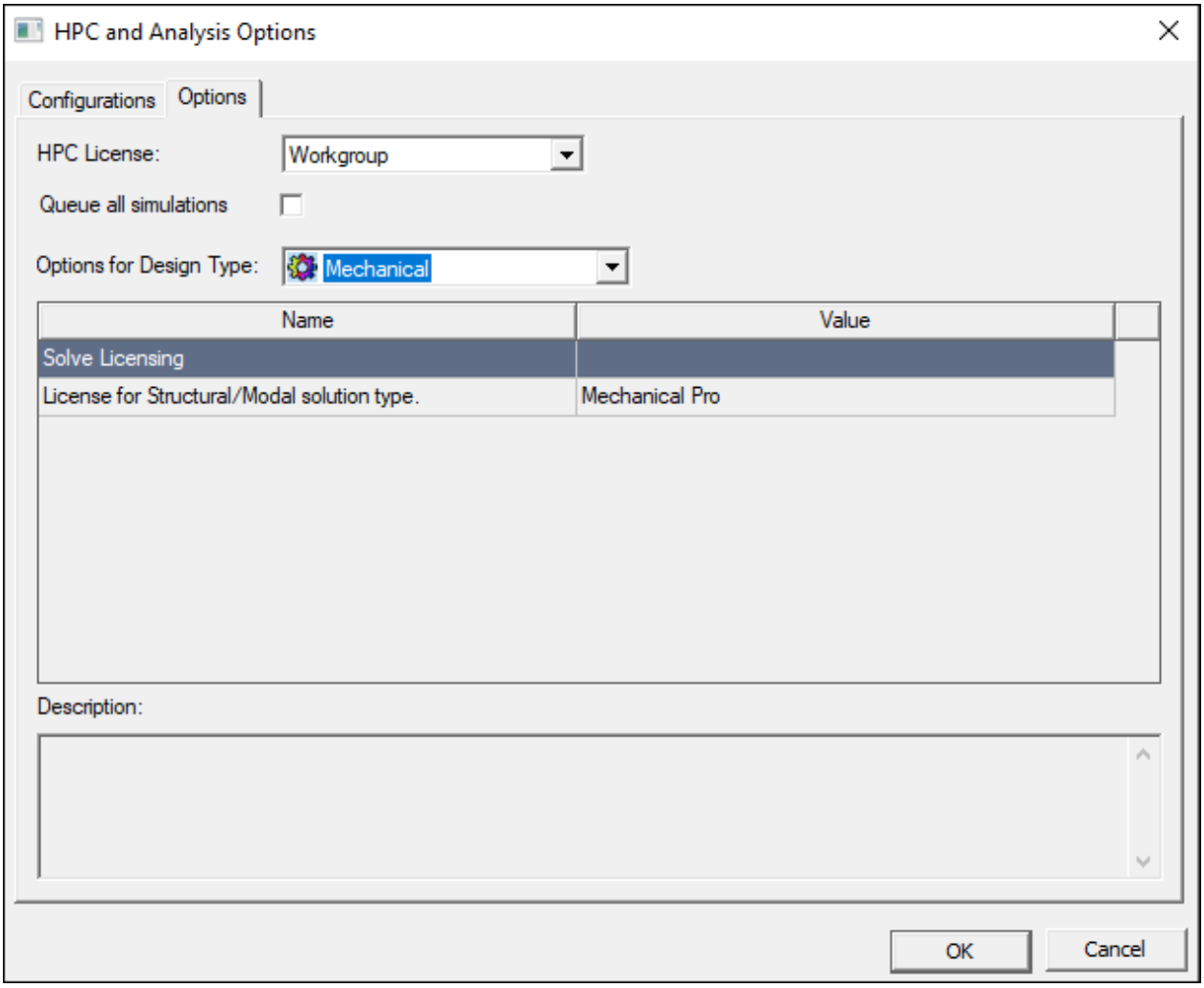
  > **Note:** Importing analysis configurations always adds the imported analysis configurations to the current design type. If there is a name conflict between an imported analysis configuration and an existing analysis configuration, the imported configuration is renamed and a notification appears.

- **Export** – allows you to export the selected analysis configurations to an *.acf file. You can then import the configurations into a different design type, or import them on a different machine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options Tab

The **Options** tab in the **HPC and Analysis Options** dialog box contains general and product-specific settings.



> **Important:**
>
> Available options vary by design type. Certain items discussed on this page may not be visible, depending on the currently selected design type.

These options are not specified for, or saved as part of, the current analysis configuration. Instead, they are global and are always in effect for the given design type when both of the following conditions are true:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A design of the matching design type is being solved.
- You have not specified corresponding overriding batch options on the command line.

From the **Options** tab, you can:

- Choose the HPC license type
- Enable queuing
- Specify Distributed Memory settings (for example, MPI for certain solvers)
- Set licensing options
- Enable GPU (for Transient, Matrix and SBR+ solves)
- Set the Default Process Priority

For **HPC License**, select **Auto**, **Workgroup** or **Pack**.

HPC licensing enables the use of cores and GPUs to accelerate simulations. In general, each core requires one unit of HPC, while each GPU requires eight units. The selected HPC license type determines which license is used, and how units of HPC are converted to license counts.

- **Workgroup** (formerly "pool") – One HPC workgroup license enables one unit of HPC.
- **Pack** – One HPC pack license enables eight units of HPC. Additional packs multiply by four, enabling 32, 128, 512,... , in the context of a single simulation.
- **Auto** – delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop.

Electronics Desktop products include four units of HPC for each licensed simulation. This means that up to four units can be used without requiring HPC licenses; license counting begins with the fifth unit. For example, a simulation that uses 36 cores requires 32 HPC units after subtracting the four included cores. This simulation will check out 32 HPC workgroup licenses, or two HPC pack licenses.

HPC licenses enable all parallel and distributed simulations, including distributed variations. Distributed variations require a single set of solver licenses, plus HPC to enable the variations.

For HPC Workgroup, distributing *N* variations requires 8*(*N*–1) workgroup licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* workgroup licenses will enable one additional HPC unit per variation. For HPC Pack, distributing *N* variations requires *N*–1 pack licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* pack licenses will enable 8, 32, 128,... additional HPC units per variation.

Ansys licensing supports distributed simulations when Electronics Desktop is called from other Ansys tools, such as optiSLang and Workbench. In such cases, distributed design points (variations) generally use HPC counts as described above.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Licensing for some calling products may include some distributed design points, in which case the total required HPC will be reduced.

If the **Queue all simulations** check box is selected, the Desktop queues any active simulations for design types that have **Save before solving** turned off in the **General Options** and then processes them in order. You can view and change the queue by using Show Queued Simulations.

For Mechanical designs, the following option is available under the heading of *Solve Licensing*:

License for Structural/Modal solution type, and the choices are as follows:

- **Mechanical Pro** (default)
- **Mechanical Premium**
- **Mechanical Enterprise**

Choose the license to use for solving Mechanical–Modal and Mechanical–Structural solutions. *Mechanical Pro* is the default selection.

> **Important:**
>
> Though all three of these licenses support the full scope of modal and structural solutions available for Mechanical designs in AEDT, you still **must** choose the specific type of license that has been purchased for your installation. Otherwise, structural and model solutions will fail to run.

To configure options:

1. For Linux authentication, specify the Remote Spawn command as RSH or SSH (the default).
2. Optionally, you can select one of the following from the **Default Process Priority** drop-down menu:
    - **Critical** (highest) (Not recommended)
    - **Above Normal** (Not recommended)
    - **Normal** (Default)
    - **Below Normal**
    - **Idle** (lowest)

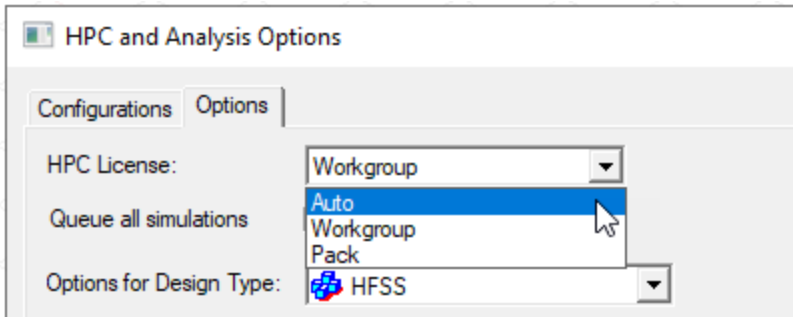
You can also set these values using VB or Python scripts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information on editing configurations, see Editing Distributed Machine Configurations for Mechanical.

Licensing Settings Tool

HPC licensing includes a choice called “Auto”, along with Workgroup and Pack. The “Auto” choice is available in the HPC License combobox found on the **Options** tab of the **HPC and Analysis Options** dialog box. If you make changes in the License Settings Tool, we recommend that you restart Electronics Desktop.



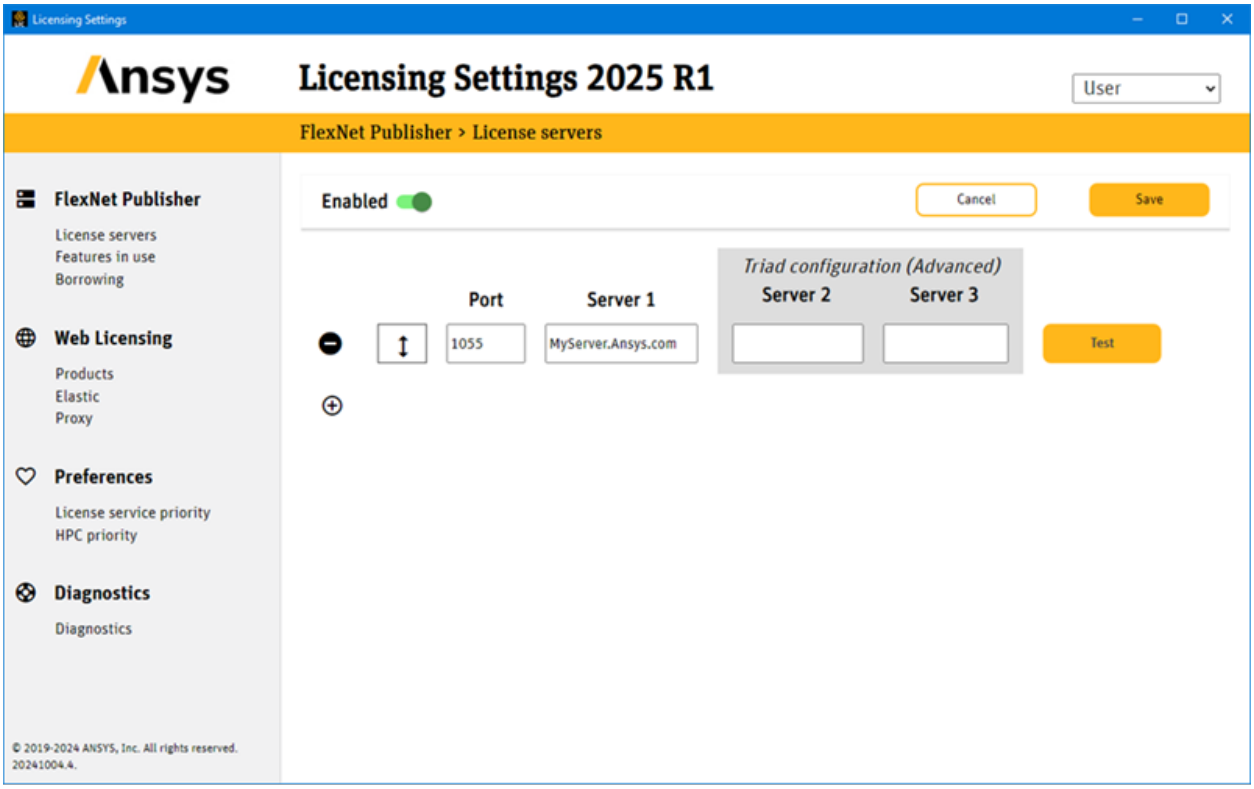
"Auto" means delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop, accessible from the **Tools** menu. The HPC settings specified by the Licensing Settings tool are shared by other Ansys products. This provides you a single place where you can set the HPC preferences.

> **Note:** Important: HPC licensing preferences in the Licensing Settings tool are used only when Auto is selected in Electronics Desktop.

Launching the Licensing Settings Tool from the Main Menu

Click **Tools** > **Licensing Settings** to launch the Licensing Settings tool.

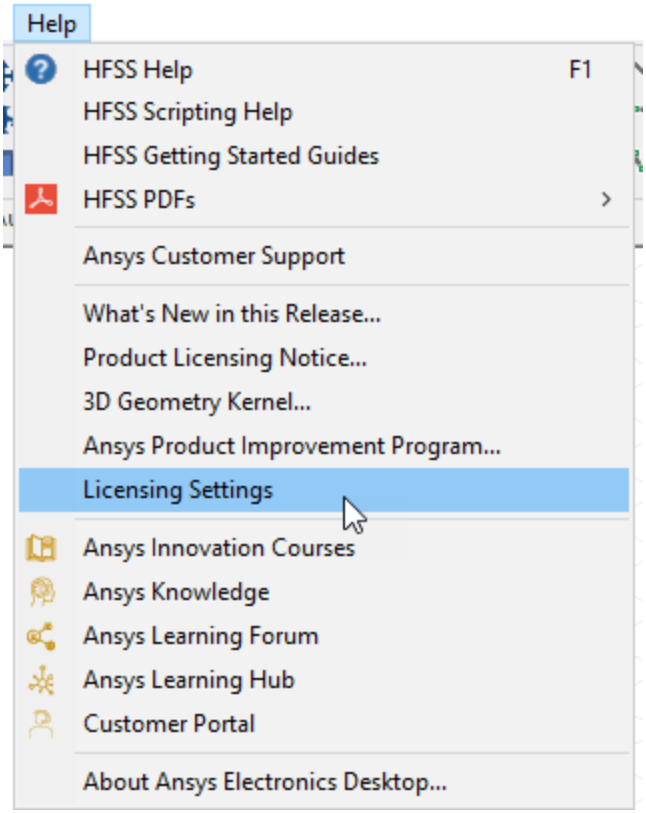
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Licensing Settings Help

You can obtain help with the Licensing Settings tool from the **Licensing Settings** menu item in the **Help** menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



License Fallbacks

When using Auto HPC licensing, in some cases if there aren't enough available licenses of the highest priority HPC license type, the lower priority HPC license type will be used. For instance, assume the user has set the priority order to "Ansys HPC, Ansys HPC Pack" in the Licensing Settings tool. If the user requests a nominal solve using 14 cores (10 Ansys HPC), but 10 Ansys HPC licenses aren't available, 2 Ansys HPC Pack licenses will be used if available.

Setting Options via Configuration Files

In addition to setting options from the Electronics Desktop user interface (UI), you can also set options in several configuration files. Option settings in configuration files may apply to all users or only to a specific user, and they may apply to all hosts or only to specific hosts. There are four levels of settings, listed below from most specific (highest precedence) to most general (lowest precedence):

- Host-dependent user options – apply to the specified user on the specified host only
- Host-independent user options – apply to the specified user on all hosts
- Host-dependent default options – apply to all users on the specified host
- Installation default – default for all users on all hosts

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the list above, settings at any level override settings at lower levels . If there is no setting in any file, the application default value is used. See UpdateRegistry for instructions on selecting these levels.

> **Important:**
>
> Options set from the Desktop UI override and update user settings in configuration files. Otherwise, the existing configuration file settings are used.

Behavior Examples

Consider running an application as user **jsmith** on host **host123**. If there is no host-dependent user setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for user **jsmith** on host **host123**, but there is a setting for the "Expand Project Tree on Insert" option in the host-independent user options config file for user **jsmith**, then the latter setting will be used if it is not overridden using the Desktop UI. Any settings in the host-dependent default options config file or the installation default config file are ignored.

As another example, consider running an application as user **jdoe** on host **host123**. If there is no setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for **jdoe** on **host123** or in the host-independent user options config file for user **jdoe** or in the host-dependent default options config file for host **host123**, then the value from the installation default config file are used, if present.

Rules for Modifying Option Settings

Option settings displayed in the Desktop UI follow the above rules. That is, if there is a setting in any of the option config files, then the setting from the highest priority config file is displayed in the Desktop UI. If there is no setting in any of the option config files, then the global default value is used. You can modify settings using the Options dialog boxes in the Desktop UI. If the dialog box is closed with the **Cancel** button, changes made on any of the tabs are discarded. If the dialog box is closed with the **OK** button, any settings that have been changed by the user are written to the host-dependent user options config file. The changed values written to this file are then used the next time that the application is run by the same user on the same host. The Desktop UI option settings are not written to any of the other option config files.

Configuration File Locations

The configuration files for host-dependent and installation **default** options reside at: "<*installation-directory*>\ANSYS Inc\v251\AnsysEM\config". The configuration files for host-dependent and host-independent **user** options reside in a subfolder of the user's Documents folder (for Windows) or the user's HOME folder (for Linux). See the tables below for specific Windows and Linux file names and paths.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Products with Multiple Desktop Versions

For products that have multiple Desktop versions, each will have a separate user-specific config directory, with a different value for the <*ApplicationName&Version*> directory name.

Table of Directories and Files

The following table shows the directories and files, where the *Level Name* is the name used to describe an options config file when using the UpdateRegistry tool.

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
host-dependent user options	user_machine	<*hostname*>_user.XML	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
host-independent user options	user	user.XML		
host-dependent default options	install_machine	<*hostname*>.XML	*"<Installation_Directory>*\ANSYS Inc\v251\AnsysEM\config"	*"<Installation_Directory>*/ansys_inc/v251/AnsysEM/config"
installation default	install	default.XML		

Note:

- **<*hostname*>** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- **<*ApplicationName&Version*>** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.1)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- **<*Installation_Directory*>** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- **<*Version*>** is the last two digits of the product version's year followed by the minor release number, without a decimal point (such as 251)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table shows an example of specific file names and directory names for a typical Ansys Electronics Desktop installation on Microsoft Windows and on Linux. These are the files that apply to software version **2022 R1**, user "**jsmith**," and hostname "**host123**":

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
host dependent user options	user_machine	host123_user.XML	C:\Users\jsmith\Documents\Ansoft\ ElectronicsDesktop2025.1\config	/home/jsmith/Ansoft/ ElectronicsDesktop2025.1/config
host independent user options	user	user.XML		
host dependent default options	install_machine	host123.XML	"C:\Program Files\ANSYS Inc\v251\ AnsysEM\config"	/opt/ansys_inc/v251/AnsysEM/config
installation default	install	default.XML		

Note:

As with the temporary file location configuration files, the settings in these options files have precedence in the following sequence: user_machine (highest precedence), user, install_machine, install (lowest precedence). The first time you start and then exit the application, the file at the "user_machine" level is created (<*hostname*>_user.XML). The other files are only created if you use the UpdateRegistry tool to specify an option at the "user," "install_machine," or "install" level. If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

UpdateRegistry is a command line tool used to modify option settings in the options config files. You can use this command to add, change or remove settings from any of the option config files. This tool is included in the installation directory of each product. This feature makes it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives.

The UpdateRegistry command has multiple command line formats, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following command line options are *mutually exclusive*:

- `-Set`
- `-Get`
- `-GetKeys`
- `-Delete`
- `-FromFile`

UpdateRegistry -Set Command

This command is used to add or modify an option setting in an option config file. If the option config file does not exist, it will be created. If the setting does not exist in the specified config file, it will be added. If the setting already exists in the specified config file, then the value will be changed to the specified value.

Example:

```
UpdateRegistry -Set -ProductName <name> -
RegistryKey <keyPath> -RegistryValue <value> [
-RegistryLevel <level> ]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory
`<value>`	The new value of the option, typically a string or a number. If the value contains spaces, it must be quoted.

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -Get Command

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is used to view an option value in an option config file. If the setting exists in the specified config file or files, then the value, the value type and the config file where the value was found will be reported. If no value is found, then that will also be reported.

Example:

```
UpdateRegistry -Get -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, then all config files are searched in order of precedence.

UpdateRegistry -GetKeys Command

This command is used to view the allowed key names for all of the option settings, or to view a subset of the key names that match a string. For each key displayed, the current value, if any, is also reported. If a key has a value in multiple config files, then only the highest precedence value is reported.

Example:

```
UpdateRegistry -GetKeys [ <pattern> ] -
ProductName <name> [ -Case ]
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"

Optional:

`<pattern>`	If no pattern is specified, then all allowed key names are reported. If a pattern is specified, then only keys that match the pattern are shown. For example, Settings/Project. If the name contains spaces, it must be quoted. By default, the pattern match is case insensitive.
`-Case`	If this command line option is specified, then the pattern match is case sensitive.

UpdateRegistry -Delete Command

This command is used to remove an option setting from an option config file. If the setting does not exist in the specified config file, the file will not be changed. If the setting exists in the specified config file, then it will be removed. A setting may need to be removed from an option config file, to allow the setting from a lower priority file to be used by the application.

Example:

```
UpdateRegistry -Delete -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -FromFile Command

You can use this form of the UpdateRegistry command to set multiple key-value pairs from a file with a single UpdateRegistry command. You specify the -FromFile command line option. This option must be followed by a filename. The file may contain multiple entries, where each entry contains a registry key and a registry value. The key-value pairs are added to the registry level specified by the -RegistryLevel command line option; if no -RegistryLevel is specified, then the default registry level (user_machine) is used.

UpdateRegistry File Format

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

The file format is similar to the -batchoptions file format. An example UpdateRegistry file is shown below:

```
$begin 'AddEntries'
  'TempDirectory'='C:/temp/AnsysEM'
  'Desktop/Settings/ProjectOptions/HPCLicenseType'='Pool'
$end 'AddEntries'
```

Additional notes on the file format:

- The file may contain an arbitrary number of entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.

Registry key pathname:

- The registry key pathname appears before the equal sign "=" on each line.
- Each registry key pathname must be enclosed in single quotes.
- This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help.

Registry value:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The registry value appears after the equal sign on each line.
- Integral registry values must not be enclosed in quotes.
- All other registry values are treated as strings, and must be enclosed in single quotes.
- The forward slash "/" may be used as a directory separator on Windows and Linux. The back slash "\" may be used as a directory separator on Windows only.
- The back slash "\" is used as an escape character in the value string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the value string. The back slash may be used to remove this special meaning, and include a single quote in the string.
- To use a back slash as a directory separator on Windows, it must be escaped. That is, a double back slash "\\" is used to denote a single directory separator.

Alternative UpdateRegistry File Format:

- Analysis Configuration File format, which is exported from the HPC and Analysis Options dialog box.

> **Note:**
>
> If a current registry does not exist, Ansys Electronics Desktop can detect earlier minor versions of same application on the same machine. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays allowing you to port the registry from an earlier version.

Example Uses for Export Options Features

The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. This section outlines some use cases enabled by this feature.

> **Note:**
>
> Functionality featured in the examples in this topic apply to multiple design types.

Options that Apply to All Users

In many cases, an Ansys Electromagnetics tool installation is administered and maintained by a single user or group and used by a number of other users or groups. The permissions of the Ansys Electromagnetics tool installation may be set so that the administrator may add, delete or modify files, but other users may only read or execute these files. The administrator may set the recommended option settings in the installation default config file and/or the host dependent

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default options config file. These config files reside within the installation directory hierarchy, and should generally have the same permissions as other Ansys Electromagnetics tool installation files. This allows that administrator to control these settings, but does not allow other users to add, remove, or change these settings.

Each user can override any of these settings, if needed. This may be done using the Desktop UI, which affects the host-dependent user options config file. It may also be done using the host-independent user options config file. If user has overridden an option setting in either of the user files, the user may revert back to the option settings provided by the administrator by removing the setting of the same option in the host-dependent user option config file and/or the host-independent user option config file.

For global defaults, the administrator may set a value in the installation default config file. These settings will to apply to all users on all hosts.

In some cases, there are significant differences between the capabilities of different hosts. The host-dependent default config file may be used to specify different default values on some hosts. Any setting in a host dependent default config file would affect all users running on the specified host. The installation default value is used if there is no value specified for the setting in the host-dependent default config file for the current host. Note that the host-dependent default config file is named <*hostname*>.xml, where *hostname* is the name of the host computer.

Example: Searching for a Registry Key Pathname

Both administrators and ordinary users may occasionally use the UpdateRegistry command line tool to add, change or delete settings. To use this tool, the registry key pathname must be known by the user. The -GetKeys option may be used to quickly search for a key pathname if some information is known about it. For example, if the administrator knows that there is a setting related to issuing warning messages when available disk space is low, but does not know the exact key name, the following command may list some of the keys related to disk space:

```
UpdateRegistry -GetKeys disk -ProductName ElectronicsDesktop2025.1
```

This will display a list of all keys that match the string "disk" case insensitively.

Typical output may look like the following:

```
Registry keys matching pattern <disk> case insensitively:
Desktop/Settings/ProjectOptions/DiskLimitForAbort: value is <0> at
level <user_machine>
```

Example: Setting an Installation Default Value

The normal default for the Options/General/Desktop Performance/Warn when available disk space is less than setting is 0 MB. If the administrator is concerned that running out of disk space might be a common problem, the administrator could set the installation default for the warn

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting setting to 1000 MB, for example. This limit would then apply to all users running on all hosts. The administrator could use the following command to change this setting for Ansys Electronics Desktop:

```
UpdateRegistry -Set -ProductName ANSYSElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryValue 1000
 -RegistryLevel install
```

Example: Setting a Host-Dependent Default Value

For this example, assume that all hosts have two cores, except for three hosts: bighost1, bighost2, and bighost3, which have eight cores each. The administrator has set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 2 in the installation default config file. The administrator may set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 8 in the host-dependent default config files for the three hosts having 8 cores: bighost1, bighost2 and bighost3. The administrator may log in to host bighost1 and run the following command to change this setting for the host-dependent default options config file for host bighost1:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/NumberOfProcessors
 -RegistryValue 8
 -RegistryLevel install_machine
```

To make this change for the other two hosts, the administrator would log in to bighost2 and bighost3, and run the same command on each of those hosts.

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

Consider the case in which Electronics Desktop was installed and the administrator initially did not set a value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the default installation config file. User jsmith (who always uses host jshost) wanted to be warned before disk space dropped to zero, so he set the Desktop/Settings/ProjectOptions/DiskLimitForAbort to 100 MB using the UI. This setting is recorded in the host dependent user options config file for host jshost and user jsmith. Now the administrator learns that many users are running into disk space issues, so that administrator sets the installation default value for the setting Desktop/Settings/ProjectOptions/DiskLimitForAbort to 1000 MB, as in the above example.

When user jsmith runs Electronics Desktop on host jshost, the disk limit is 100 MB, not 1000 MB, because the host-dependent user options config file overrides all of the other config files. User jsmith may revert to the administrator provided default by removing this setting from the host dependent user options config file for host jshost and user jsmith. The following command may be run by user jsmith on host jshost to remove this setting:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user_machine
```

If user jsmith had added a value for this setting to the host-independent user options config file, then user jsmith would also run the following command to remove this setting from the host-independent user options config file:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user
```

User Options and the UpdateRegistry Tool

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

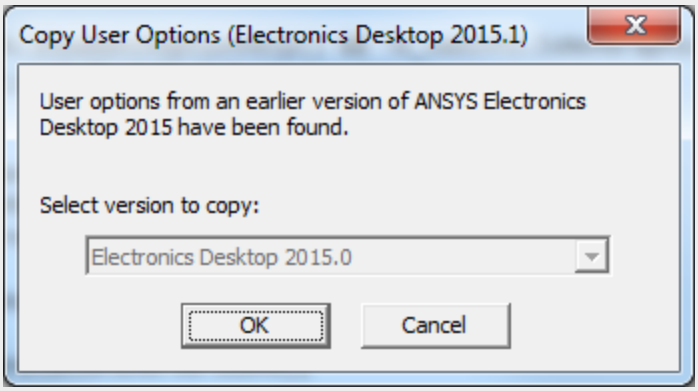
When you change an options value using the Desktop UI, the new value is stored in the host-dependent user options config file. You can use the UpdateRegistry tool to add or modify settings in the host-dependent user options config file; however, you cannot use the Desktop UI to remove settings from the host-dependent user options config file. *You must use the UpdateRegistry tool to remove settings from the host-dependent user options config file.*

If a user has not explicitly created a host-dependent user options config file or a host-independent user options config file, when they first run an Ansys Electromagnetics tool on a host, all settings will come from the host-dependent default options config file or the installation default options config file. Any settings for another host in a host-dependent user options config file will not be carried over to the new host. This may be inconvenient if the user has preferred option settings that differ from the settings that apply to all users, especially if the user runs the Ansys Electromagnetics tool on a number of different hosts. In this case, the user may set these option values in the user's host-independent user options config file. Then, these option values will be used on all new hosts, overriding any values set by the administrator to apply to all users. Any changes made in the UI will only affect the user's host-dependent user options config file for the current host.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If a current registry does not exist, the Ansys Electronics Desktop can detect earlier minor versions of same application on the same machine. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays from which you can select an earlier version from which the registry will be ported.



Copy over registry entries (both Windows and Ansys .xml files).

Example: Removing a Host-Dependent User Option Setting

For this example, user jsmith always uses host jshost to run Ansys Electronics Desktop. At some point, jsmith set the Autosave interval in **General Options** > **Project Options** tab to 1000 edits, and this value was written to the jsmith's host-dependent user options config file for host jshost. Now, jsmith wants to remove this setting and return to the default value of 10. User jsmith may run the following command on host jshost to remove the Desktop/Settings/ProjectOptions/AutoSaveInterval option value from this config file:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
-RegistryKey Desktop/Settings/ProjectOptions/AutoSaveInterval
-RegistryLevel user_machine
```

Example: Getting a Value from a Specific Configuration File

In the previous example, the user jsmith may decide to check the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the host-independent user configuration file before making any changes to this setting. The following command may be used to quickly view this setting before making the change:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.1 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort -
RegistryLevel user
```

Example: Getting a Value Using Precedence Rules

In may cases, the user is more interested in the value of a setting that will be applicable when running the product than in the setting in a single configuration file. If the -Get option is used with no -RegistryLevel specified, then the value reported is the value found in the highest precedence configuration file. If the user jsmith is interested in the highest precedence value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting, then the following command may be used to report this information:

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.1 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
```

Example: Adding a Host-Independent User Option Setting

Consider the case in which there is no value set for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting for all users for Ansys Electronics Desktop. The default is then 0 MB. User jsmith uses a variety of hosts and wants to be warned whenever disk space drops to 250 MB on any host. User jsmith may use the following command to set the Desktop/Settings/ProjectOptions/DiskLimitForAbort option value to 250 MB for all hosts:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.1
-RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
-RegistryValue 250 -RegistryLevel user
```

Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.1 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "m"

registry key is not defined in this product

Register value <3D Editors/Preferences/Geometry3D/q> is created with
value <m>
```

Example: Change a Newly Set Registry Key

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.1 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "n"
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Register value <3D Editors/Preferences/Geometry3D/q> is set to <n>
```

If you attempt to set a registry key to the wrong data type, a message like the following appears.

```
Cannot set the registry value to different data type
The work around is delete it, then create it again
error on to set registry value at <3D
Editors/Preferences/Geometry3D/q>
```

Setting the Temporary Directory

The temporary directory may be viewed or set using the Electronics Desktop user interface (UI), or from the command line.

To set the directory via the UI:

Navigate to **Tools** > **Options** > **General Options**. In the tree at the left side of the dialog box, expand the **General** branch and select **Directories**. Then, use the **Temp** field and **Override** check box to enter a desired directory path. Values set in this manner are written to the user_machine level configuration file for the temporary directory. If the **Override** check box is cleared, clicking **OK** changes the user_machine level setting for the temporary directory to an empty string. This enables settings from the next highest precedence configuration file. The file that provides the currently active temporary directory setting is shown under the **Temp** edit box.

To set the temporary directory from the command line, using the -batchoptions command line option. See: Running Ansys Electronics Desktop from a Command Line and -Batchoptions Command Line Examples.

The temporary directory may be configured with an installation default value, as well as a host-dependent default value, a host-independent user-specified value and a host-dependent user-specified value. Temporary directory settings are stored in different files from the other option settings. These files are located in the same directories as the configuration files for the other option settings. The following table shows the directories and files used to store temporary directory settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
Host-dependent, user-specific temporary directory	user_machine	<*hostname*>.cfg	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
Host-independent, user-specific temporary directory	user	default.cfg		
Host-dependent, default temporary directory	install_machine	<*hostname*>.cfg	"<*InstallationDirectory*>\ANSYS Inc\v251\AnsysEM\config"	<*InstallationDirectory*>/ansys_inc/v251/AnsysEM/config
Installation default temporary directory	install	default.cfg		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- ***<hostname>*** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- ***<ApplicationName&Version>*** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.1)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- ***<InstallationDirectory>*** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- ***<Version>*** is the last two digits of the product version's year followed by the minor release number, without a decimal point (such as 251)

As with other options, the settings in these files have precedence in the following sequence: user_machine (highest precedence), user, install_machine, install (lowest precedence). The installer creates the file at the "install" level (default.cfg). The first time you start and then exit the application, the file at the "user_machine" level is created (<*hostname*>.cfg). The other files are only created if you use the UpdateRegistry tool to specify an option at the "user" or "install_machine" level. If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Temporary Directory Configuration File Format

This section describes the format of temporary directory configuration files. The format is the same for files at all four levels: user_machine, user, install_machine, and install. These files are text files, so any text editor may be used to modify or create them.

An example temporary directory configuration file is shown below:

```
$begin 'Config'
tempdirectory='C:/TEMP/AnsysEM'
$end 'Config'
```

The temporary directory specified by this configuration file is C:/TEMP/AnsysEM.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- The string containing the pathname of the temporary directory must be enclosed in single quotes.
- The forward slash (/) may be used as a directory separator on Windows and Linux. The backslash (\) may be used as a directory separator on Windows only.
- The backslash (\) is used as an escape character in the tempdirectory string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the tempdirectory string. The backslash may be used to remove this special meaning, and include a single quote in the string.
- To use a backslash as a directory separator on Windows, it must be escaped. That is, a double backslash "\\" is used to denote a single directory separator.
- On Windows, a UNC path normally begins with two backslash characters. In a tempdirectory string, each of these backslash characters must be doubled, so four consecutive backslashes "\\\\" are used in the config file.

Example: Config File with UNC

```
$begin 'Config'
tempdirectory='\\\\hostxyz\\TEMP\\abc'
$end 'Config'
```

Here, hostxyz is a host with a sharename TEMP having subdirectory abc used as the temporarydirectory. This shows that four backslashes are required for UNC names and that backslashes used as directory separators must be doubled.

Example: Config File with Single Quote

```
$begin 'Config'
tempdirectory='C:/TEMP/ab\'cd'
$end 'Config'
```

Temporary directory is C:/TEMP/ab'cd. This shows how to include a single quote in a tempdirectory pathname. It also shows that forward slashes may be used as directory separators on Windows.

UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files

The UpdateRegistry command line tool, described above, may be used to view, add, change or remove the temporary directory setting from any of the temporary directory config files. The registry key for viewing or modifying the temporary directory is TempDirectory.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -Get, -Set, and -Delete options are valid for viewing, adding, changing, or deleting a temporary directory setting.

The -GetKeys option does not list the temporary directory key.

Batchoptions Command Line Examples

The `-batchoptions` entries command line argument may be used to specify one or more batchoptions settings from the command line. To specify multiple entries using a single `-batchoptions` argument, the entries should be enclosed in double quotes. Alternatively, the batchoptions may be specified in a file using the `-batchoptions <filename>` command line argument format. In this case, the filename is an absolute or relative pathname of the file containing the batchoptions, as described above.

> **Important:**
>
> The two approaches may not be combined: either all batchoptions must be in a file or all batchoptions must be specified explicitly on the command line.

Batchoptions File Format

> **Note:**
>
> Functionality featured in the example in this section applies to multiple design types.

An example batchoptions file is shown below:

```
$begin 'Config'
   'Desktop/ProjectDirectory'='C:/test/projects'
   'Desktop/Settings/ProjectOptions/NumberofProcessors'=2
$end 'Config'
```

Additional notes on the file format:

- The file may contain an arbitrary number of batchoption entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.
- The **Registry Key** appears before the equals sign (=) on each line and must be enclosed in single quotes ('). The registry key includes the key path and the name of the registry value.
- The registry value (or option value) appears after the equals sign on each line.
- Registry keys are case-insensitive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- There are two supported types of registry values—string and integer:
  - Each **string** value must be enclosed in single quotes (').
  - Do **not** enclose **interger** values in quotes.
- For file paths within string values, the forward slash (/) may be used as a directory separator on both Windows and Linux systems.
- Alternatively, on Windows only, the customary backslash (\ ) may be used as a directory separator. However, in string values, the backslash is used as an escape code for indicating special characters that cannot be typed directly (such as \n for a new line). Therfore, if you use the backslash as a directory path separator, each instance must be doubled (\\ ). An example is: '\\\\host3\\temp\\Ansoft'. In this case, each double backslash is interpreted as a single backslash (\\host3\temp\Ansoft).
- The single quote character (') normally ends a string value. If you need to include a single quote (or apostrophe) within a string, use the backslash-apostrophe (**\'**) escape sequence. For example, the string '%UserProfile%/Documents/Ansoft/John\'s_Files', is interpreted as: %UserProfile%/Documents/Ansoft/John's_Files.

Example -BatchOptions with -Remote (Windows)

> **Note:**
>
> Functionality featured in the examples in this section applies to multiple design types.

In this example, we run a batch HFSS analysis of project file project1.aedt. We want all temporary files and directories to be created in C:\temp\HFSS instead of using the default temporary directory. We decide that the analysis will be done on a remote host, at IP address 12.34.56.78. Because of limited memory on the remote host, we decide to run the analysis using only a single COM engine. Because the remote host has four cores, we decide to use four four processors for of the analysis. We can use the **-Remote** option to specify that there will be a single remote COM engine.

Here is a sample command line for this analysis, where the project file \\somehost\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Remote -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:/temp/HFSS'
'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
\\somehost\projects\project1.aedt
```

An alternative is to use the **-Distributed** command line option. Because the -Machinelist list contains only one host, there is a single remote COM engine in this case also.

```
ansysedt -BatchSolve -Distributed -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:\\temp\\HFSS'
 'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
 \\somehost\projects\project1.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The second line of the first example shows that you can use the forward slash "/" as a Windows directory separator within option value 'strings'. In this case, it is used in the TempDirectory path. You can also use the customary backslash "\" as a Windows directory separator, but it must be doubled to "\\" because the backslash is also an escape character within parameter strings. This usage is demonstrated in the second line of the second example, again in the TempDirectory path.

Example -Batchsolve with -Machinelist (Windows)

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

Suppose that we want to run a batch HFSS analysis of project file project1.aedt. Because all of our hosts have multiples of 2 cores, we specify that we will use two threads for multiprocessing for both the distributed (HFSS/NumCoresPerDistributedTask) and non-distributed (Desktop/Settings/ProjectOptions/NumberOfProcessors) parts of the job. The analysis contains a sweep that will be distributed across three hosts: ahmed, bill, and catherine. The hosts ahmed and bill have four cores each, so we run two distributed COM engines on each of these hosts, each using two threads. Host catherine has only two cores, so we specify only one distributed COM engine on this host. This COM engine will also use two threads.

> **Note:**
>
> The **-distributed** option and associated **NumCoresPerDistributedTask** parameter are not currently supported for Mechanical solutions.

Here is a sample command line for this analysis, where the project file \\dennis\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Distributed
-Machinelist list=ahmed,ahmed,bill,bill,catherine
-batchoptions "'Hfss/
'Desktop/Settings/ProjectOptions/NumberOfProcessors'
'Hfss/NumCoresPerDistributedTask'=2"
\\dennis\projects\project1.aedt
```

Example -Batchsolve with -Machinelist (Linux)

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, we run a batch HFSS analysis of project file project2.aedt. We have four identical host computers—host1, host2, host3, and host4 for analysis, and each host has 4 cores. We do not use multiprocessing for the distributed analysis, so NumCoresPerDistributedTask=1. As each host has four cores, we specify multiprocessing using 4 threads for the non-distributed part of the analysis, so NumberOfProcessors=4. Because we do not use multiprocessing for the distributed analysis, we will run four distributed COM engines on each host, with a singe core available for each engine.

> **Note:**
>
> The **-distributed** option and associated **NumCoresPerDistributedTask** parameter are not currently supported for Mechanical solutions.

Here is a sample command line for this analysis, where the project file /home/jsmith/projects/project2.aedt is located in a shared directory:

```
hfss -BatchSolve -Distributed
-Machinelist file=/home/jsmith/hosts/list2
-batchoptions "HFSS/
'Desktop/Setings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=1"
/home/jsmith/projects/project2.aedt
```

For this example, the hostnames are in the text file /home/jsmith/hosts/list2. The file contents are as follows:

```
host1
host1
host1
host1
host2
host2
host2
host2
host3
host3
host3
host3
host4
host4
host4
host4
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example -Batchsolve for Local (Windows)

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

In this example, we run a batch analysis of project file testproject.adsn on the local host. We want all temporary files and directories created in directory C:\temp\ansysedt instead of using the installation default temporary directory. Because the local host has four cores, we decide to use four threads for multiprocessing, for both distributed and non-distributed parts of the analysis.

Here is a sample command line for this analysis, where the project file \\host123\projects\testproject.adsn is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Local -batchoptions
"TempDirectory='C:/temp/ansysedt'
'Planar EM/SolverOptions/NumProcessors'=4
'Planar EM/SolverOptions/NumProcessorsDistrib'=4"
\\host123\projects\testproject.adsn
```

Note that the batchoptions pathnames 'Planar EM/SolverOptions/NumProcessors' and 'Planar EM/SolverOptions/NumProcessorsDistrib' must be in single quotes because they both contain embedded spaces.

Batch Options and Analysis Configurations in the Registry

Analysis configurations are used to specify machines and options for local, remote, and distributed analysis, including capabilities that are enabled by HPC licenses.

How Analysis Configurations are Stored in the Registry

A configuration contains information beyond the machine or machines to use for a solution. Examples are the number of processors to allocate to the analysis for each machine in the list, memory limits, directory locations for personal libraries and temporary files, and many other preferences.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software. For more information concerning the configuration files comprisisng the options registry, see the following topics:

  Setting Options Via Configuration Files
  Setting the Temporary Directory

- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

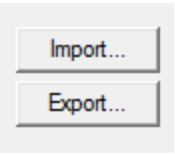
  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

General settings are associated with the Desktop application or 3D Editor. "HPCLicenseType" and "tempdirectory" values are at the root level of the registry. Other options are specific to a particular product or design type. For example, certain mesh, boundary, and memory limit settings as well as many other preferences are product-specific and are therefore associated only with the applicable design types. Such options appear with a consistent value name but in a different registry path for each applicable design type.

Copying a Configuration from one Design Type or Product to Another

To copy a configuration from one design type (or product) to another:

1. Click **Tools** > **Options** > **HPC and Analysis Options**. The *HPC and Analysis Options* dialog box appears.
2. On the **Configurations** tab of the **HPC and Analysis Options** dialog box, use the **Export…** button to export the configuration to a file.



3. Switch to the destination design type (or product) and use the **Import…** button to import the configuration data.

   Any data that is not applicable to the destination design type is ignored; any settings present in the destination design type that were not present in the source configuration will be assigned default values. The user may then edit the copy, as desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using HPC and Analysis for Configurations

Due to the complexity of the registry values for the configurations we do not recommend directly editing these values using the *UpdateRegistry* tool. Instead, use the **HPC and Analysis Options** dialog box to edit or create a configuration. (See: Setting HPC and Analysis Options.) Configurations created or edited using the GUI are stored in the **user_machine** level of the registry. You can create a configuration for one of the other registry levels using the following steps:

1. Create the configuration using the Analysis Configurations GUI, then export the configuration to a file.
2. Delete the configuration using the GUI so that it will not be present in the **user_machine** level. Then, exit the GUI.
3. Use the **UpdateRegistry** tool to import the data into the desired registry level using the **-FromFile** option to specify the file exported via the GUI, and using the **-RegistryLevel** option to specify the registry level where the configuration is to be stored. For example, an administrator may use this approach to create a configuration at the **install** level that may be used by any user on any machine.

Batch Options

There is a large number of both general and product-specific options supported by the software. These options have evolved over time. Therefore, older batchoptions files may no longer be valid, and the options listed in the user interface of the current software may differ from earlier versions you have used. Additionally, there are options beyond those listed within the user interface (that is, the UI provides a subset of the available options). You can generate a more complete list of options by running the UpdateRegistry tool with the -GetKeys switch and piping the output to a text file, as detailed in the following Windows procedure:

1. In a command window, navigate to the following folder:

   "<*installation_directory***>\ANSYS Inc\v251\AnsysEM"**

2. Type and enter the following command:

   UpdateRegistry -GetKeys -ProductName ElectronicsDesktop20*xx.y* **>** <*text_file_path*>**\Batchoptions.txt**

   Substitute the last two digits of the installed product version year and the minor release number for *xx.y* (such as **25.1**). Also, substitute the desired *text_file_path* (such as **%UserProfile%\Desktop**).

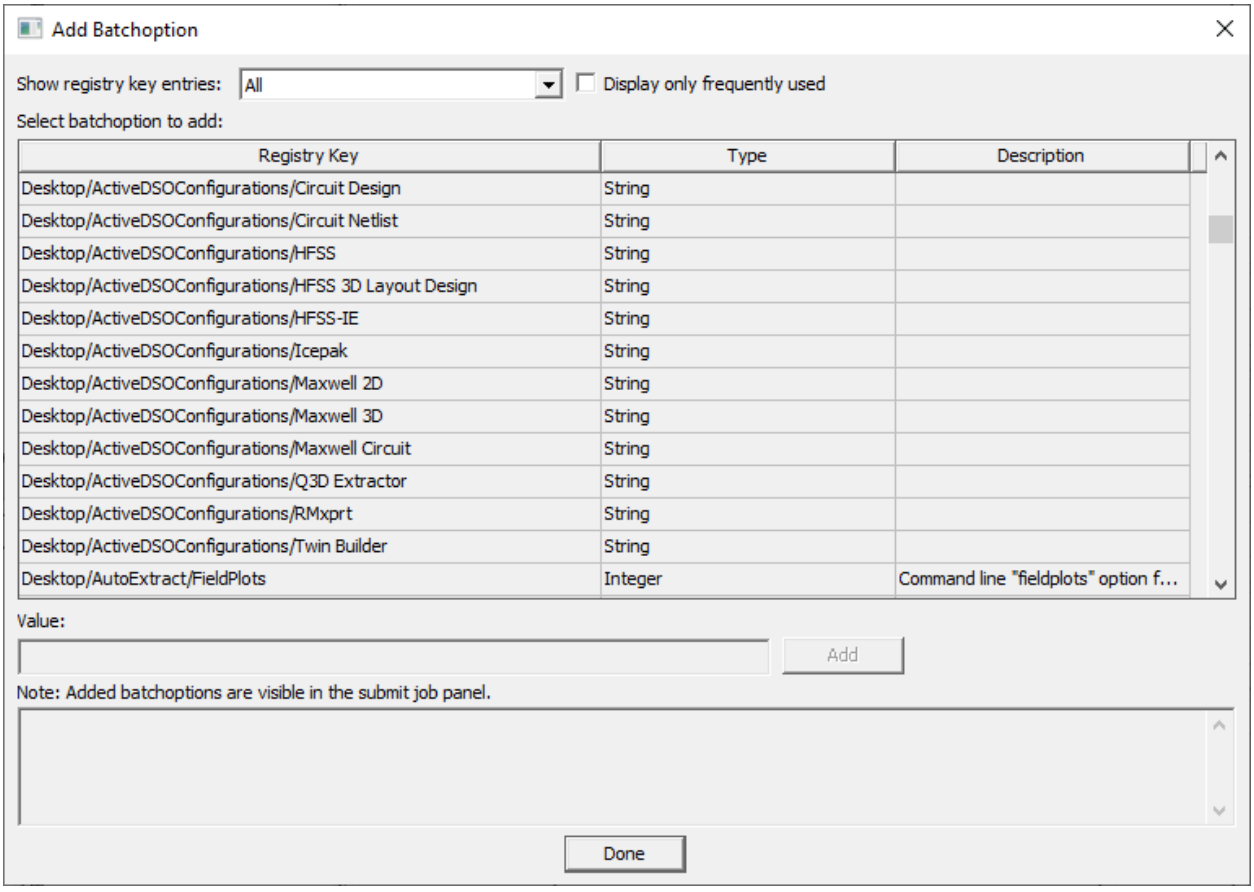
   The resulting **Batchoptions.txt** file will contain a nearly complete list of available options.

This procedure also works from the Linux <*installation_directory*>, which has the same downstream path (for example, under "opt/") as a Windows installation (typically under "C:\Program Files\").

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

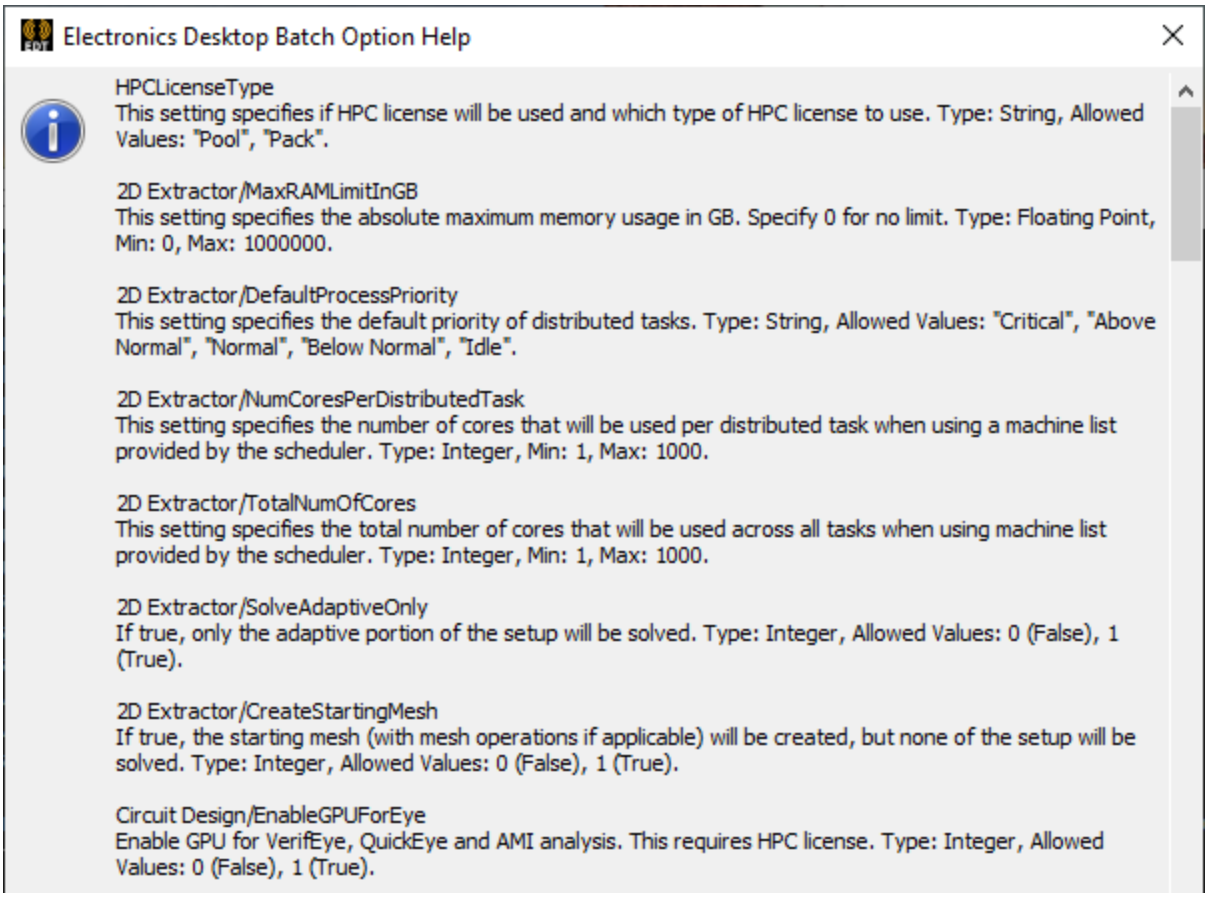
> **Note:**
>
> For special product-specific options that are neither available from the GUI nor listed by the UpdateRegistry tool, see Special Batch Options. Even though the -GetKeys switch does not list these special options, you can still use the UpdateRegistry tool to set them.

When you submit jobs to a remote computer or cluster, you can specify batch options using the job submission GUI. When using the GUI, you select the batch options from a list and therefore avoid typographical errors. For the most commonly used batch options, there is detailed information about the allowed values. Click **Submit** on the **Simulation** ribbon tab to access the *Submit Job To* dialog box. Then, click **Add** to access the **Add Batchoption** dialog box pictured below:



To assist users who need to specify batch options and are unable to use the job submission GUI, a new help option has been added. If the Electronics Desktop application is launched with the **-batchoptionhelp** command line argument, a message box is displayed which lists and describes the most common design type-specific batch options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Managing Projects and Designs

An Ansys Electronics Desktop *project* is essentially a folder that includes one or more models, or *designs*.

Some basic tasks that can be performed on projects and designs include:

- Opening Projects
- Creating Projects
- Saving Projects
- Importing and Exporting Projects and Data
- Setting Project Options
- Validating Projects
- Copying and Pasting Projects and Designs
- Renaming Projects
- Deleting Projects or Designs
- Setting Read Only Designs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

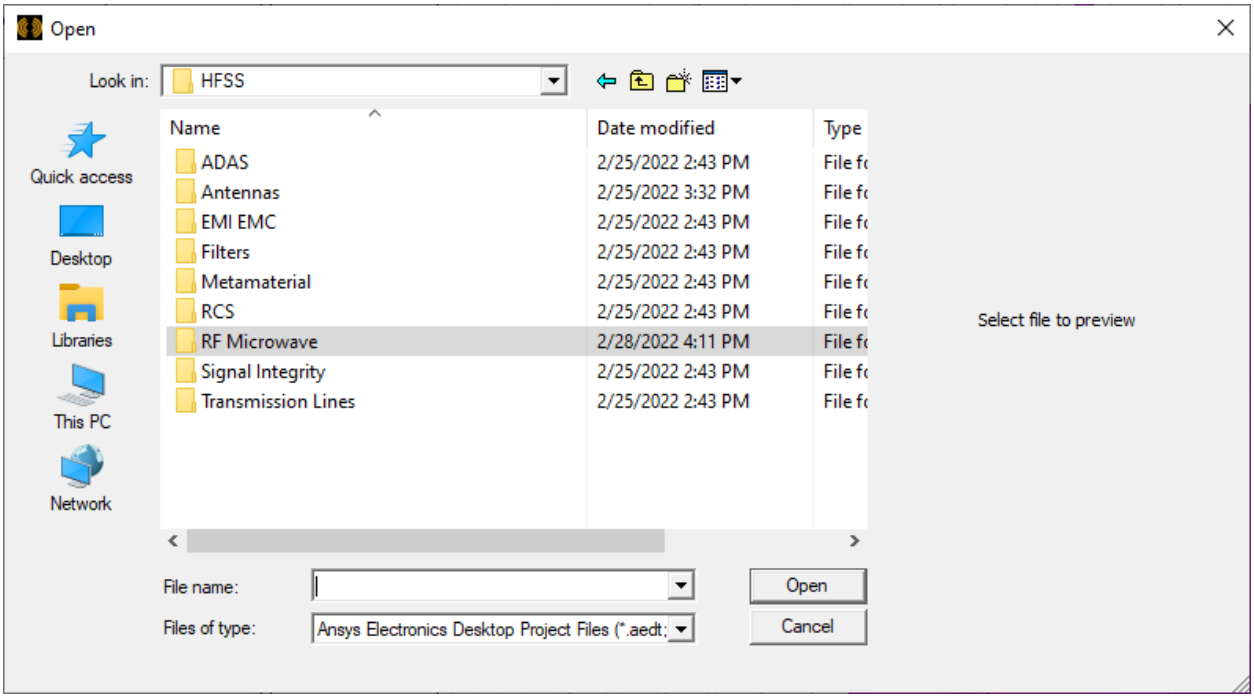
- Updating Design Components
- Undoing and Redoing Commands
- Inserting a Documentation File
- Saving Project Notes
- Printing

Opening Projects

Open a previously saved project using the **File** > **Open** command.

1. Click **File** > **Open** or click the **Open** icon on the **Desktop** tab of the ribbon.

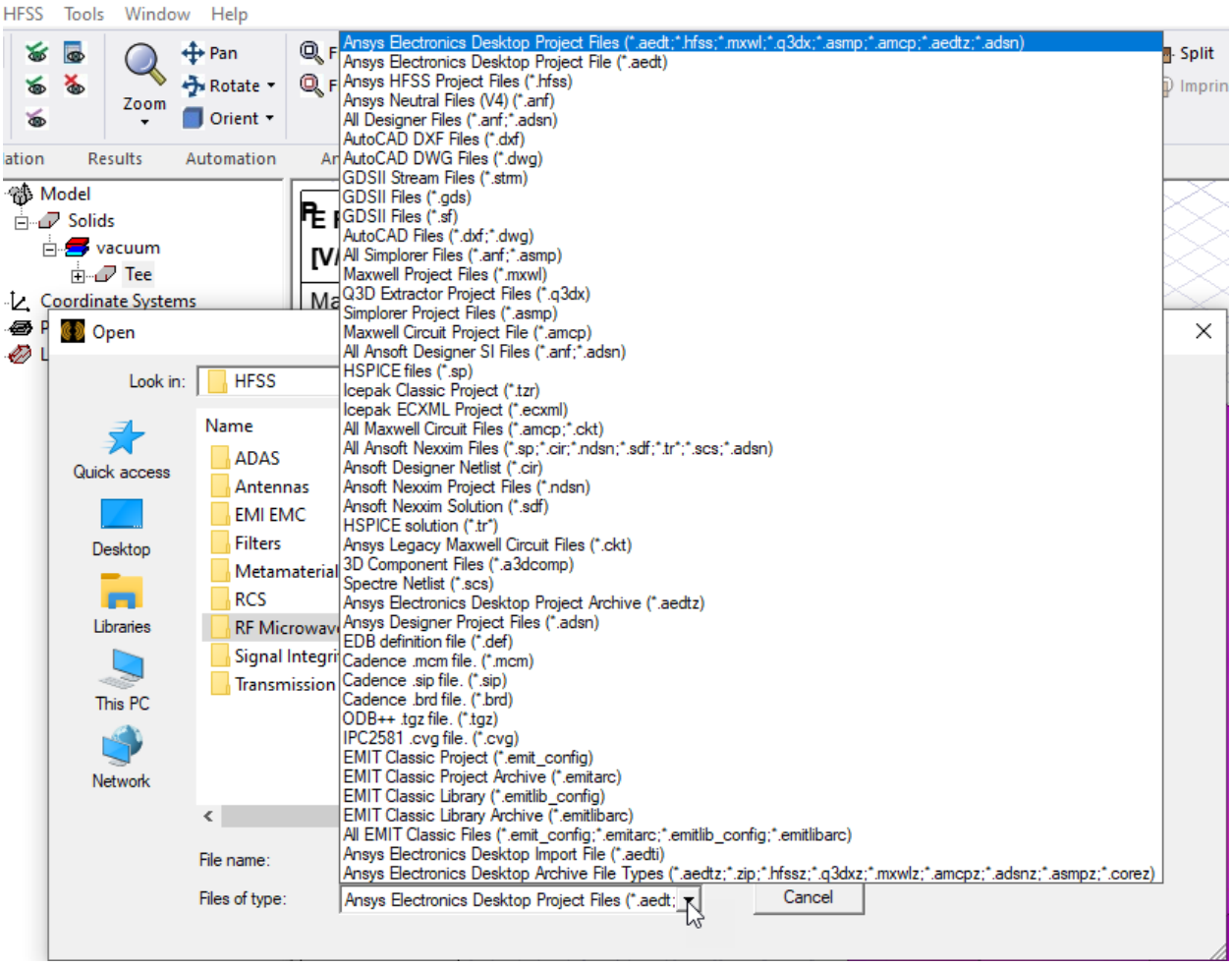
2. Use the file browser to find the project file.



By default, only files that can be opened or translated display. *.aedt is the main default, although, you can also view several types of legacy files. The dropdown for Files of type

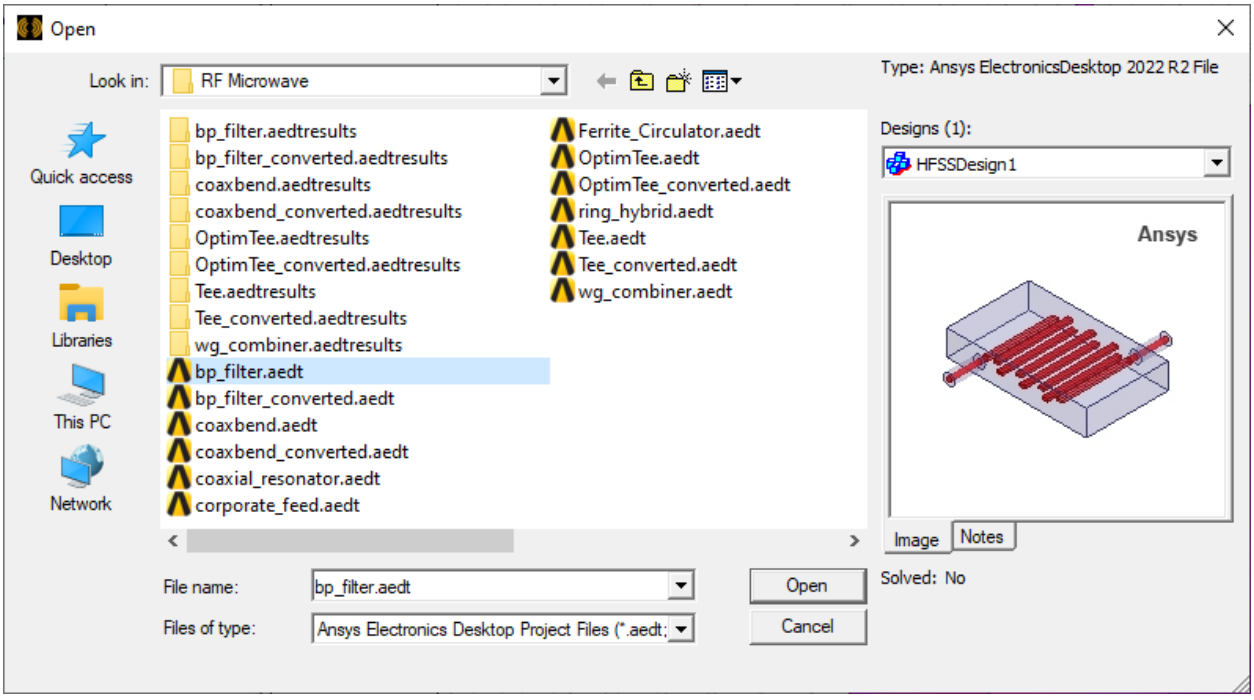
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lists the kinds file and archive formats for which you can select filters.



3. Most of these will be familiar to those seeking them. The Ansys Electronics Desktop Import file (*.aedti) format is for importing exported projects from ANSYS Discovery, an ANSYS Workbench tool. This format allows import of side by side *.aedti and the correponding geometry definition *.sat into the Ansys Electronics Desktop, and creates a ready-to-solve HFSS project.
4. Select the file you want to open.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

   The project information appears in the Project tree.

If you open another project without editing the automatically created project, the automatically created project is removed.

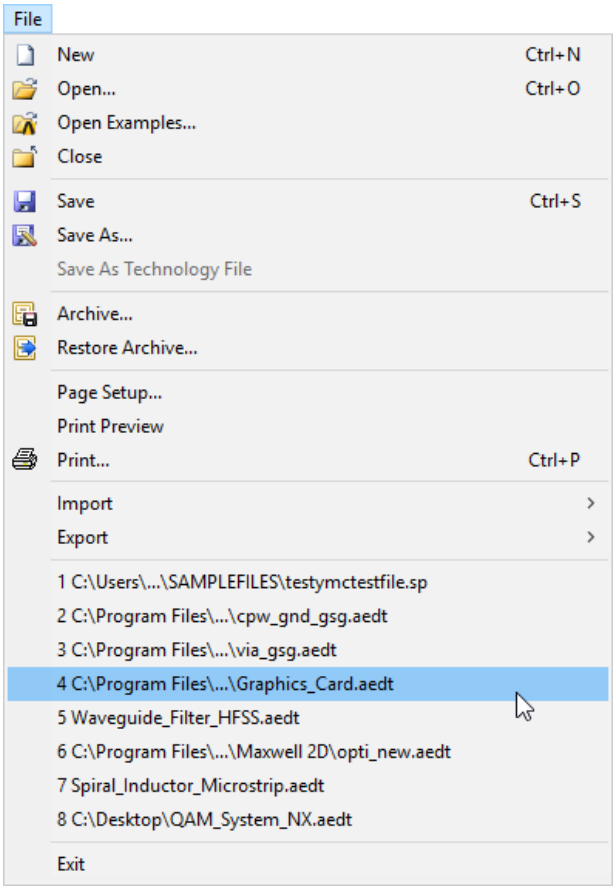
You can also open a saved project by:

- Dragging a project file icon to the Open icon
- Dragging a project file icon to the desktop
- Double-clicking a project file icon

Opening Recent Projects

Recently opened projects appear at the bottom of the **File** menu. To open one of these files, click its file name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Opening Example Projects

You can directly access and open example projects included with product installation using **File** > **Open Examples**, or by clicking **Open Examples** on the **Desktop** tab of the ribbon.
See Example Projects for additional description of these files.

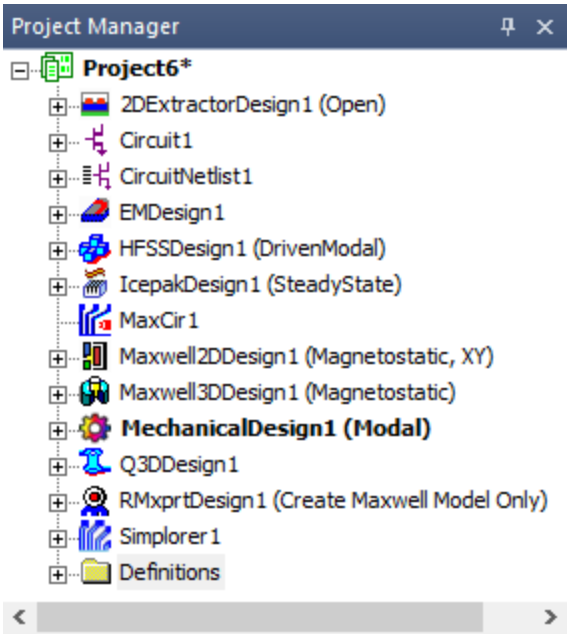
Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current project folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

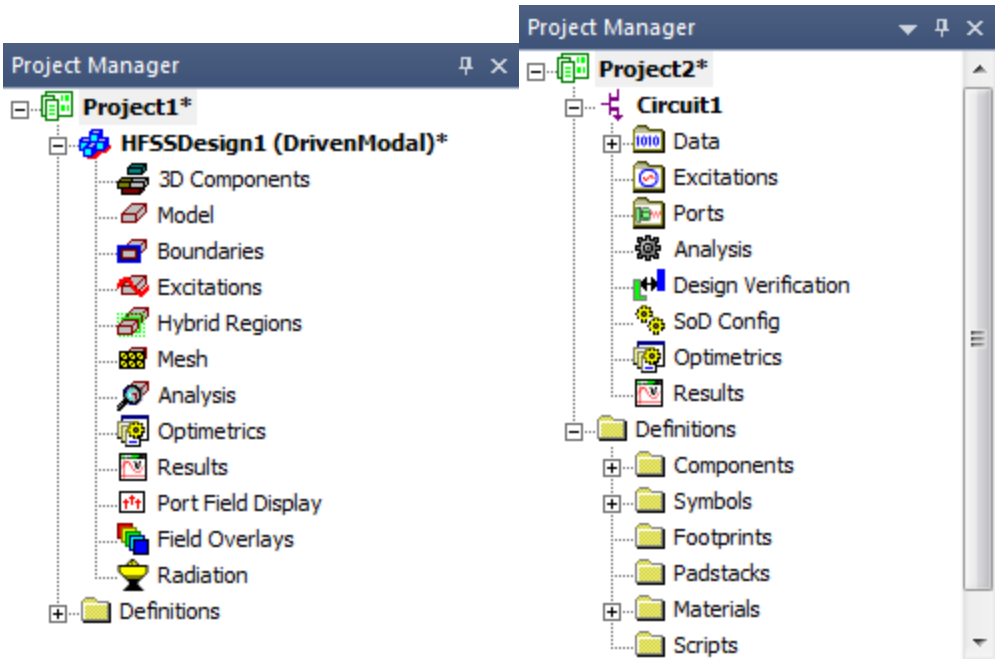
You can insert designs of any type into the project, where they are stored in the project tree.



The default name for each inserted project is <designType>*n*. You can also specify the name of the project when you save it using the **File** > **Save** or **File** > **Save As** commands.

You can view the contents of a project by clicking the plus sign (+) for each level of the hierarchy in the project or design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Target Design from 3D Designs

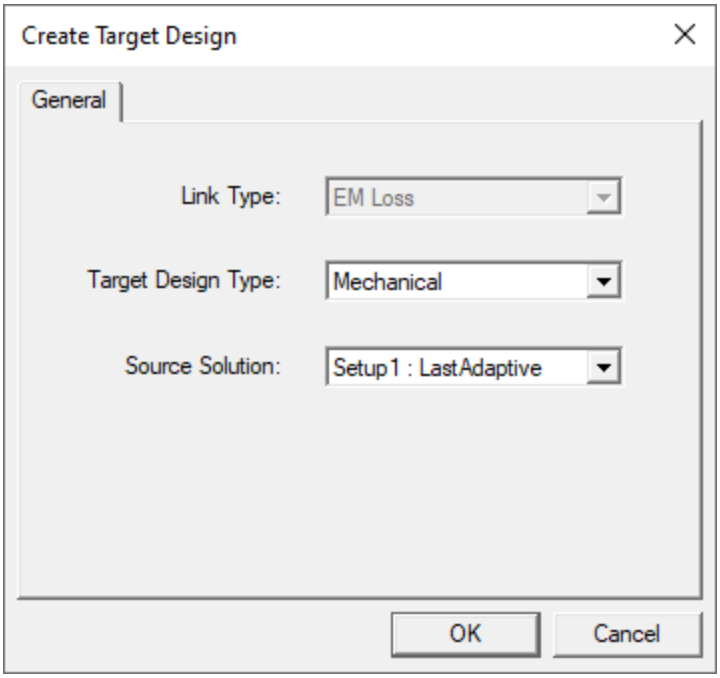
In an HFSS, Q3D, or Maxwell 3D design, **Create Target Design** automatically generates and inserts a thermal design (Icepak or Mechanical Thermal) into your project.

To create a target design from an existing **HFSS**, **Q3D Extractor**, or **Maxwell 3D** design:

1. Click **HFSS**, **Q3D Extractor**, or **Maxwell 3D** > **Create Target Design**.

   The **Create Target Design** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the **Target Design Type** to be created.
3. Select the **Source Solution** to specify the solution data to be linked to the target design.
4. Click **OK**.

The Mechanical Thermal design is generated based on the source design:

- Model geometry is created with corresponding material assignments. For any vacuum objects, **Solve Inside** is disabled.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Mechanical.
- A convection boundary is assigned to the faces of all 3D objects (not 2D sheet objects). **A Film Coefficient** is specified as **Uniform** at 10 w_per_m2kel. Objects with **Solve Inside** disabled and those with air assigned as the material are not assigned to the convection boundary.
- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.

Saving Projects

Use the **File** > **Save As** command or select the **Desktop** tab of the Ribbon and click the **Save As** icon to:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Save a new project.
- Save the active project with a different name or in a different location.
- Save the active project in another file format for use in another program.

Use the **File** > **Save** command or select the **Desktop** tab of the Ribbon and click the **Save** icon to save the active project.

> **Warning:**
>
> Be sure to save models periodically. Saving frequently helps prevent the loss of work if a problem occurs.
>
> Although Ansys Electronics Desktop has an auto-save feature, it may not save frequently enough for your needs.

Each solver has a Save Before Solving setting located in the **Tools** > **Options** window. This setting is enabled by default.

A prompt appears when you attempt to save a previously versioned file. If you agree to the prompt, the file is upgraded to the Ansys Electronics Desktop version in which you are running the software. In this case the file may no longer be compatible with previous versions. If you do not agree to the prompt, the file is not saved, so the file retains its previous compatibility.

If you have a simulation running, you see a warning that if you continue, Ansys Electronics Desktop will abort the simulation. If you OK the warning, Ansys Electronics Desktop aborts the simulation and saves the project.

Path Name Length Issues for Windows

For most Windows programs, the current directory pathname length is limited to 259 characters on startup. Thus, the pathname length of the directory containing installed programs should be limited to no more than 259 characters because double-clicking on an application in Windows Explorer will set the working directory to the directory containing the application when the application is started.

Win32 Long Paths Not Enabled

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If win32 long paths are not enabled on the analysis host, then essentially all files are limited to a maximum absolute pathname length of 259 characters. Directories may be limited to a maximum absolute pathname length of about 246 characters.

Win32 Long Paths Are Enabled

If win32 long paths are enabled on the analysis host, then many files are not subject to the maximum absolute pathname length limit of 259 characters. Below is a partial list of files that are still subject to this limit even if win32 Long Paths are enabled:

- Project files
- Project Archive files
- Ansys EDB (Electronics Database) files, typically stored in the `ProjectName.aedb` folder
- The temporary directory and most temporary files

  Because some temporary files and directories have automatically generated names that are long, it is best to use a short pathname for the temporary directory.

  > **Important:**
  >
  > Although most temporary files are created within the temporary directory, there may be some temporary files within the project directory. As a result, the project file directory pathname should be well below the 259 character limit.

To enable Long Path support on Windows, both of the following requirements must be met:

- Long paths must be enabled on the machine

  Use either of two ways to do this:

  - **Via Registry Setting:** `HKLM\SYSTEM\CurrentControlSet\Control\FileSystem : LongPathsEnabled=1`
  - **Via Group Policy Tool:** The “Enable Win32 long paths” setting is in the folder: `Computer Configuration\Administrative Templates\System\Filesystem`

Changing Auto-Save Settings

Recent actions performed on the active project are stored in an auto-save file in case of a sudden workstation crash or other unexpected problem. The auto-save file is stored in the same directory as the project file and is named Project*n*.aedt.auto by default. Ansys Electronics Desktop automatically saves all data for the project to the auto-save file, except solution data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, Ansys Electronics Desktop automatically saves project data after every 10 edits. An edit is any action performed that changes data in the project or the design, including actions associated with project management, model creation, and solution analysis.

After a problem occurs, you may be able to choose to re-open the original project file in an effort to recover the solution data, or to open the auto-save file. If the original file is not available, attempting to open the file provides a message that the auto-save is being used. If neither file is available, an error message displays.

To modify auto-save settings:

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

   The **Options** window appears.

2. Under **Desktop Configuration**, verify that **Do Autosave** is selected.

   This option is selected by default.

3. In the **Autosave interval** box, enter the number of *edits* that you want to occur between automatic saves. By default, this option is set to 10.

   > **Note:**
   >
   > Auto-save *always* increments forward; therefore, even when you undo a command, Ansys Electronics Desktop counts it as an edit.

4. Click **OK** to apply the specified auto-save settings.

   Once the specified number of edits is carried out, a "model-only" save will occur. This means that Ansys Electronics Desktop does not save solutions data or clear any undo/redo history. When Ansys Electronics Desktop auto-saves, an ".auto" extension is appended to the original project file name. For example, Project1.aedt will automatically be saved as Project1.aedt.auto.

> **Warning:**
>
> When you close or rename a project, Ansys Electronics Desktop deletes the auto-save file.

Save Before Solving Option

The **Save before solving** option forces a full save before running the solve and is enabled by default. For efficiency reasons, the project is saved only if it has been modified since its last save.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change this setting, click **Tools** > **Options** to open the Options window. Select the **General** tab and use the **Save before solving** check box to toggle the option.

You can start a new solve while running another without having to abort the running solve. If you start a solve while another solve is running and the Save before solving option is set, Ansys Electronics Desktop asks if you want solve without saving first. This lets you perform multiple solves, and if you have not edited the project in between solves, crash recovery will work.

Recovering Project Data from an Auto-Save File

Following a sudden workstation crash or other unexpected problem, you can recover project data from its auto-save file.

> **Warning:**
>
> If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

Importing and Exporting Projects and Data

Ansys Electronics Desktop can import and export a variety of file types.

Importing Files

You can import the following types of files into a Mechanical design:

- 2D model (GDSII) files
- 3D model files
- DXF and DWG format files

Object, material, and parameter names with non-ASCII characters are not allowed, and therefore, not allowed for data transfer. Such transfers fail and produce an error message.

For certain file types, the import dialog contains **Validation and Healing Options**, which are enabled by default.

These types are:

- ACIS SAB files (*.sab)
- ACIS SAT files (*.sat)
- Ansys 3D Modeler files (*.sm3)
- Autodesk Inventor files (*.ipt, *.iam)
- CATIA V4/V5 files (*.exp, *.model, *.CATpart, *.CATproduct)
- Creo Parametric files (*.prt, *.asm)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

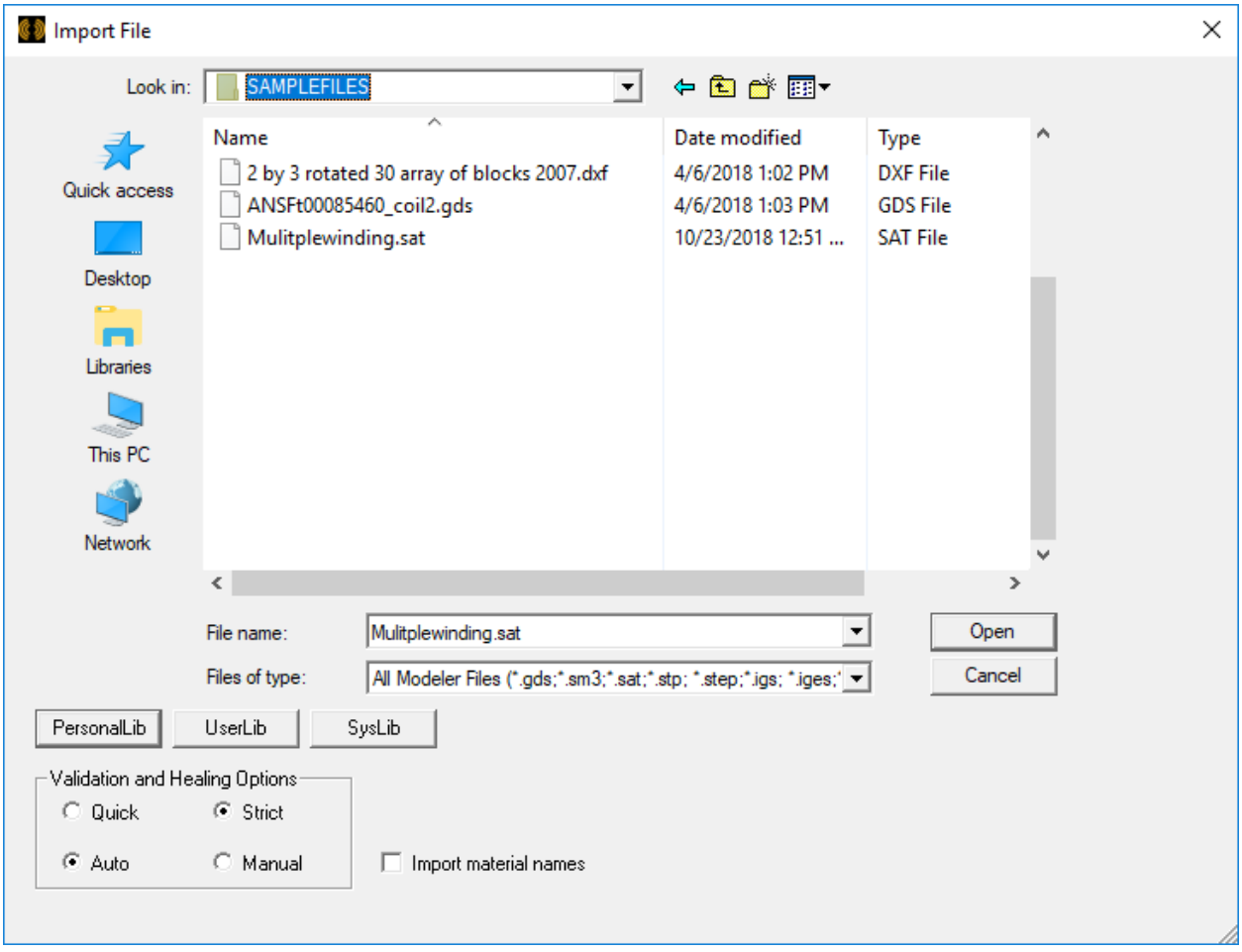
- IGES files (*.iges, *.igs)
- JT files (*.jt)
- Nastran files (*,nas)
- NX files (*.prt)
- Parasolid files (*.x_t, and *.x_b)
- SOLIDWORKS Files (*.SLDPRT, *.SLDASM)
- STEP files (*.step, *.stp)

For more information, see Heal.

To import a file:

1. Click **Modeler > Import**.

   The **Import File** window appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If applicable to the file type, **Validation and Healing Options** are enabled by default. You can change them as desired.

You can also use the check box to choose whether to **Import material names**.

2. Click **Open**.
3. Depending on the file type, additional options may appear. Set these as appropriate.
4. Click **OK**.

   The file loads.

Importing 2D Model (GDSII) Files

2D model files can be imported directly into the active **Modeler** window. GDSII is the industry standard file format for 2D graphical design layout data.

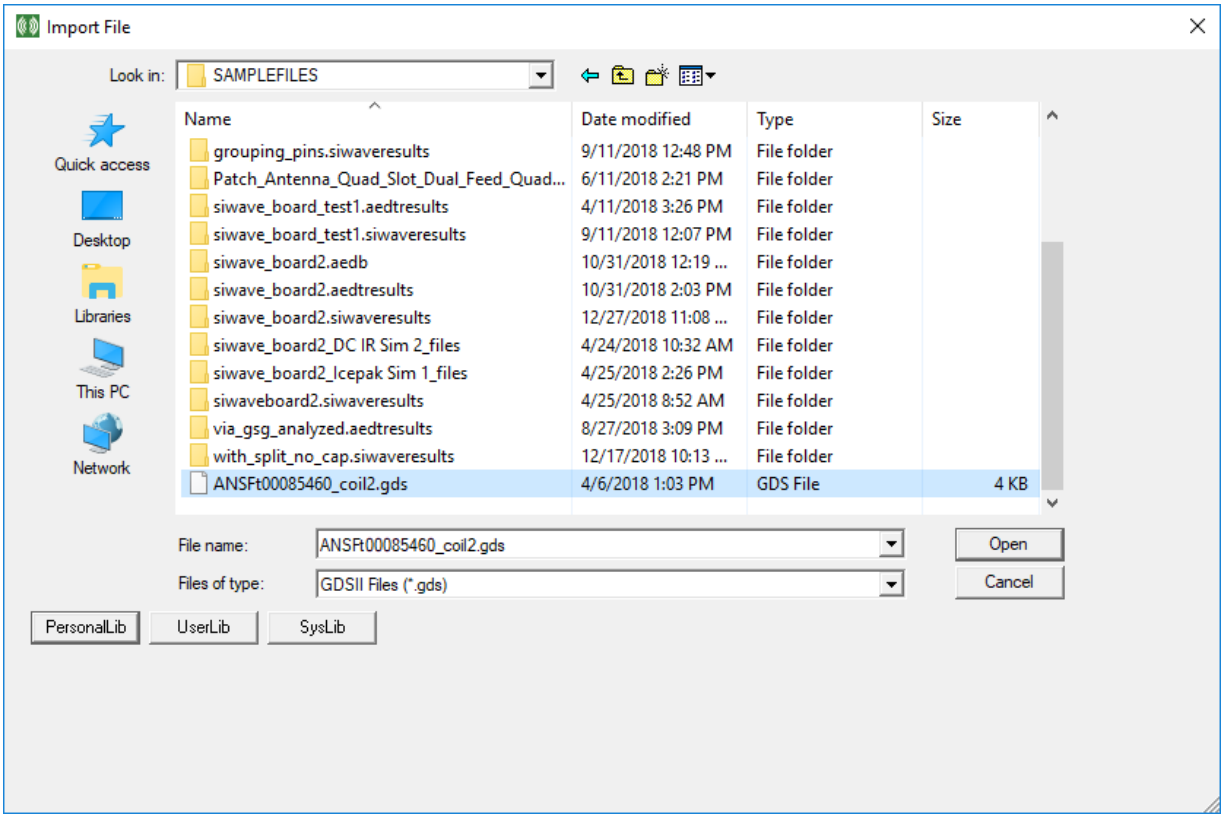
> **Note:**
>
> - Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
> - If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 2D model file:

1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

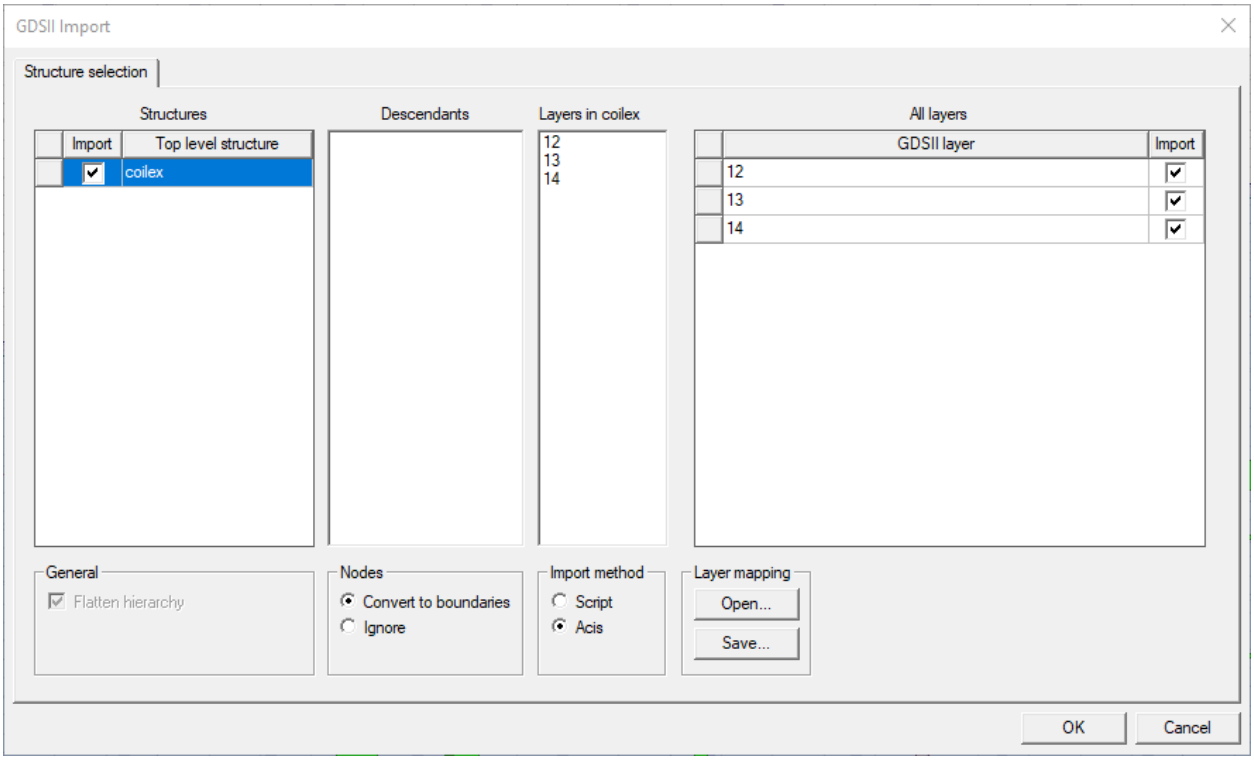
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a file type from the **Files of type** drop-down menu. For 2D model files, this is **GDSII Files (*.gds)**.
3. Use the file browser to find and select the file you want to import.
4. Click **Open**.

   The **GDSII Import** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the panes to select your desired options:
    - **Structure Selection** – a GDSII file may contain several top-level structures. Use this pane to select structure(s) for import. When multiple structures are imported, Mechanical creates multiple designs under the current project, one for each of the GDSII structures.
    - **Descendants** – GDSII files are hierarchical and may contain many sub-layouts. This panel shows the sub-layouts in the selected top-level designs.
    - **Layers in [Structure Name]** – this panel's name changes based on the name of the selected structure, and displays that structure's layers, by layer number.
    - **All Layers** – lists all layers from all structures in the file. Use the **Import** check boxes to select or deselect layers for import.
    - **General** – the **Flatten hierarchy** check box is selected by default. Mechanical always flattens any hierarchical geometry in the GDSII.
    - **Nodes** – GDSII supports nodes and boundaries as separate data types. Normally, boundaries represent polygons. Mechanical can either convert objects that use the nodes data type to boundary types, or ignore them. Use the radio buttons to select either **Convert to boundaries** (default) or **Ignore**.
    - **Import Method** – select either Script or Acis.
    - **Layer Mapping** – if desired, create a mapping of GDSII layer numbers to layer names in the design stackup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK**.

   The file is imported into the active **Modeler** window.

Creating GDSII Layer Mapping Files

While importing a 2D model (GDSII) file, you can create a mapping of the GDSII layer numbers to layer names in the design stackup.

To create and use the mapping file:

1. Use a text editor to create a text file that maps GDSII layer numbers to layer names in the stackup. The layer mapping file must have a ***.tech** suffix.

   In *.tech format:

   - / is the comment character
   - Units may be specified with a line UNITS <string> before the lines of layer information. <string> is any of the allowed desktop length units. The default unit is nm.
   - Each layer is specified by a line that contains <*import layer*> <*product layer*> <*layer color*> <*layer elevation*> <*layer thickness*>

   where:

   - <*import layer*> is the name of the DXF layer
   - <*destination layer*> is the name to map the DXF layer to
   - <*layer color*> is a string from the choices listed here
   - <*layer elevation*> is specified in nms
   - <*layer height*> is specified in nms

   For example:

```
/ ------------------------------------------------------------
/ import#  destination     Color    Elevation  Thickness
/ ------------------------------------------------------------
S12         signal12         blue     1100       530
TR          trace            red      6620       530
S3          signal3          yellow   8150       2000
```

2. Save the *.tech file locally.
3. During import, click **Open** in the **Layer mapping** panel to locate and open the existing layer mapping file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

   The file is imported into the active **Modeler** window.

Tech File Colors with RGB Values

Consult the table below for color string choices and their corresponding RGB values.

> **Note:**
>
> - Colors listed as two words can also be entered in Pascal case (e.g., GhostWhite is a valid substitute for ghost white).
> - Gray can be spelled using either the American or British spelling (Grey).

Color String(s)	RGB
snow1, snow	{ 255, 250, 250 }
snow2	{ 238, 233, 233 }
snow3	{ 205, 201, 201 }
snow4	{ 139, 137, 137 }
ghost white	{ 248, 248, 255 }
white smoke	{ 245, 245, 245 }
gainsboro	{ 220, 220, 220 }
floral white	{ 255, 250, 240 }
old lace	{ 253, 245, 230 }
linen	{ 250, 240, 230 }
antique white	{ 250, 235, 215 }
AntiqueWhite1	{ 255, 239, 219 }
AntiqueWhite2	{ 238, 223, 204 }
AntiqueWhite3	{ 205, 192, 176 }
AntiqueWhite4	{ 139, 131, 120 }
papaya whip	{ 255, 239, 213 }
blanched almond	{ 255, 235, 205 }
bisque1, bisque	{ 255, 228, 196 }
bisque2	{ 238, 213, 183 }
bisque3	{ 205, 183, 158 }
bisque4	{ 139, 125, 107 }
PeachPuff1, peach puff	{ 255, 218, 185 }
PeachPuff2	{ 238, 203, 173 }
PeachPuff3	{ 205, 175, 149 }
PeachPuff4	{ 139, 119, 101 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
NavajoWhite1, navajo white	{ 255, 222, 173 }
NavajoWhite2	{ 238, 207, 161 }
NavajoWhite3	{ 205, 179, 139 }
NavajoWhite4	{ 139, 121, 94 }
moccasin	{ 255, 228, 181 }
cornsilk1, cornsilk	{ 255, 248, 220 }
cornsilk2	{ 238, 232, 205 }
cornsilk3	{ 205, 200, 177 }
cornsilk4	{ 139, 136, 120 }
ivory1, ivory	{ 255, 255, 240 }
ivory2	{ 238, 238, 224 }
ivory3	{ 205, 205, 193 }
ivory4	{ 139, 139, 131 }
LemonChiffon1, lemon chiffon	{ 255, 250, 205 }
LemonChiffon2	{ 238, 233, 191 }
LemonChiffon3	{ 205, 201, 165 }
LemonChiffon4	{ 139, 137, 112 }
seashell1, seashell	{ 255, 245, 238 }
seashell2	{ 238, 229, 222 }
seashell3	{ 205, 197, 191 }
seashell4	{ 139, 134, 130 }
honeydew	{ 240, 255, 240 }
honeydew2	{ 224, 238, 224 }
honeydew3	{ 193, 205, 193 }
honeydew4	{ 131, 139, 131 }
mint cream	{ 245, 255, 250 }
azure1, azure	{ 240, 255, 255 }
azure2	{ 224, 238, 238 }
azure3	{ 193, 205, 205 }
azure4	{ 131, 139, 139 }
alice blue	{ 240, 248, 255 }
lavender	{ 230, 230, 250 }
LavenderBlush1, lavender blush	{ 255, 240, 245 }
LavenderBlush2	{ 238, 224, 229 }
LavenderBlush3	{ 205, 193, 197 }
LavenderBlush4	{ 139, 131, 134 }
MistyRose1, misty rose	{ 255, 228, 225 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
MistyRose2	{ 238, 213, 210 }
MistyRose3	{ 205, 183, 181 }
MistyRose4	{ 139, 125, 123 }
DarkSlateGray1	{ 151, 255, 255 }
DarkSlateGray2	{ 141, 238, 238 }
DarkSlateGray3	{ 121, 205, 205 }
DarkSlateGray4	{ 82, 139, 139 }
dark slate gray	{ 47, 79, 79 }
SlateGray1	{ 198, 226, 255 }
SlateGray2	{ 185, 211, 238 }
SlateGray3	{ 159, 182, 205 }
SlateGray4	{ 108, 123, 139 }
slate gray	{ 112, 128, 144 }
light slate gray	{ 119, 136, 153 }
midnight blue	{ 25, 25, 112 }
navy, navy blue	{ 0, 0, 128 }
cornflower blue	{ 100, 149, 237 }
slate blue	{ 106, 90, 205 }
medium slate blue	{ 123, 104, 238 }
light slate blue	{ 132, 112, 255 }
SlateBlue1	{ 131, 111, 255 }
SlateBlue2	{ 122, 103, 238 }
SlateBlue3	{ 105, 89, 205 }
SlateBlue4	{ 71, 60, 139 }
dark slate blue	{ 72, 61, 139 }
medium blue	{ 0, 0, 205 }
royal blue	{ 65, 105, 225 }
RoyalBlue1	{ 72, 118, 255 }
RoyalBlue2	{ 67, 110, 238 }
RoyalBlue3	{ 58, 95, 205 }
RoyalBlue4	{ 39, 64, 139 }
blue1, blue	{ 0, 0, 255 }
blue2	{ 0, 0, 238 }
blue3	{ 0, 0, 205 }
blue4, dark blue	{ 0, 0, 139 }
DodgerBlue1, dodger blue	{ 30, 144, 255 }
DodgerBlue2	{ 28, 134, 238 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
DodgerBlue3	{ 24, 116, 205 }
DodgerBlue4	{ 16, 78, 139 }
DeepSkyBlue1, deep sky blue	{ 0, 191, 255 }
DeepSkyBlue2	{ 0, 178, 238 }
DeepSkyBlue3	{ 0, 154, 205 }
DeepSkyBlue4	{ 0, 104, 139 }
sky blue	{ 135, 206, 235 }
SkyBlue1	{ 135, 206, 255 }
SkyBlue2	{ 126, 192, 238 }
SkyBlue3	{ 108, 166, 205 }
SkyBlue4	{ 74, 112, 139 }
light sky blue	{ 135, 206, 250 }
LightSkyBlue1	{ 176, 226, 255 }
LightSkyBlue2	{ 164, 211, 238 }
LightSkyBlue3	{ 141, 182, 205 }
LightSkyBlue4	{ 96, 123, 139 }
steel blue	{ 70, 130, 180 }
SteelBlue1	{ 99, 184, 255 }
SteelBlue2	{ 92, 172, 238 }
SteelBlue3	{ 79, 148, 205 }
SteelBlue4	{ 54, 100, 139 }
light steel blue	{ 176, 196, 222 }
LightSteelBlue1	{ 202, 225, 255 }
LightSteelBlue2	{ 188, 210, 238 }
LightSteelBlue3	{ 162, 181, 205 }
LightSteelBlue4	{ 110, 123, 139 }
light blue	{ 173, 216, 230 }
LightBlue1	{ 191, 239, 255 }
LightBlue2	{ 178, 223, 238 }
LightBlue3	{ 154, 192, 205 }
LightBlue4	{ 104, 131, 139 }
powder blue	{ 176, 224, 230 }
pale turquoise	{ 175, 238, 238 }
PaleTurquoise1	{ 187, 255, 255 }
PaleTurquoise2	{ 174, 238, 238 }
PaleTurquoise3	{ 150, 205, 205 }
PaleTurquoise4	{ 102, 139, 139 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
dark turquoise	{ 0, 206, 209 }
medium turquoise	{ 72, 209, 204 }
turquoise	{ 64, 224, 208 }
Turquoise1	{ 0, 245, 255 }
Turquoise2	{ 0, 229, 238 }
Turquoise3	{ 0, 197, 205 }
Turquoise4	{ 0, 134, 139 }
cyan1, cyan	{ 0, 255, 255 }
cyan2	{ 0, 238, 238 }
cyan3	{ 0, 205, 205 }
cyan4, dark cyan	{ 0, 139, 139 }
LightCyan1, light cyan	{ 224, 255, 255 }
LightCyan2	{ 209, 238, 238 }
LightCyan3	{ 180, 205, 205 }
LightCyan4	{ 122, 139, 139 }
cadet blue	{ 95, 158, 160 }
CadetBlue1	{ 152, 245, 255 }
CadetBlue2	{ 142, 229, 238 }
CadetBlue3	{ 122, 197, 205 }
CadetBlue4	{ 83, 134, 139 }
medium aquamarine	{ 102, 205, 170 }
aquamarine1, aquamarine	{ 127, 255, 212 }
aquamarine2	{ 118, 238, 198 }
aquamarine3	{ 102, 205, 170 }
aquamarine4	{ 69, 139, 116 }
dark green	{ 0, 100, 0 }
dark olive green	{ 85, 107, 47 }
DarkOliveGreen1	{ 202, 255, 112 }
DarkOliveGreen2	{ 188, 238, 104 }
DarkOliveGreen3	{ 162, 205, 90 }
DarkOliveGreen4	{ 110, 139, 61 }
dark sea green	{ 143, 188, 143 }
DarkSeaGreen1	{ 193, 255, 193 }
DarkSeaGreen2	{ 180, 238, 180 }
DarkSeaGreen3	{ 155, 205, 155 }
DarkSeaGreen4	{ 105, 139, 105 }
SeaGreen1	{ 84, 255, 159 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
SeaGreen2	{ 78, 238, 148 }
SeaGreen3	{ 67, 205, 128 }
SeaGreen4, sea green	{ 46, 139, 87 }
medium sea green	{ 60, 179, 113 }
light sea green	{ 32, 178, 170 }
pale green	{ 152, 251, 152 }
PaleGreen1	{ 154, 255, 154 }
PaleGreen2	{ 144, 238, 144 }
PaleGreen3	{ 124, 205, 124 }
PaleGreen4	{ 84, 139, 84 }
SpringGreen1, spring green	{ 0, 255, 127 }
SpringGreen2	{ 0, 238, 118 }
SpringGreen3	{ 0, 205, 102 }
SpringGreen4	{ 0, 139, 69 }
medium spring green	{ 0, 250, 154 }
lawn green	{ 124, 252, 0 }
light green	{ 144, 238, 144 }
green1, green	{ 0, 255, 0 }
green2	{ 0, 238, 0 }
green3	{ 0, 205, 0 }
green4	{ 0, 139, 0 }
chartreuse1, chartreuse	{ 127, 255, 0 }
chartreuse2	{ 118, 238, 0 }
chartreuse3	{ 102, 205, 0 }
chartreuse4	{ 69, 139, 0 }
green yellow	{ 173, 255, 47 }
lime green	{ 50, 205, 50 }
yellow green	{ 154, 205, 50 }
forest green	{ 34, 139, 34 }
olive drab	{ 107, 142, 35 }
OliveDrab1	{ 192, 255, 62 }
OliveDrab2	{ 179, 238, 58 }
OliveDrab3	{ 154, 205, 50 }
OliveDrab4	{ 105, 139, 34 }
dark khaki	{ 189, 183, 107 }
khaki	{ 240, 230, 140 }
khaki1	{ 255, 246, 143 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
khaki2	{ 238, 230, 133 }
khaki3	{ 205, 198, 115 }
khaki4	{ 139, 134, 78 }
LightYellow1, light yellow	{ 255, 255, 224 }
LightYellow2	{ 238, 238, 209 }
LightYellow3	{ 205, 205, 180 }
LightYellow4	{ 139, 139, 122 }
yellow1, yellow	{ 255, 255, 0 }
yellow2	{ 238, 238, 0 }
yellow3	{ 205, 205, 0 }
yellow4	{ 139, 139, 0 }
gold1, gold	{ 255, 215, 0 }
gold2	{ 238, 201, 0 }
gold3	{ 205, 173, 0 }
gold4	{ 139, 117, 0 }
pale goldenrod	{ 238, 232, 170 }
light goldenrod yellow	{ 250, 250, 210 }
light goldenrod	{ 238, 221, 130 }
LightGoldenrod1	{ 255, 236, 139 }
LightGoldenrod2	{ 238, 220, 130 }
LightGoldenrod3	{ 205, 190, 112 }
LightGoldenrod4	{ 139, 129, 76 }
goldenrod	{ 218, 165, 32 }
goldenrod1	{ 255, 193, 37 }
goldenrod2	{ 238, 180, 34 }
goldenrod3	{ 205, 155, 29 }
goldenrod4	{ 139, 105, 20 }
dark goldenrod	{ 184, 134, 11 }
DarkGoldenrod1	{ 255, 185, 15 }
DarkGoldenrod2	{ 238, 173, 14 }
DarkGoldenrod3	{ 205, 149, 12 }
DarkGoldenrod4	{ 139, 101, 8 }
rosy brown	{ 188, 143, 143 }
RosyBrown1	{ 255, 193, 193 }
RosyBrown2	{ 238, 180, 180 }
RosyBrown3	{ 205, 155, 155 }
RosyBrown4	{ 139, 105, 105 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
indian red	{ 205, 92, 92 }
IndianRed1	{ 255, 106, 106 }
IndianRed2	{ 238, 99, 99 }
IndianRed3	{ 205, 85, 85 }
IndianRed4	{ 139, 58, 58 }
saddle brown	{ 139, 69, 19 }
sienna	{ 160, 82, 45 }
sienna1	{ 255, 130, 71 }
sienna2	{ 238, 121, 66 }
sienna3	{ 205, 104, 57 }
sienna4	{ 139, 71, 38 }
peru	{ 205, 133, 63 }
burlywood	{ 222, 184, 135 }
burlywood1	{ 255, 211, 155 }
burlywood2	{ 238, 197, 145 }
burlywood3	{ 205, 170, 125 }
burlywood4	{ 139, 115, 85 }
beige	{ 245, 245, 220 }
wheat	{ 245, 222, 179 }
wheat1	{ 255, 231, 186 }
wheat2	{ 238, 216, 174 }
wheat3	{ 205, 186, 150 }
wheat4	{ 139, 126, 102 }
sandy brown	{ 244, 164, 96 }
tan	{ 210, 180, 140 }
tan1	{ 255, 165, 79 }
tan2	{ 238, 154, 73 }
tan3	{ 205, 133, 63 }
tan4	{ 139, 90, 43 }
chocolate	{ 210, 105, 30 }
chocolate1	{ 255, 127, 36 }
chocolate2	{ 238, 118, 33 }
chocolate3	{ 205, 102, 29 }
chocolate4	{ 139, 69, 19 }
firebrick	{ 178, 34, 34 }
firebrick1	{ 255, 48, 48 }
firebrick2	{ 238, 44, 44 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
firebrick3	{ 205, 38, 38 }
firebrick4	{ 139, 26, 26 }
brown	{ 165, 42, 42 }
brown1	{ 255, 64, 64 }
brown2	{ 238, 59, 59 }
brown3	{ 205, 51, 51 }
brown4	{ 139, 35, 35 }
dark salmon	{ 233, 150, 122 }
salmon	{ 250, 128, 114 }
salmon1	{ 255, 140, 105 }
salmon2	{ 238, 130, 98 }
salmon3	{ 205, 112, 84 }
salmon4	{ 139, 76, 57 }
LightSalmon1, light salmon	{ 255, 160, 122 }
LightSalmon2	{ 238, 149, 114 }
LightSalmon3	{ 205, 129, 98 }
LightSalmon4	{ 139, 87, 66 }
orange1, orange	{ 255, 165, 0 }
orange2	{ 238, 154, 0 }
orange3	{ 205, 133, 0 }
orange4	{ 139, 90, 0 }
dark orange	{ 255, 140, 0 }
DarkOrange1	{ 255, 127, 0 }
DarkOrange2	{ 238, 118, 0 }
DarkOrange3	{ 205, 102, 0 }
DarkOrange4	{ 139, 69, 0 }
coral	{ 255, 127, 80 }
coral1	{ 255, 114, 86 }
coral2	{ 238, 106, 80 }
coral3	{ 205, 91, 69 }
coral4	{ 139, 62, 47 }
light coral	{ 240, 128, 128 }
tomato1, tomato	{ 255, 99, 71 }
tomato2	{ 238, 92, 66 }
tomato3	{ 205, 79, 57 }
tomato4	{ 139, 54, 38 }
OrangeRed1, orange red	{ 255, 69, 0 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
OrangeRed2	{ 238, 64, 0 }
OrangeRed3	{ 205, 55, 0 }
OrangeRed4	{ 139, 37, 0 }
red1, red	{ 255, 0, 0 }
red2	{ 238, 0, 0 }
red3	{ 205, 0, 0 }
red4, dark red	{ 139, 0, 0 }
hot pink	{ 255, 105, 180 }
HotPink1	{ 255, 110, 180 }
HotPink2	{ 238, 106, 167 }
HotPink3	{ 205, 96, 144 }
HotPink4	{ 139, 58, 98 }
DeepPink1, deep pink	{ 255, 20, 147 }
DeepPink2	{ 238, 18, 137 }
DeepPink3	{ 205, 16, 118 }
DeepPink4	{ 139, 10, 80 }
pink	{ 255, 192, 203 }
pink1	{ 255, 181, 197 }
pink2	{ 238, 169, 184 }
pink3	{ 205, 145, 158 }
pink4	{ 139, 99, 108 }
light pink	{ 255, 182, 193 }
LightPink1	{ 255, 174, 185 }
LightPink2	{ 238, 162, 173 }
LightPink3	{ 205, 140, 149 }
LightPink4	{ 139, 95, 101 }
maroon	{ 176, 48, 96 }
maroon1	{ 255, 52, 179 }
maroon2	{ 238, 48, 167 }
maroon3	{ 205, 41, 144 }
maroon4	{ 139, 28, 98 }
pale violet red	{ 219, 112, 147 }
PaleVioletRed1	{ 255, 130, 171 }
PaleVioletRed2	{ 238, 121, 159 }
PaleVioletRed3	{ 205, 104, 137 }
PaleVioletRed4	{ 139, 71, 93 }
medium violet red	{ 199, 21, 133 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
violet red	{ 208, 32, 144 }
VioletRed1	{ 255, 62, 150 }
VioletRed2	{ 238, 58, 140 }
VioletRed3	{ 205, 50, 120 }
VioletRed4	{ 139, 34, 82 }
magenta1, magenta	{ 255, 0, 255 }
magenta2	{ 238, 0, 238 }
magenta3	{ 205, 0, 205 }
magenta4, dark magenta	{ 139, 0, 139 }
violet	{ 238, 130, 238 }
plum	{ 221, 160, 221 }
plum1	{ 255, 187, 255 }
plum2	{ 238, 174, 238 }
plum3	{ 205, 150, 205 }
plum4	{ 139, 102, 139 }
orchid	{ 218, 112, 214 }
orchid1	{ 255, 131, 250 }
orchid2	{ 238, 122, 233 }
orchid3	{ 205, 105, 201 }
orchid4	{ 139, 71, 137 }
medium orchid	{ 186, 85, 211 }
MediumOrchid1	{ 224, 102, 255 }
MediumOrchid2	{ 209, 95, 238 }
MediumOrchid3	{ 180, 82, 205 }
MediumOrchid4	{ 122, 55, 139 }
dark orchid	{ 153, 50, 204 }
DarkOrchid1	{ 191, 62, 255 }
DarkOrchid2	{ 178, 58, 238 }
DarkOrchid3	{ 154, 50, 205 }
DarkOrchid4	{ 104, 34, 139 }
dark violet	{ 148, 0, 211 }
blue violet	{ 138, 43, 226 }
purple	{ 160, 32, 240 }
purple1	{ 155, 48, 255 }
purple2	{ 145, 44, 238 }
purple3	{ 125, 38, 205 }
purple4	{ 85, 26, 139 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
medium purple	{ 147, 112, 219 }
MediumPurple1	{ 171, 130, 255 }
MediumPurple2	{ 159, 121, 238 }
MediumPurple3	{ 137, 104, 205 }
MediumPurple4	{ 93, 71, 139 }
thistle	{ 216, 191, 216 }
thistle1	{ 255, 225, 255 }
thistle2	{ 238, 210, 238 }
thistle3	{ 205, 181, 205 }
thistle4	{ 139, 123, 139 }
gray0, black	{ 0, 0, 0 }
gray1	{ 3, 3, 3 }
gray2	{ 5, 5, 5 }
gray3	{ 8, 8, 8 }
gray4	{ 10, 10, 10 }
gray5	{ 13, 13, 13 }
gray6	{ 15, 15, 15 }
gray7	{ 18, 18, 18 }
gray8	{ 20, 20, 20 }
gray9	{ 23, 23, 23 }
gray10	{ 26, 26, 26 }
gray11	{ 28, 28, 28 }
gray12	{ 31, 31, 31 }
gray13	{ 33, 33, 33 }
gray14	{ 36, 36, 36 }
gray15	{ 38, 38, 38 }
gray16	{ 41, 41, 41 }
gray17	{ 43, 43, 43 }
gray18	{ 46, 46, 46 }
gray19	{ 48, 48, 48 }
gray20	{ 51, 51, 51 }
gray21	{ 54, 54, 54 }
gray22	{ 56, 56, 56 }
gray23	{ 59, 59, 59 }
gray24	{ 61, 61, 61 }
gray25	{ 64, 64, 64 }
gray26	{ 66, 66, 66 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray27	{ 69, 69, 69 }
gray28	{ 71, 71, 71 }
gray29	{ 74, 74, 74 }
gray30	{ 77, 77, 77 }
gray31	{ 79, 79, 79 }
gray32	{ 82, 82, 82 }
gray33	{ 84, 84, 84 }
gray34	{ 87, 87, 87 }
gray35	{ 89, 89, 89 }
gray36	{ 92, 92, 92 }
gray37	{ 94, 94, 94 }
gray38	{ 97, 97, 97 }
gray39	{ 99, 99, 99 }
gray40	{ 102, 102, 102 }
gray41, dim gray	{ 105, 105, 105 }
gray42	{ 107, 107, 107 }
gray43	{ 110, 110, 110 }
gray44	{ 112, 112, 112 }
gray45	{ 115, 115, 115 }
gray46	{ 117, 117, 117 }
gray47	{ 120, 120, 120 }
gray48	{ 122, 122, 122 }
gray49	{ 125, 125, 125 }
gray50	{ 127, 127, 127 }
gray51	{ 130, 130, 130 }
gray52	{ 133, 133, 133 }
gray53	{ 135, 135, 135 }
gray54	{ 138, 138, 138 }
gray55	{ 140, 140, 140 }
gray56	{ 143, 143, 143 }
gray57	{ 145, 145, 145 }
gray58	{ 148, 148, 148 }
gray59	{ 150, 150, 150 }
gray60	{ 153, 153, 153 }
gray61	{ 156, 156, 156 }
gray62	{ 158, 158, 158 }
gray63	{ 161, 161, 161 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray64	{ 163, 163, 163 }
gray65	{ 166, 166, 166 }
gray66	{ 168, 168, 168 }
dark gray	{ 169, 169, 169 }
gray67	{ 171, 171, 171 }
gray68	{ 173, 173, 173 }
gray69	{ 176, 176, 176 }
gray70	{ 179, 179, 179 }
gray71	{ 181, 181, 181 }
gray72	{ 184, 184, 184 }
gray73	{ 186, 186, 186 }
gray74	{ 189, 189, 189 }
gray	{ 190, 190, 190 }
gray75	{ 191, 191, 191 }
gray76	{ 194, 194, 194 }
gray77	{ 196, 196, 196 }
gray78	{ 199, 199, 199 }
gray79	{ 201, 201, 201 }
gray80	{ 204, 204, 204 }
gray81	{ 207, 207, 207 }
gray82	{ 209, 209, 209 }
light gray	{ 211, 211, 211 }
gray83	{ 212, 212, 212 }
gray84	{ 214, 214, 214 }
gray85	{ 217, 217, 217 }
gray86	{ 219, 219, 219 }
gray87	{ 222, 222, 222 }
gray88	{ 224, 224, 224 }
gray89	{ 227, 227, 227 }
gray90	{ 229, 229, 229 }
gray91	{ 232, 232, 232 }
gray92	{ 235, 235, 235 }
gray93	{ 237, 237, 237 }
gray94	{ 240, 240, 240 }
gray95	{ 242, 242, 242 }
gray96	{ 245, 245, 245 }
gray97	{ 247, 247, 247 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray98	{ 250, 250, 250 }
gray99	{ 252, 252, 252 }
gray100, white	{ 255, 255, 255 }

Importing DXF and DWG Format Files

You can import AutoCAD versions 2.5 (AC1002) through 2012 (AC1024) on Linux and versions 2.5 (AC1002) through 2017 (AC1027) on Windows. The entities are imported as 2D, not 3D. The types of entities imported are:

- 2D Polyline
- Polyline
- Line
- Arc
- Circle
- Ellipse
- Solid
- Block

To import a ***.dxf** or ***.dwg** model file (which may use an associated *.tech file):

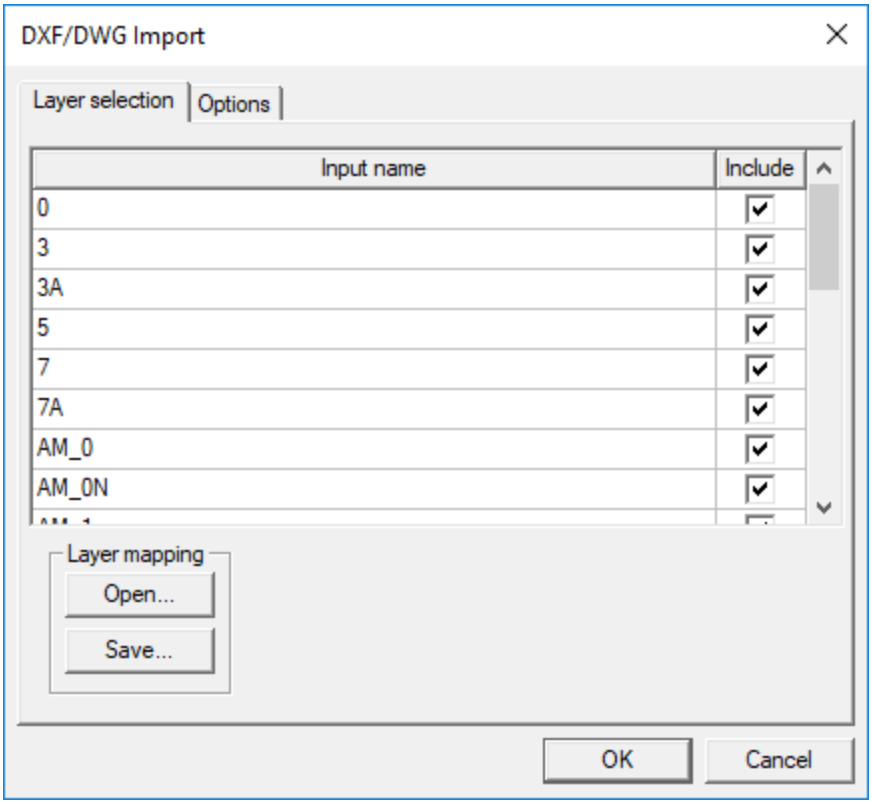
1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

2. Use the file browser to find and select an AutoCAD file for import.
3. Click **Open**.

   The **DXF/DWG Import** dialog box opens with the **Layer Selection** tab selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Input Name** field shows the name of the layer in the DXF/DWG file. This field is not editable.

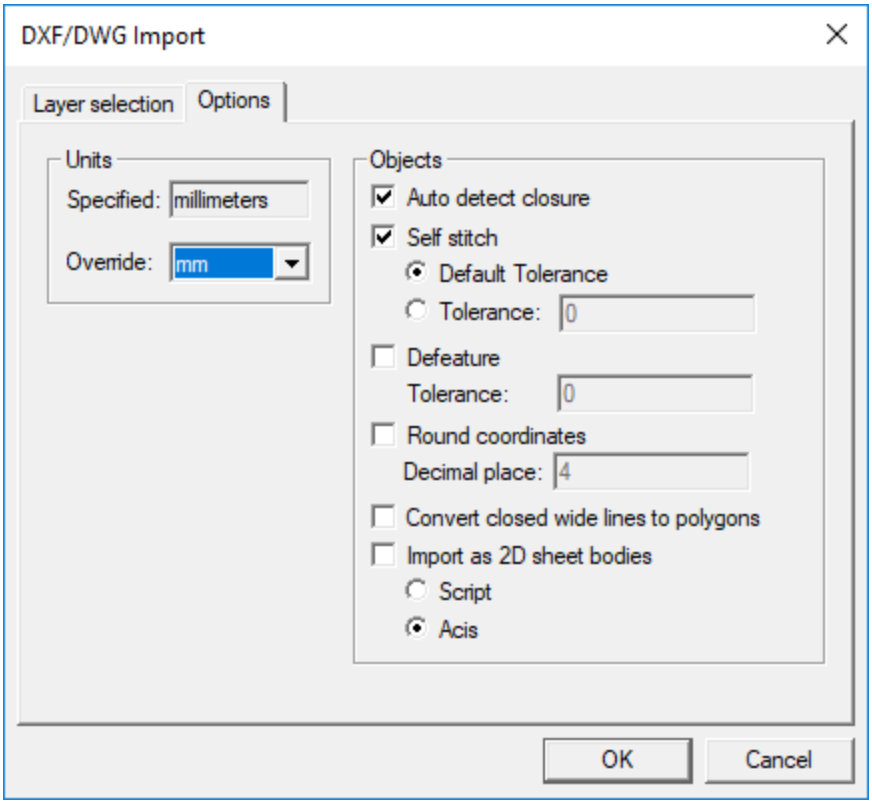
4. Use the **Include** check boxes to specify which layers to import from the selected file.
5. If there is an associated *.tech file, click **Open** to find and select it. A *.tech file is a plain text file that includes units, layer names, color, elevation, and thickness information. For example:

```
units um

//Layer_Name Color Elevation Thickness

BOTTOMLAYER purple 0 200

MIDLAYER green 500 200

TOPLAYER blue 1000 200
```

6. Click **Options**.

   The **Options** tab displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. In the **Units** area, the **Specified** field displays the layout units in the file for import. Use the **Override** drop-down menu to select a different unit of measure.
8. In the **Objects** area, use the check boxes to fine tune the import:
    - **Auto-detect closure** causes polylines to be checked to see whether or not they are closed. If a polyline is closed, the modeler creates a polygon in the design.
    - **Self-stitch** joins multiple straight line segments to form polylines. If the resulting polyline is closed, a polygon is created.
        - **Default Tolerance** uses default values to whether two coordinates should be considered for joining.
        - **Tolerance** allows you to specify a specific value for tolerance. This is useful if particular features in a model are outside of a normal tolerance allowance.
    - **De-feature Tolerance** removes certain small features in the imported geometry to reduce complexity. The features that are removed include: multiple points placed within the specified distance; thin or narrow regions ("thins" and "spikes"); and extraneous points along straight line segments.
    - **Round Coordinates Decimal Place** rounds all imported data to the specified number of decimal points.
    - **Convert closed wide lines to polygons** imports wide polylines as polygons. This affords more flexibility in changing the shape of such an object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import as 2D sheet bodies** causes imported objects to be organized in terms of 2D sheets.

9. Select either **Script** or **Acis** for the import method.
10. Click **OK**.

    The file is imported into the active **Layout** window.

Importing 3D Model Files

3D model files can be imported directly into the active **Modeler** window. Supported 3D model file formats are listed in the table that follows.

Type	Extension(s)	Supported Version(s)
ACIS SAB	*.sab	Up to 2024
ACIS SAT	*.sat	Up to 2024
Ansoft Geometry	*.AnstGeom	All
Ansys 3D Modeler	*.sm3	Up to 2022R2 See UDMs and CAD integration with Workbench.
AutoCAD	*.dxf, *.dwg	**Linux**: Versions 2.5 (AC1002) through 2012 (AC1024) **Windows**: Versions 2.5 (AC1002) through 2017 (AC1027) See Importing DXF and DWG Format Files.
Autodesk Inventor	*.ipt, *.iam	Up to 2025
CATIA	*.exp, *.model, *.CATPart, *.CATproduct	CATIA V4 – Up to 4.2.5 CATIA V5 – Up to V5_6R2024
Creo Parametric	*.prt, *.asm	Pro/Engineer 19.0 to Creo 11.0
IGES	*.iges, *.igs	5.1, 5.2, 5.3
JT	*.jt	Up to v10.9
NASTRAN	*.nas	CROD, CBEAM, CTRIA3, and CQUAD4 element types
Siemens NX	*.prt	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2406
Parasolid	*.x_t, *.x_b	9.0 through 36.1.185
SOLIDWORKS	*.SLDPRT, *.SLDASM	From 97 up to 2024

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Extension(s)	Supported Version(s)
STEP	*.step, *.stp	AP 203 E1/E2, AP 214, AP 242 (geometry only)
STL	*.stl	2.0 (see Lightweight Geometry in SBR+)

Note:

- Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
- If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 3D model file:

1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

2. Select the file type you want from the **Files of type** drop-down menu.
3. Set any import options available for the selected file type. Options may include:
   - **Validation and Healing Options** – see Healing an Imported Object.
   - **Import Material Names** – use the check box to import material names.
   - **Import Free Surfaces** – for Creo Parametric files, use the check box to import surfaces as well as parts.
   - **Stitch Tolerance and Units** – for STEP and IGES files, specify stitch tolerance and units. The default value (auto) comes from Healing options.
   - **STL Import Options** – for STL files for Modal or Terminal solutions, select the modeling units to which the imported model is scaled and whether to merge faces that are on the same plane, and whether to Heal on import. Heal attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.

   The generic format specification for STL does not include units. When “Auto” is chosen for the file units, the current Model units are normally used.

   Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Create Groups for Sub Assembly** – for formats that allow importing MCAD Assemblies as groups, use the check box to retain the assembly structure of objects using groups. For every subassembly in the model, a group is created and it retains hierarchical information by creating group hierarchy. See: Group Commands for Modeler Objects.

4. Use the file browser to find and select a file for import.
5. Click **Open**.

   The file is imported into the active **Modeler** window

> **Note:**
>
> While objects created in Mechanical can always be classed in the history tree as either a solid, sheet, or wire, some imported objects may have mixture of these. Mechanical places such objects in an **Unclassified** folder in the history tree.

Exporting Files

You can export the following types of files from Ansys Electronics Desktop projects:

- 2D model files
- 3D model files
- Graphics files
- Reports as data or graphics files

Exporting 2D Model Files

Mechanical exports 2D models in AutoCAD *.dxf format. In this format, the geometry located within the XY plane is exported.

> **Note:**
>
> If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin.

To export a file to ***.dxf** format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select **AutoCAD DXF Files (*.dxf)**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

Exporting 3D Model Files

You can export 3D models to 3D model file formats.

Supported 3D model file formats include the following:

Type	Extension(s)
ACIS SAB (exported with version 5.0)	*.sab
ACIS SAT (exported with version 5.0)	*.sat
AutoCAD	*.dxf
GDSII	*.gds
IGES	*.iges, *.igs
OBJ Files (for importing models to Ensight.)	*.obj
Parasolid	*.x_t, *.x_b
STEP	*.step, *.stp
STL	*.stl
GLTF (only available on Windows)	.gltf or .glb

To export a file to a 3D model format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

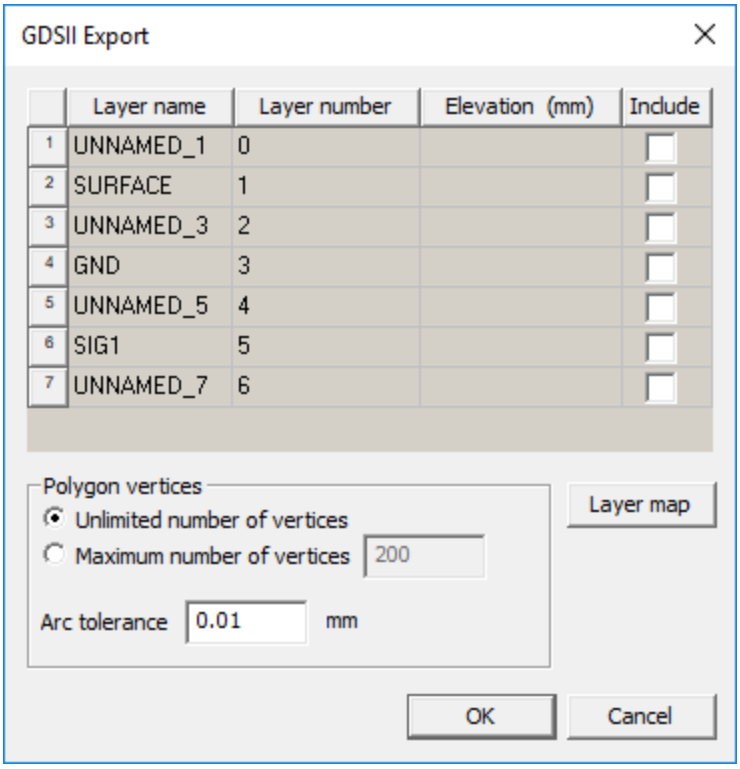
2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select the desired 3D file type.
5. Click **Save**.

   The file is exported to the specified location with the appropriate file format. Some file types launch an additional dialog box for specifying settings prior to export. These are described below.

GDSII Files

There are additional options for *.gds files:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

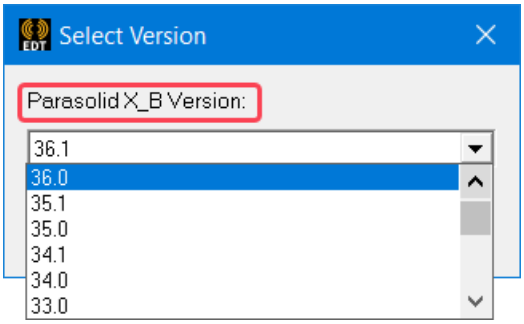


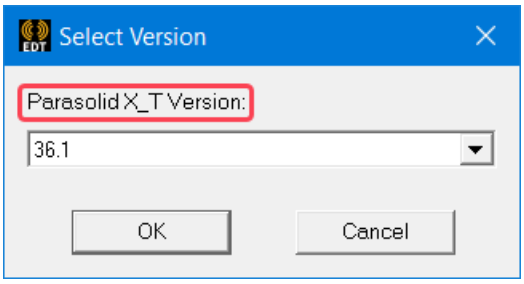
- Use the **Include** check boxes to select layers for export.
- Click **Layer map** to define a *.layermap file, if desired. This is a text file that maps GDSII layer numbers to layer names in the stackup. The *.layermap file can have the same format as the .tech file used in GDSII import, but it only needs the layer name and number. Any other information is ignored.
- In the **Polygon Vertices** area, select either **No Limit to the number of vertices** or **Limit the number of vertices** and specify a value.
- For **Arc Tolerance**, leave the default or specify a value.

Parasolid Files

There is one additional option for Parasolid files:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Use the drop-down menu to select a Parasolid version.

Exporting Graphics Files

You can export to the following graphics formats:

Extension	Contents
.bmp	Bitmap files
.gif	Graphics Interchange Format files
.jpg	Joint Photographics Experts Group files
.png	Portable Network Graphics format files
.tiff	Tagged Image File Format files.
.wrl	Virtual Reality Modeling Language (VRML) files.
.gltf or **.glb**	Graphics Language Transmission Format (GLTF); only available on Windows

To export a Modeler image to a graphics format:

1. Click **Modeler** > **Export**.

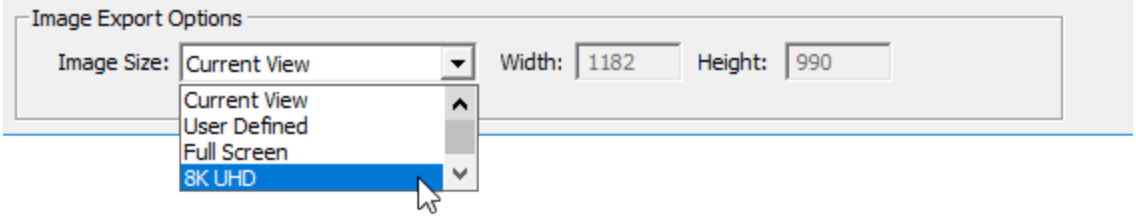
   The **Export File** window appears.

2. Browse to and select a location to save your file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select the desired image format.

   When you select an image format, **Image Export Options** appear. Note that this window does not appear for .gltf, .glb, or .wrl files, which are not image formats; if you are saving to one of these formats, skip to Step 6.



5. Choose your **Image Size**.

   Options include: Current View, User Defined (which allows you to specify **Width** and **Height**), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD.

6. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

You can also export an image file to a specified resolution using scripting commands. Fonts and line thickness are not scaled, only the image. You will have to iteratively increase font sizes until you find a suitable output.

Example Script:

```
oEditor.ExportModelImageToFile("C:/Users/Documents/highresexample_
image.jpg", 7680, 4320,

[

"NAME:SaveImageParams",

"ShowAxis:="        , "True",

"ShowGrid:="        , "True",

"ShowRuler:="       , "True",

"ShowRegion:="      , "Default",

"Selections:="      , ""

])
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting Options Files

Ansys Electronics Desktop can export user and host options files in XML format.

Exporting Options Files Using the Desktop UI

Export options files by selecting **Tools** > **Options** > **Export Options Files**. This brings up a browser dialog box to select the destination directory.

Click **Open** to copy all config files for the current user and current host to the specified directory. Config files for the install, install_machine, user, and user_machine levels will be copied, if they exist. One additional file, admin.XML, will also be copied to the destination directory; this file does not contain user configurable options.

Exporting Options Files Using a Script

A script command has been added that exports options config files:

ExportOptionsFiles

Syntax	`ExportOptionsFiles <DestinationDirectory>`
Return Value	None.
Parameters	BSTR <DestinationDirectory>

VBScript Example	`Dim oAnsoftApp`
	`Dim oDesktop`
	`Set oAnsoftApp = CreateObject ("Ansoft.ElectronicsDesktop")`
	`Set oDesktop = oAnsoftApp.GetAppDesktop()`
	`oDesktop.ExportOptionsFiles "D:/test/export/"`

IPY Example	`oDesktop.ExportOptionsFiles ('D:/test/export')`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

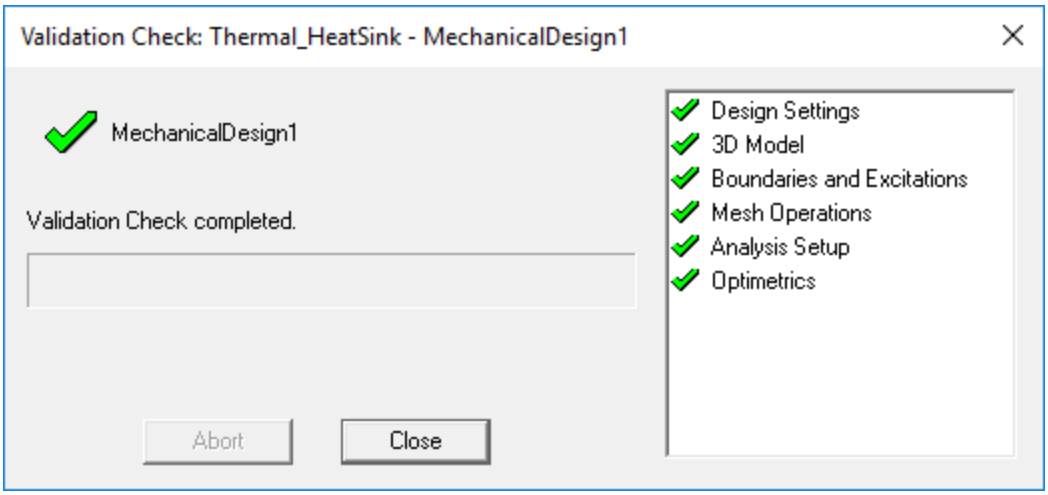
Validating Projects

Before running an analysis on a model, it is important to first perform a validation check on the project. During a validation check, the solver examines the setup details of the active project to verify that all the necessary steps have been completed and that their parameters are reasonable.

To perform a validation check on the active project:

1. Click **Mechanical** > **Validation Check**. You can also click the **Validate** icon on the **Simulation** ribbon tab.

   Electronics Desktop checks the project setup, and the **Validation Check** window appears.

2. View the results of the validation check in the **Validation Check** window.

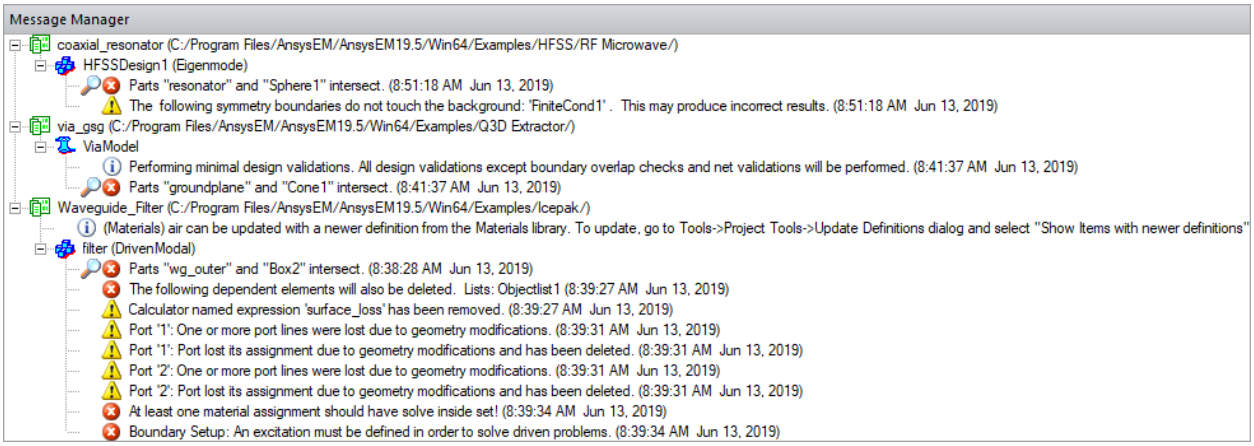
   The following icons can appear next to an item:

	Indicates the step is complete.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Indicates the step is incomplete.
	Indicates the step may require your attention.

3. View any messages in the **Message Manager** window.



4. If the validation check indicates that a step in your project is incomplete or incorrect, carefully review the setup details for that particular step and revise them as necessary.
5. After you have revised any setup details, click **Mechanical** > **Validation Check** to run another validation check. Repeat until the Validation Check completes with no issues.

> **Note:**
>
> You can change the strictness with which 3D Model faults are validated.

6. Click **Close**.

Modeler Validation Settings

You can adjust the degree to which the software checks a model for faults that could jeopardize mesh accuracy.

There are three levels of model validation for a given design: **Warning Only**, **Basic**, and **Strict**.

> **Note:**
>
> This setting only affects the "3D Model" stage of a design validation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Warning Only** – allows all models to pass 3D Model validation regardless of any faults that are found (acis_entity check errors). These faults are posted in the message window as warnings.
- **Basic** – allows most models to pass 3D Model validation. This excuses non-manifold errors and most acis_entity_check errors. Some faults are flagged as model errors (basic entity check errors), thereby prohibiting a design from proceeding to the meshing stage of an analysis. You must either correct such errors before attempting to analyze the design under the **Basic** setting, or change the **Model Validation** level to **Warning Only**.
- **Strict** – enforces a tighter tolerance for model faults than the **Warning Only** and **Basic** settings. All model faults found during 3D Model validation are posted to the message window. These errors must be corrected before attempting to analyze the design under the **Strict** setting, otherwise you must change the **Model Validation** level to **Basic** or **Warning Only**.

To set the Model Validation level:

1. Select > **Design Settings**.

   The **Design Settings** dialog box appears. The **Validations** tab lets you set the validation as **Basic**, **Strict**, or **Warning Only** and includes solver-specific selections.

2. Choose the desired level of validation from the **Entity Check Level** drop-down menu.

   You can also click the **Save as Default** button to make the current selection the default, or select **Restore Default**.

3. Click **OK** to accept the selection and close the dialog box.

> **Note:** For more information on Maxwell-specific settings, see Maxwell Validation Tab.

Copying and Pasting a Project or Design

To **copy** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Copy**.
2. Click **Edit** > **Copy**.

   The project or design is copied for pasting.

To **paste** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Paste**.
2. Click **Edit** > **Paste**.

   The project or design is pasted under the selected project, and an icon is added to the project tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can also use keyboard shortcuts.

Renaming Projects

In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

To rename an existing, active project:

1. Select the project in the Project Tree.
2. Right-click and select **Rename**.

   This activates the text field for the project name.

3. Type the new project name and press **Enter**.

   The new project name appears in the directory and the project remains in the original location.

Archiving Projects

Use the **File** > **Archive** command to place a project and any other files related to the project that you want to include in a *.<product>z file or *.zip format archive. You can make notes about the contents of the archive and specify whether to include results and solutions files. The **Archive** command attempts to automatically detect the necessary files for linked projects and automatically include them in the archive. You can also add additional files to the archive, including results files, external files and projects. For example, if a project linked to the main project also has linked or associated files, you can add them.

Archive File Types

Internally, project archive files are .zip files, and are compatible with any program that can read .zip files. Project archive files have an extension that is unique for each product, and is generated by adding a 'z' to the project file extension (for example, *.aedtz, *.adsnz). This extension displays as the default when saving and restoring archive files.

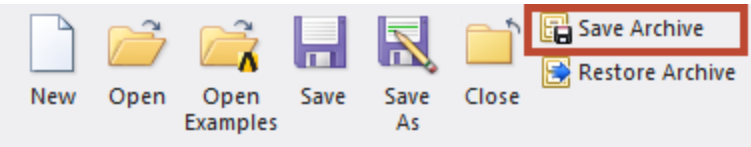
Archive Preview

The **Archive** command includes a preview feature that lets you review the contents of a planned archive.

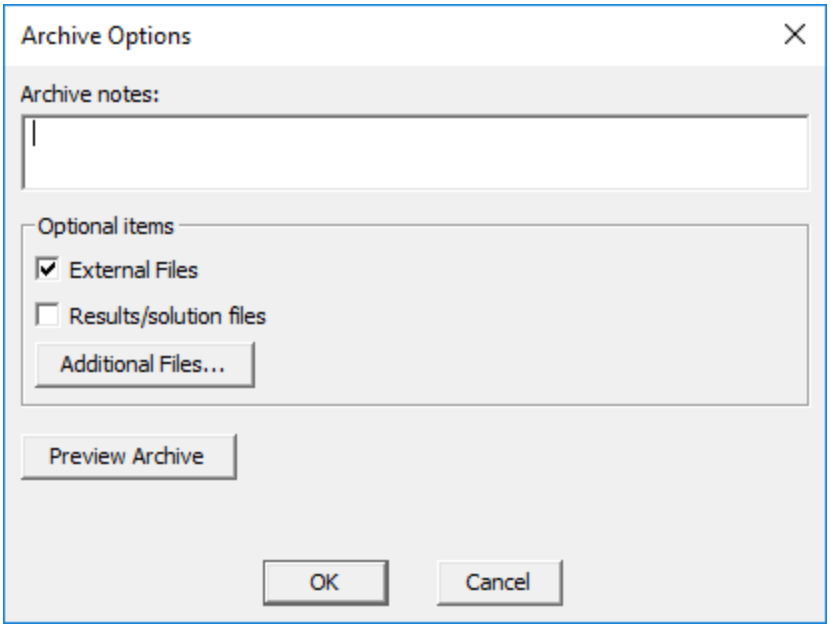
To archive the current project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **File** > **Archive** or select the **Desktop** tab of the ribbon and click **Save Archive**.

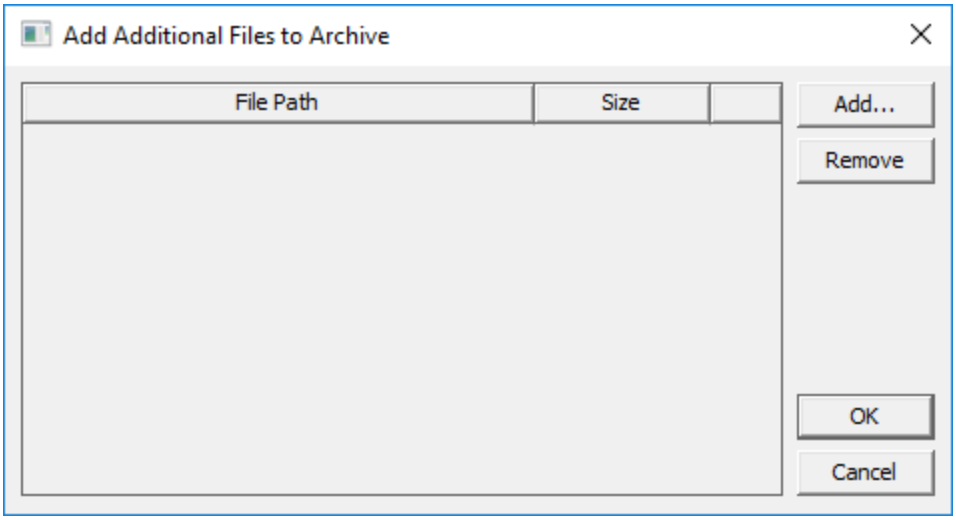


The **Archive Options** dialog box opens.



- **Archive Notes** – in this field, you can specify notes that will be visible when previewing the archive. These notes can be viewed from the preview dialog box without restoring the archive.
- **External Files** – selecting this check box causes all external files to be included in the archive. These include any existing files associated with the project, such as linked files, or files added through the **Project** > **Insert Documentation File** command or **Project** > **Data Set** command.
- **Results/Solution Files** – selecting this check box causes the entire results directory to be included in the archive. This may greatly increase the size of the archive file.
- **Additional Files** – clicking this opens the **Add Additional Files to Archive** dialog box.

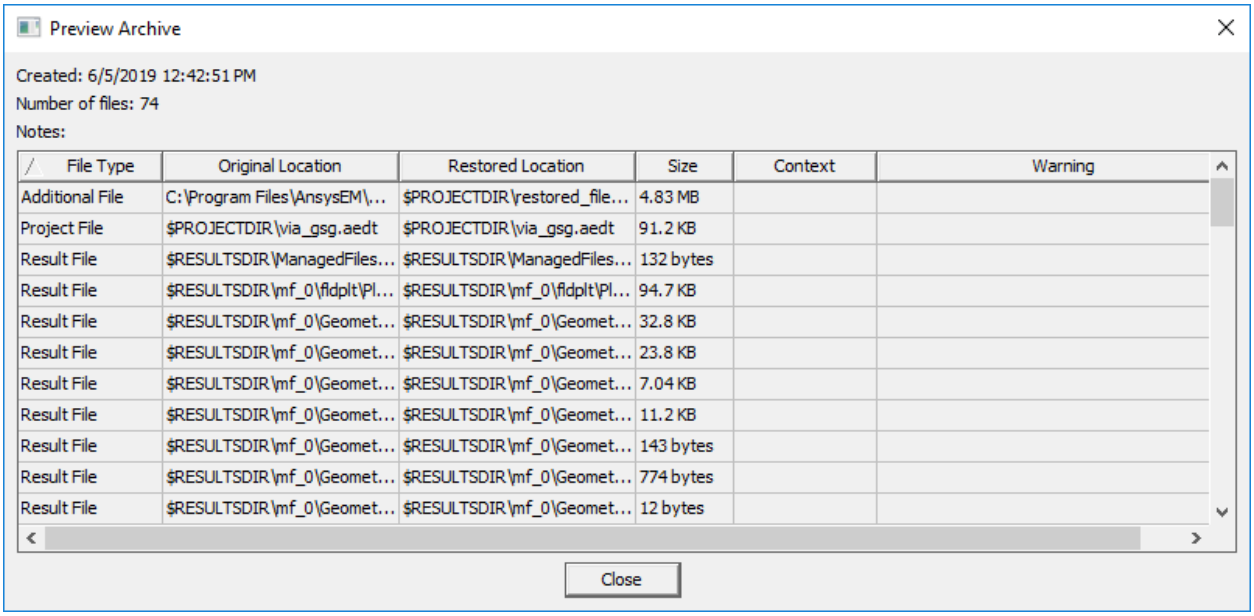
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can click **Add** to browse and locate additional files you want to include in the archive. You can select and the **Remove** any files listed.

You can click **OK** to accept changes or **Cancel** any proposed changes.

2. Select any optional items, and make any desired notes in the text field.
3. When you have made your selections, click **Preview Archive** to view archive contents, file types, the locations of the archive files, and the locations where restoring from archive would place them. Additionally, there may be context or warning information.



To read longer locations, you can drag the column header to expand them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previewing an archive before creating the archive can be helpful in order to see exactly what files will be included in an archive, as well as how those files are being relocated. Another purpose of previewing an archive is to view warnings and consider if any additional files need to be added to the archive.

4. When you are ready to create the archive, **Close** the preview and click **OK**.

   A browsing window appears.

5. Specify a name for your file and select the format you want to use from the **Save as type** drop-down menu.
6. Click **OK** to create the archive.

File Relocation

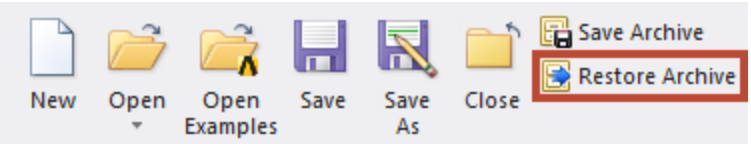
In a project to be archived, external files can be located anywhere on the user's system. One of the goals is for the restored project to be relatively self contained, and to NOT allow the restoring of an archived project to haphazardly write files anywhere on the restoring user's system.

To achieve this, it is sometimes necessary to change the location of files in the archived project so that the external files are located in the project directory. At archive time, any external files not located in the project directory are relocated to the restored_files subdirectory of the project directory in the archived project. Any external files located in the user library or system library will be relocated to the personal library directory. Note that the project file that is written into the archive will be updated to refer to the files at the new locations, and the original project file will remain unaltered.

Restoring an Archived Project

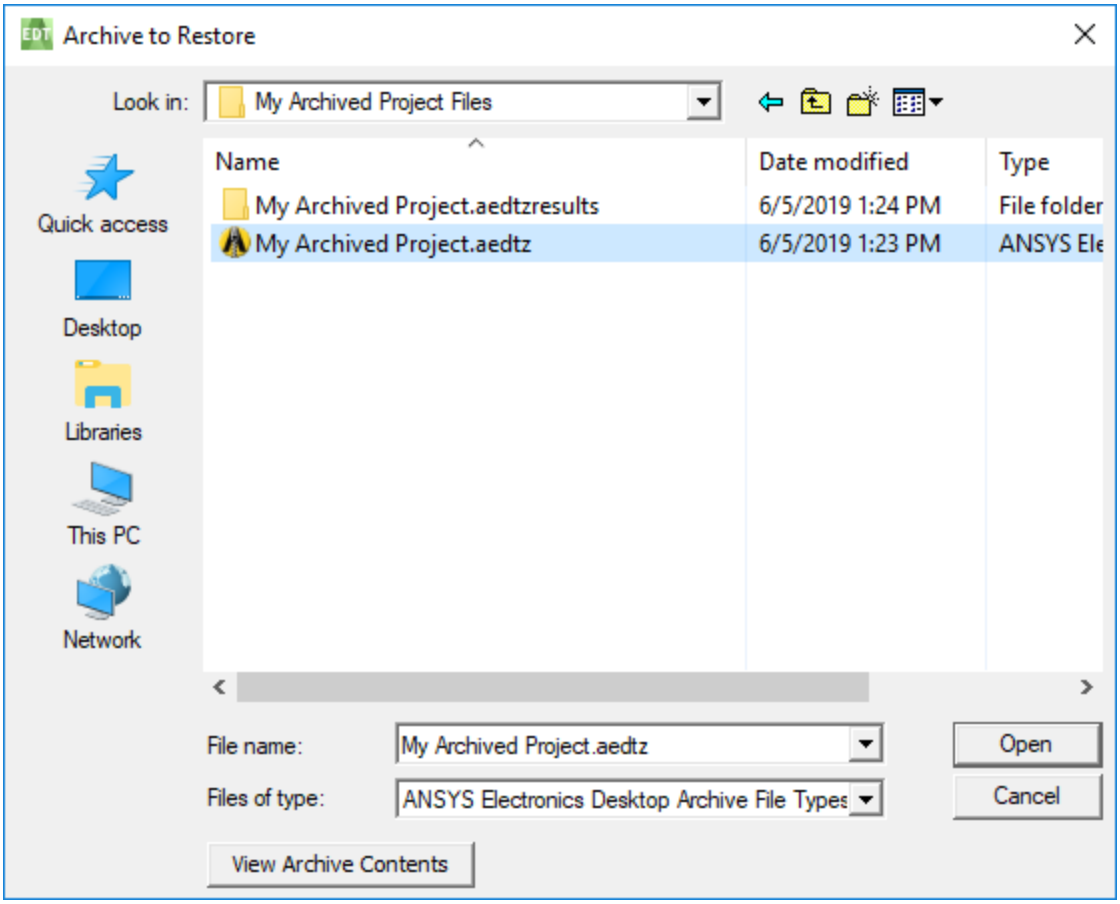
To restore an existing archive created with **File** > **Archive...**, use the **File** > **Restore Archive** command.

1. Click **File**> **Restore Archive** or select the **Desktop** tab of the ribbon and click **Restore Archive**.



   This displays an **Archive to Restore** browser window that lets you navigate your file system for *.aedtz, legacy *.*<solversuffix>z* or *.zip archive files.

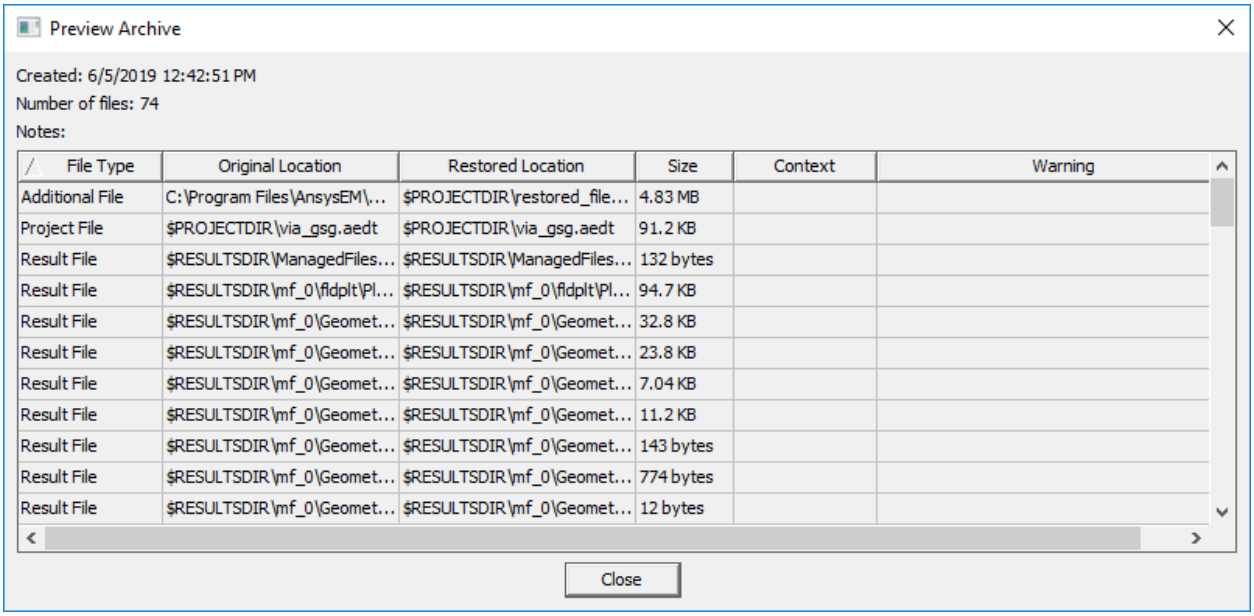
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you select a valid archive file, you can click **View Archive Contents** to preview the contents.

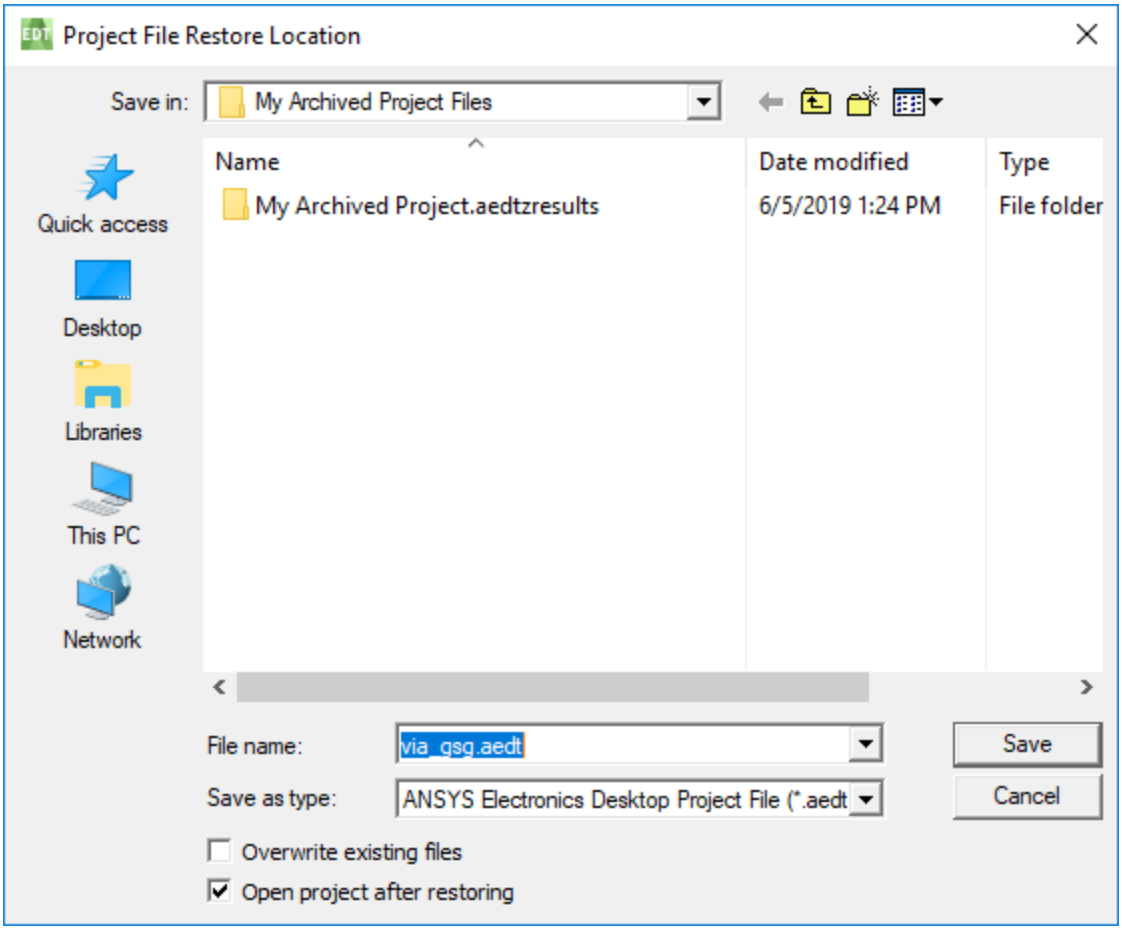
   The **Preview Archive** dialog box lists all files in the archive, notes, the original and restored locations, and any warnings that were generated at archive time. These warnings may be useful to identify additional steps that are needed to update any files to refer to files which had to be manually added to the archive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **Close** to exit the preview.
4. In the browsing window, click **Open** to change to a **Project File Restore Location** browser.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



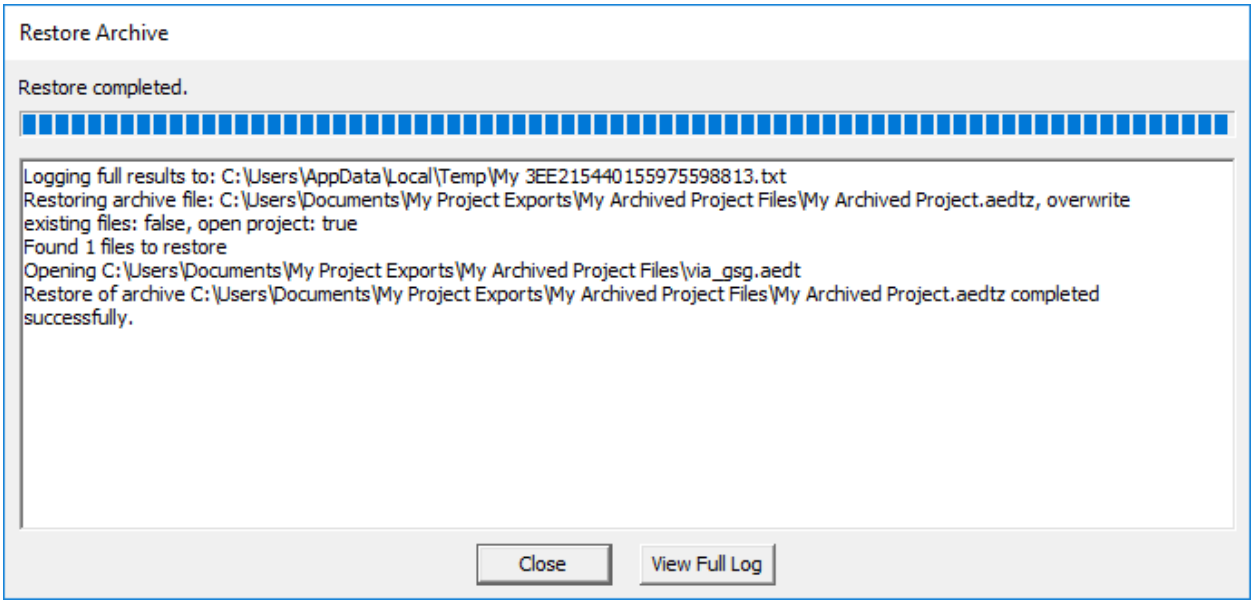
You can edit the file name, and select options:

- **Overwrite existing files** – If this check box is selected, restored files will automatically overwrite existing files during the restore process. If this button is deselected, existing files will not be modified.
- **Open project after restoring** – If this check box is selected, the project will be opened in this instance of the application after all files have been restored.

5. Click **Save** to restore the archived file.

   While restoring an archive, a dialog box displays showing restore results. A progress bar shows the relative progress, and the text window displays important information and warnings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Electronics Desktop also generates a full log file that contains detailed information about the restore process. The first line in the text window displays the location of the full log file. After the restore has been completed, you can click **View Full Log** to open this log file, or use a text editor and open the file at the specified location.

Deleting a Project or Design

To delete a project or design:

1. In the Project Tree, select a project or design.
2. Click **Edit** > **Delete**, or press the **Delete** key.
3. Confirm the warning box to complete the delete operation.

   The project or design is removed from the Project tree.

Setting Read Only Designs

Designs can be set as either Read Only or Full Access. Full Access is the default status of a design and allows you to modify the design. In Read Only mode, you may only run the design or link it to another design. Setting a design as Read Only protects it from accidental modification.

Read Only designs are marked with a red lock in the Project Manager:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change a single design's designation:

1. In the **Project Manager**, right-click the design.
2. Select either **Convert to Read Only** or **Convert to Full Access**.

To change the designation of *every design* in a project:

1. In the **Project Manager**, right-click the project.
2. Select either **Convert All Designs to Read Only** or **Convert All Designs to Full Access**.

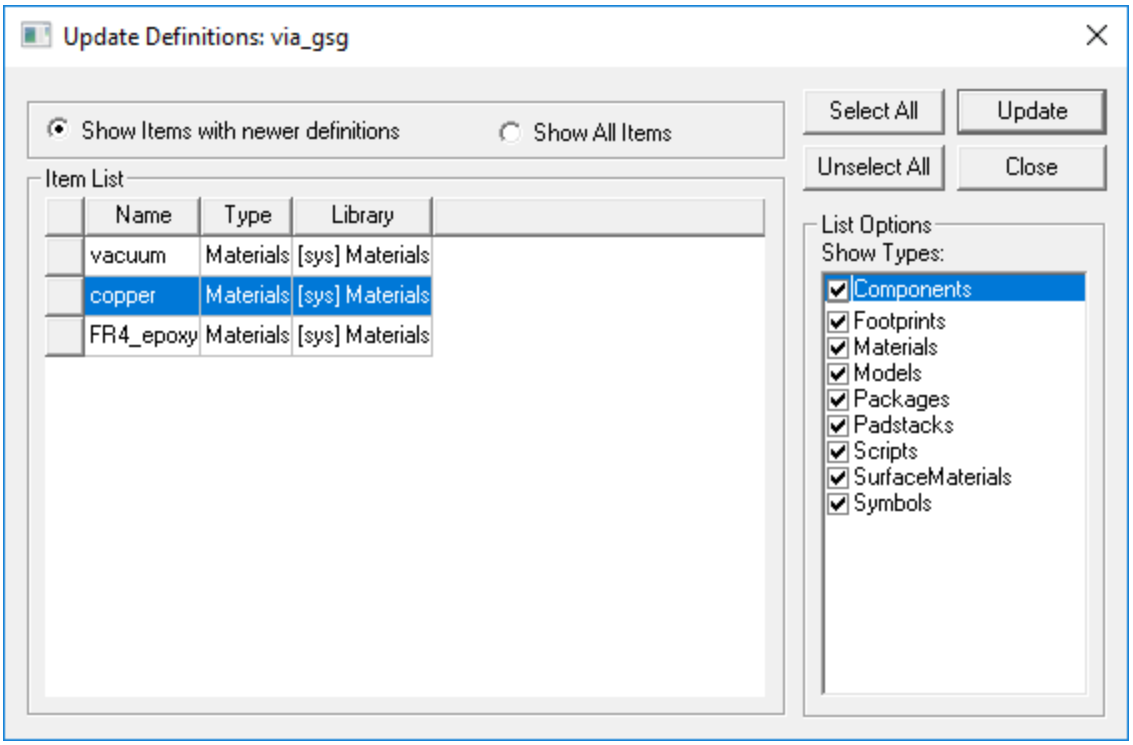
> **Note:**
>
> Read Only settings are not saved when you save the project file.

Updating Design Components

To update components defined in the current design:

1. Click **Tools** > **Project Tools** > **Update Definitions**.

   The **Update Definitions** dialog box appears.



2. Select either **Show Items with newer definitions** or **Show All Items**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Show Types** list in the **List Options** section, select the types of definitions you want to show in the **Item List**.
4. Select the item(s) you want to update from the **Item List**, or use **Select All.**
5. Click **Update**.

   A message appears telling you the update was successful.

6. Click **OK** to close the message.

When you are finished updating definitions, click **Close**.

Undoing and Redoing Commands

The **Edit** menu contains **Undo and** Redo commands.

Use the **Undo** command to cancel the last action you performed on the active project or design.

Use the **Redo** command to reperform an action you previously undid.

Both commands are useful for dealing with unintended actions related to project management, model creation, and post-processing.

To use **Undo** or **Redo**:

1. In the **Project Manager** window, do one of the following:
   - To undo or redo your last project-level action, such as inserting a design or adding project variables, click the project icon.
   - To undo or redo your last design-level action, such as drawing an object or deleting a field overlay plot, click the design icon.

   > **Note:**
   >
   > You cannot undo an analysis that you've performed on a model—that is, the **Mechanical** > **Analyze** command.

2. Click **Edit** > [**Undo/Redo**], or click the **Undo** or **Redo** button ( ) on the **Desktop** tab.

   Your action is undone or redone.

> **Important:**
>
> When you save a project, Electronics Desktop clears the entire undo/redo history for the project and its designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inserting a Documentation File

You may want to add a documentation file to the project tree.

1. Click **Project** > **Insert Documentation File**.

   This opens a file browser that lets you navigate your file system.

2. Select the documentation file and click **OK**.

   The documentation file is placed in the Project Tree.

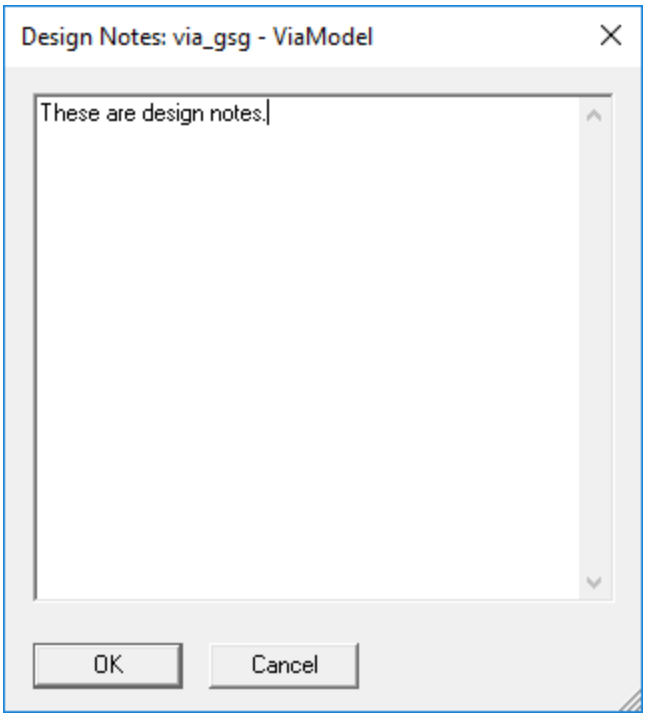
Working with Design Notes

You can save design notes, such as information about the creation date or a description of the device being modeled. This is useful for keeping a running log.

To add notes:

1. Click **Mechanical** > **Edit Notes**.

   The **Design Notes** window appears.



2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To edit notes, use the same window or double-click the **Notes** icon (Notes) in the Project Tree.

To delete the existing notes for a design:

1. Select the **Notes** icon in the Project Tree.
2. Click **Edit** > **Delete**, or right-click and select **Delete**.

   The Notes icon is removed from the Project Tree.

> **Note:**
>
> Notes are used to document aspects of designs only.
>
> For project-level documentation, you can insert a documentation file into a project with the **Project** > **Insert Documentation File** command.

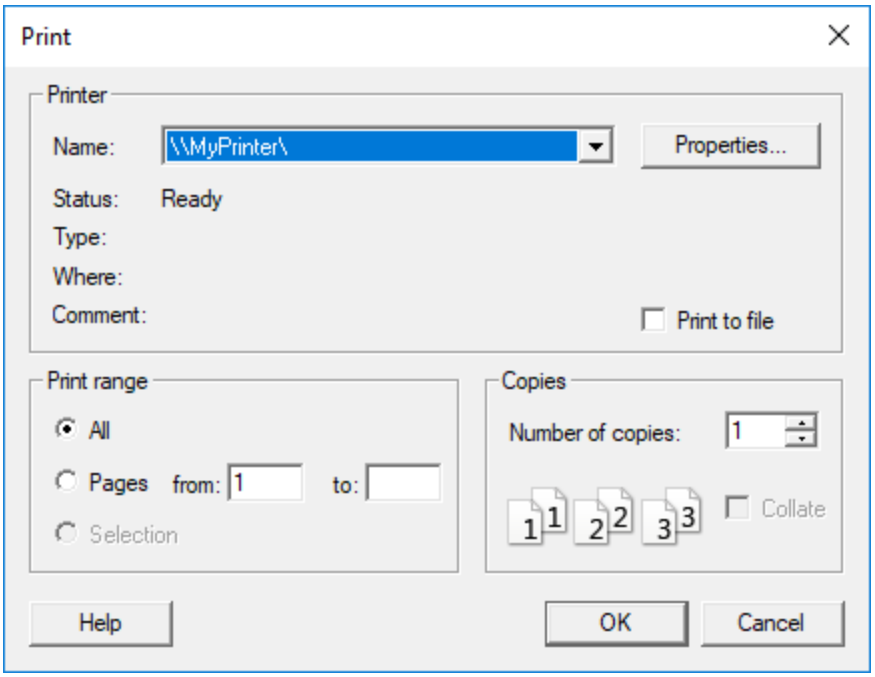
Printing

Electronics Desktop printing commands enable you to send an image of the active window to the printer.

To print the project:

1. Click **File** > **Print**.

   A dialog box similar to the following one appears:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Print options will vary based on your printer. Specify your desired options.
3. Click **OK** to print.

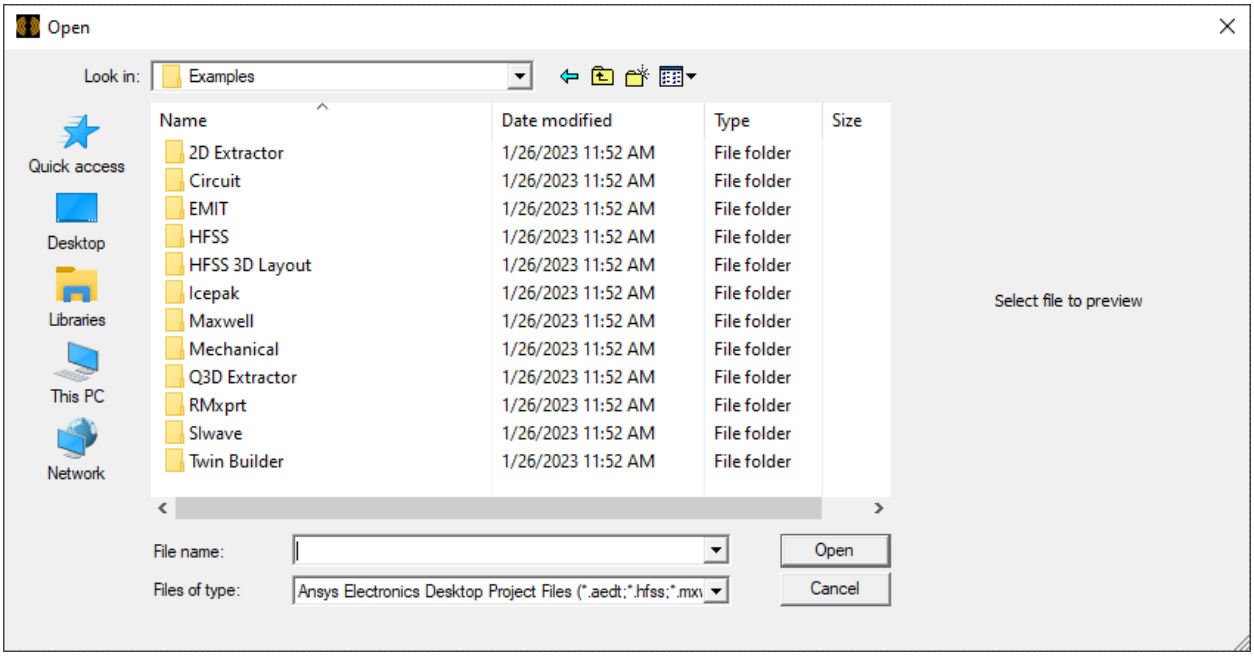
To change additional print settings, click **File** > **Page Setup**.

To preview what your printout will look like, click **File** > **Print Preview**.

> **Important:**
>
> To print from Ansys software on Linux, you must first configure a printer from the MainWin control panel.
>
> For more information, consult documentation for your specific instance of Linux.

Example Projects

Your Ansys Electronics Desktop installation includes an example directory containing projects folders for several kinds of designs.



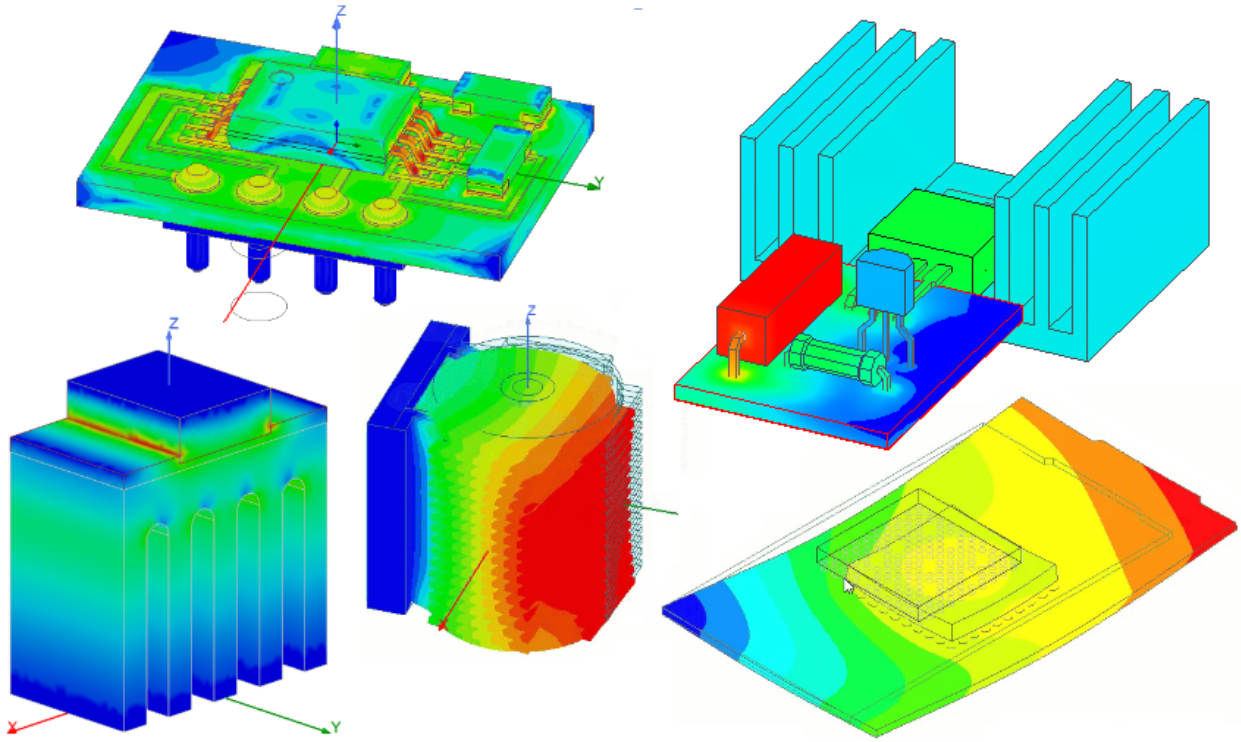
Example projects are organized by the design type.

Several of these projects are associated with detailed getting started guides.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanical Example Projects

The Mechanical examples folder contains four projects—one modal, one thermal, and two structural. These projects are fully set up with boundaries, excitations, solution setups, results plots, and mesh and field overlays. However, the solutions have not been run, and you cannot review the results until you save the projects to a working folder and solve the analyses. These projects are not associated with any Mechanical Getting Started Guides but are in addition to them.



Click one of the following links for more information:

- **Modal Example**:
  - Motor Housing
- **Thermal Examples:**
  - **Steady-State:**
    - Transistor with Heat Sink
  - **Transient:**
    - Current Regulator
- **Structural Examples:**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- PCB Stress
- Substrate Warpage

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in **Mechanical**. You can assign a variable to any dimension, material property, or output value. Variables are useful in the following situations:

- You expect to change a parameter often.
- You expect to use the same parameter value often.
- You intend to run a parametric analysis, in which you specify a series of variable values within a range to solve.
- You intend to optimize a parameter value by running an optimization analysis or using Design of Experiments to generate a response surface.
- You intend to animate a plot against a variable.

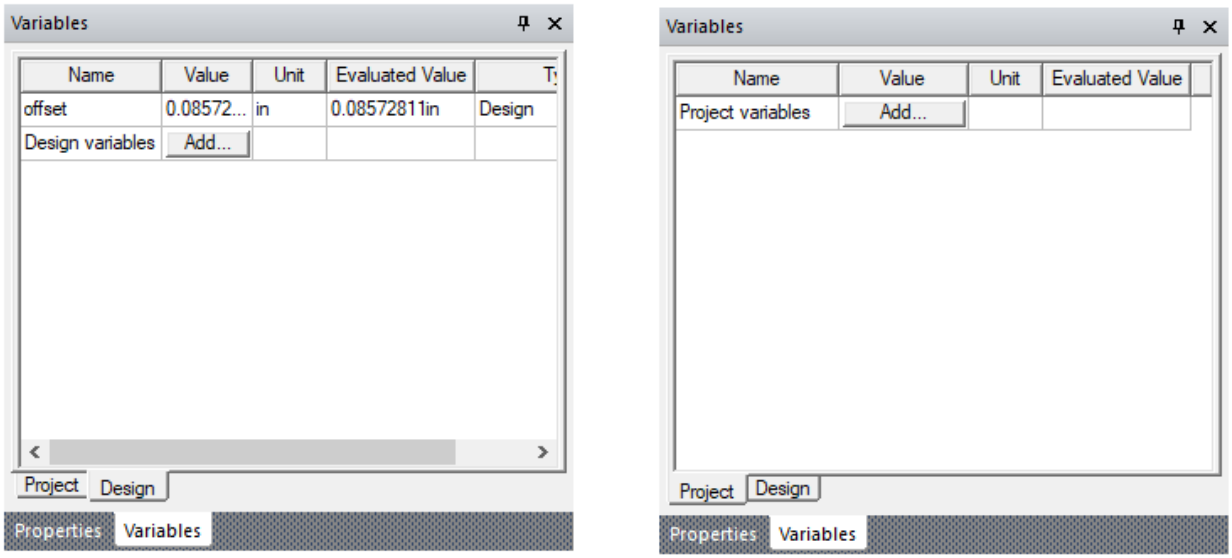
Variables can be regular variables, array index variables, or arrays. A project variable's value can reference other project variables or intrinsics. Values can be a number, variable, or mathematical expression.

There are two types of variables in Mechanical:

Project Variables	A project variable can be assigned to any parameter value in the Mechanical project in which it was created. Mechanical differentiates project variables from other types of variables by prefixing the variable name with dollar sign symbol ($). You can manually include the $ in the project variable's name, or Mechanical will automatically append the project variable's name after you define the variable. Project variables can be designated as Design, ArrayIndex or Separator variables .
Design (Local) Variables	A design variable can be assigned to any parameter value in the Mechanical design in which it was created. From the Design Variables **Properties** dialog box, you can Add, Add Array, Edit, or Remove design variables. Design Variables can be designated as Design, ArrayIndex or Separator variables

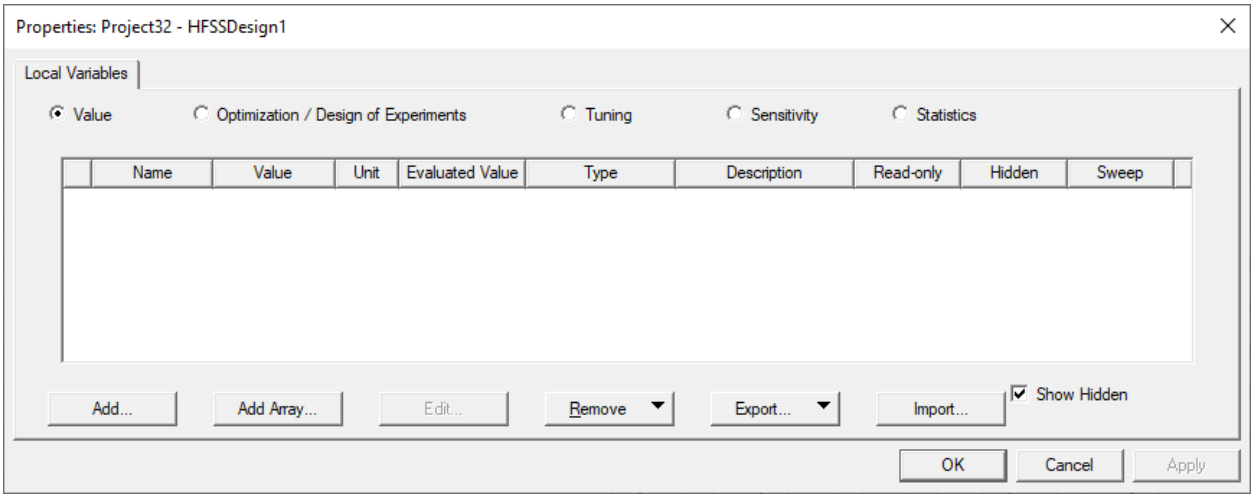
Clicking **View** > **Variables** brings up a dockable variable window that is associated with the active project and/or design. When there is an active project, there will be a corresponding project variable tab. When there is an active design, there will be a corresponding design variable tab. Each tab contains an **Add...** button allowing creation of new variable of this type. If variables exist for the Project or Design, they are shown in the corresponding tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Importing and Exporting Variables

Once you have defined Design and/or Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for Design Variables and Project variables include **Import...** and **Export...** buttons.



Working with Project Variables

Project variables are available across all hierarchical levels of a project. A project variable can be assigned to a parameter value in the Mechanical project in which it was created. Mechanical

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

differentiates project variables from other types of variables by prefixing the variable name with the dollar sign symbol ($). You can manually include the $ in the project variable's name when you create it, or Mechanical will automatically append it after you define the variable.

The **Project Variables** menu item allows you to access three tabs:

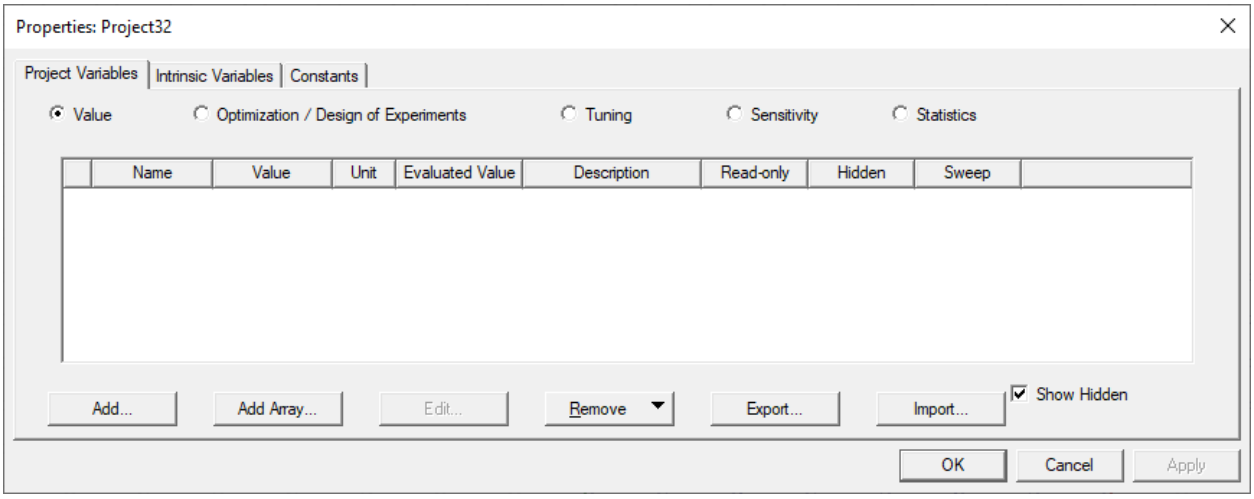
- **Project Variables** – variables, which you define, that apply to the entire project.
- **Intrinsic Variables** – pre-defined variables that cannot be changed. See: List of Intrinsic Variables.
- **Constants** – pre-defined constants that cannot be changed. See: List of Constants.

Adding a Project Variable

To define a Project Variable:

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**.

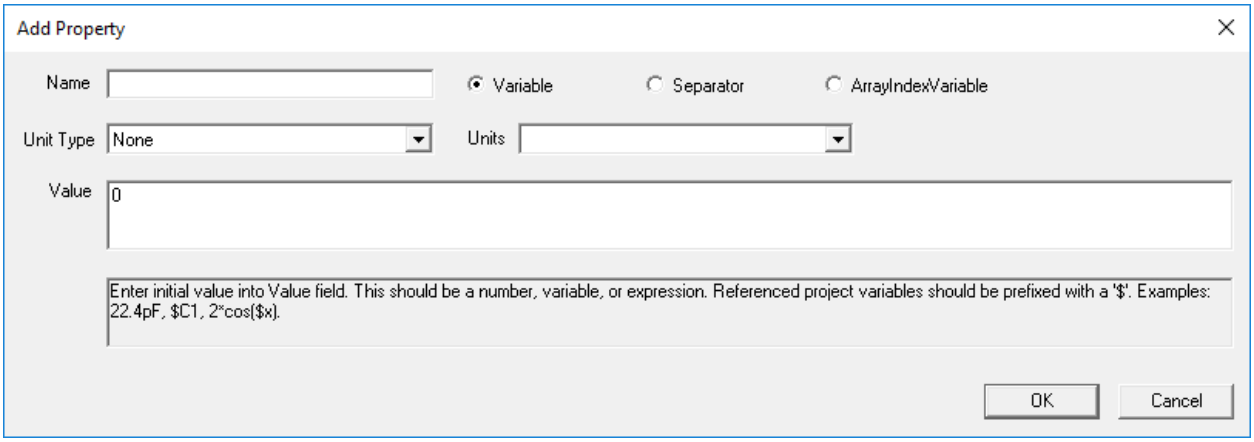
   The **Properties** window appears, on the **Project Variables** tab.



2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – use the **Variable**, **Separator**, and **ArrayIndexVariable** radio buttons to select the variable type.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value

   - **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...). “None” is the default.

     When you select a Unit Type, the choices in drop-down menu for the Units text box adapt to that unit type. For example, selecting Length as the Unit Type causes the Unit menu to show a range of metric and english units for length. Similarly, if you select the Unit Type as Resistance, the Units drop down lists a range of standard Ohm units.

   - **Units** – for Variables, use the drop-down menu to select a unit of measure.
   - **Value** – for Variables and ArrayIndexVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

   The new variable appears in the list. You can sort project variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

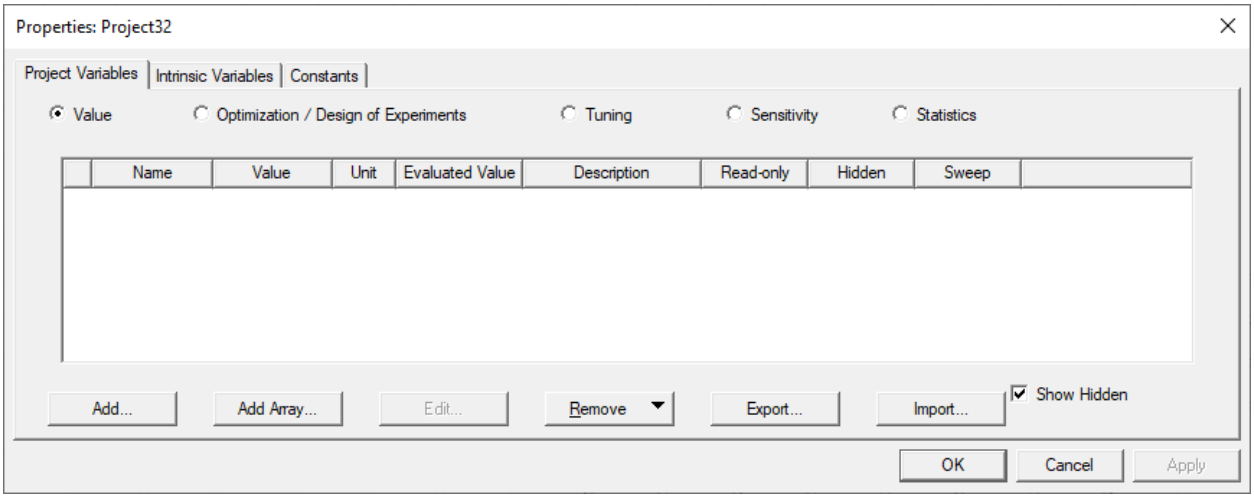
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
   - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click OK to dismiss the warning.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing and Exporting Project Variables

Once you have defined Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for and Project variables include **Import...** and **Export...** buttons.



When you click **Export....** a dialog lets you name the file and select a location. When you **Save**, if the file exists, an overwrite prompt should pop up.

Deleting a Project Variable

To remove a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

2. Remove one or more variables:
   - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
   - To remove all unused variables, click **Remove** > **Remove All Unused**.
   - To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Editing a Project Variable

To edit a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Adding a Project Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values.

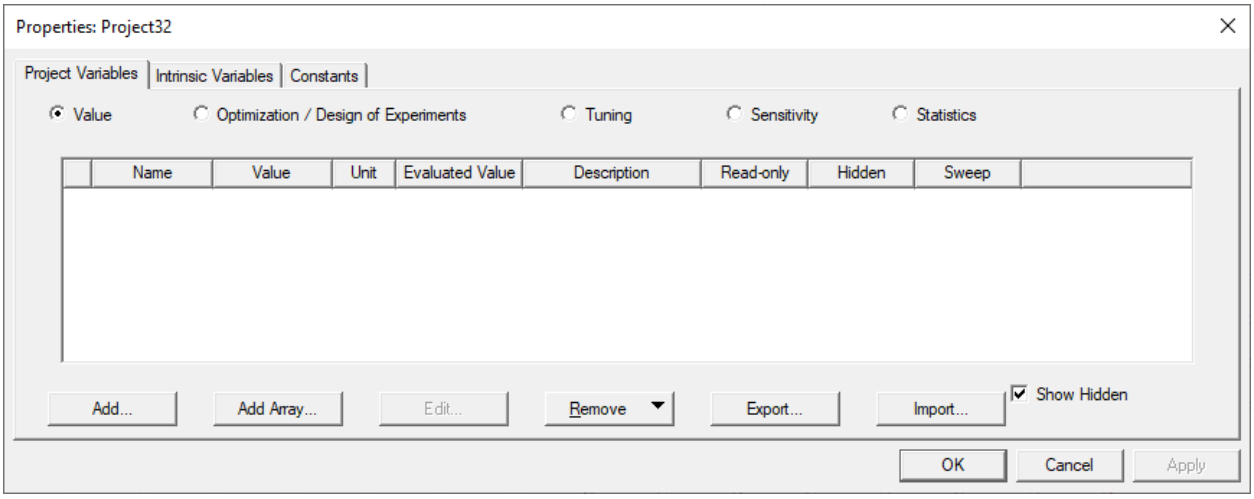
> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:
>
> - V_PRBS
> - V_PRBS_JITTER
> - V_PRBSD
> - V_PRBSG_JITTER
> - V_PSK
> - V_QAM
> - V_CPM

To define an array variable, you must first define a Project Variable Array:

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**

   The **Properties** window appears, on the **Project Variables** tab.

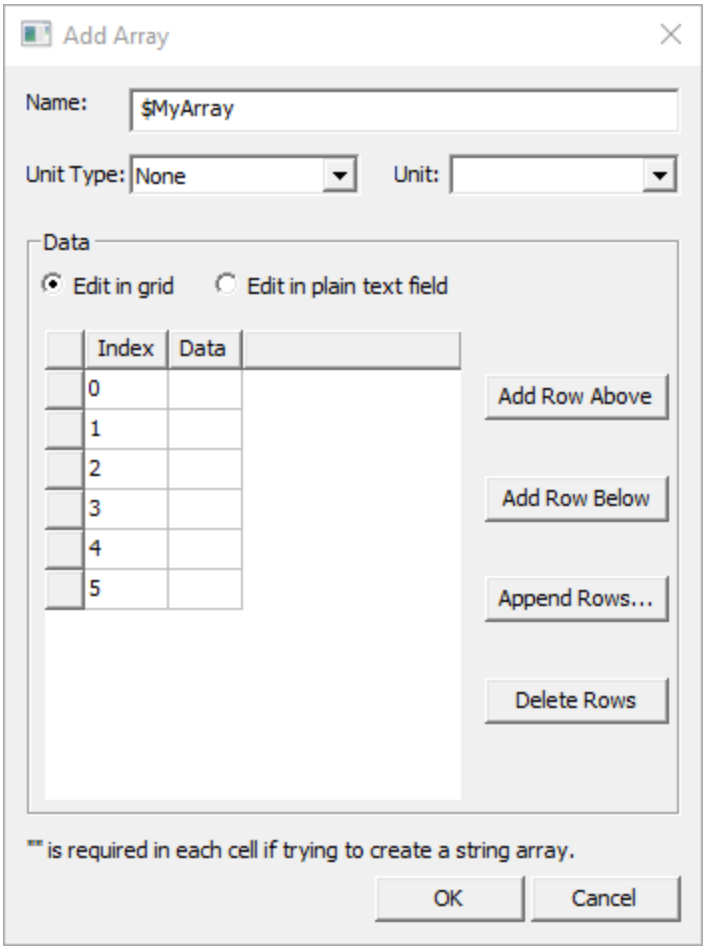
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add Array**.

   The **Add Array** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

3. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
4. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

> **Note:**
>
> Quotation marks (“ ”) are *required* as delimeters when array values are entered in either the grid or text field.

5. When you have finished entering values, click **OK**.

   The array appears in the **Project Variables** list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

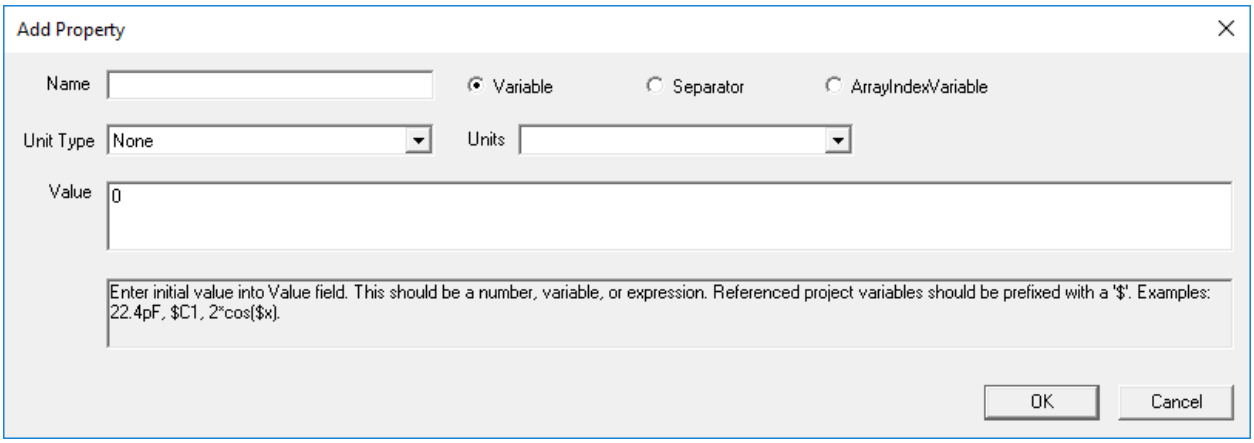
You can add a variable to the array by adding an ArrayIndexVariable on the **Project Variables** tab.

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Project Variables**.

   The **Properties** window appears, on the **Project Variables** tab.

2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

List of Intrinsic Variables

AEDT recognizes a set of intrinsic variables that can be used to define expressions. Intrinsic variable names are reserved and may not be used as user-defined variable names.

To view the list of Intrinsic Variables, click **Project** > **Project Variables** and select the **Intrinsic Variables** tab.

Name	Unit	Description
_Empty		Empty value, taken to be model default by simulator (Twin Builder)
_I1	mA	Armature current; **this variable is only used in RMxprt**.
_I2	mA	Exciting current; **this variable is only used in RMxprt**.
_I3	mA	Current; **this variable is only used in RMxprt**.
_I4 through _I9		Not used
_t,**_u**,**_v**		Variables to define parametric equation-based curve.
_V1 to **_V9**	mV	Port Voltage in user-defined model (V).
Ang	Ang	Angle. Postprocessing variable
Budget Index		Postprocessing variables; cannot be set by the user
Distance	mm	
Electrical Degree	deg	Electric degree of the rotating machine; cannot be set by the user.
F	GHz	Frequency of the circuit/system analysis
F1, **F2**, **F3**	GHz	Frequency tones 1, 2, 3 in harmonic balance analysis. (Hz).
FNoi	GHz	Offset noise frequency in harmonic balance noise analysis.
Freq	Hz	Frequency for analysis and postprocessing; cannot be set by the user

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Unit	Description
Hmax	ns	
Hmin	ns	
Ia, **Ib**	mA	Postprocessing variables; cannot be set by the user
Index		Identifier for a data point; cannot be set by the user
IWave Phi, **IWave Theta**	deg	Incident wave spherical coordinate variables. Phi is the angle from the origin in the z direction and Theta the angle from the x-axis.
Normalized Deformation, **Normalized Distance**		Postprocessing variables; cannot be set by the user
OP	mW	Postprocessing variables; cannot be set by the user
Pass		Post processing variables, not settable by the user.
Phase	deg	Angle of a complex number.
Phi	deg	The angle measured from the x-axis and can be from 0 to 360 degrees.
R	mm	R is the cylindrical coordinate system variable.
Rho		Rho is the spherical coordinate system variable.
Rspeed	rpm	Speed of the machine.
Spectrum	GHz	Postprocessing variable.
Temp	cel/deg	Analysis temperature (deg)
Tend	ns	
Theta	deg	The angle measured from the z-axis, which is the axis perpendicular to the plane of the work space, and must be from 0 to 180 degrees.
Time	ns	Time point in transient analysis
Time0	ns	Time 0 point in a transient analysis
Vac, **Vbe**, **Vce**, **Vds**, **Vgs**	mV	Post processing variables; cannot be set by the user
X,**Y**,**Z**	mm	Point coordinate variables in the modeler.
ZAng, **XRho**	deg	A spherical coordinate system; cannot be set by the user

List of Constants

To view the list of constants, click **Project** > **Project Variables** and select the **Constants** tab.

Name	Value	Description
abs0	-273.15	Absolute zero (C)
boltz	1.3806503e-23	Boltzmann constant (J/K)
c0	299792458	Speed of light in a vacuum (m/s)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Value	Description
e0	8.854187817e-12	Permittivity of vacuum (F/m)
elecq	1.602176462e-19	Electron charge (C)
eta	376.730313461	Impedance of vacuum (Ohm)
false	0	Boolean False
g0	9.80665	Acceleration due to gravity of Earth (m/s^2)
mathE	2.718281828	Euler's number (Napier's constant)
pi	3.1415926535898	Ratio of circumference to diameter
planck	6.6260755e-34	Planck's constant (m^2*kg/s)
true	1	Boolean True
u0	1.2566370614359e-06	Permeability of vacuum (H/m)

Working with Design Variables

Design Variables (Local Variables) are only available within the Mechanical design for which they are defined. A design variable can be assigned to a parameter value in the design. You can also add a variable defined with an array of values.

The **Design Properties** menu item allows you to access up to three tabs, depending on your design:

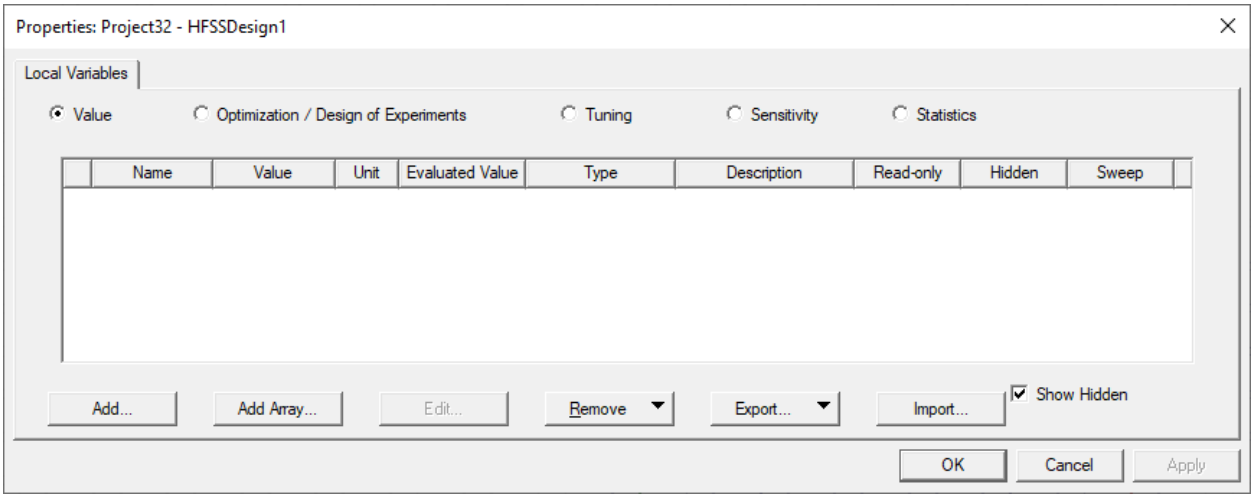
- **Parameter Defaults** – local variables with default values that can be overridden in instances of a design. For example, if three subcircuit instances contain a parameter default C1 that is defined as equal to 11.3pF, C1 may be overridden as 11.8pF in the first instance, overridden as 10.9pF in the second, and left at its default value of 11.3pF in the third. A property value that has been set by means of a parameter default is called a *passed parameter*. Parameter defaults can be defined at the design level.
- **Local Variables** – variables, which you define, that apply to the current design only. Local variables can be defined at the design level.
- **General** – pre-defined design variables that cannot be changed. You cannot use the names of these variables when creating a new variable.

> **Note:**
>
> Not all tabs appear for all designs.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining Local Variables at the Design Level

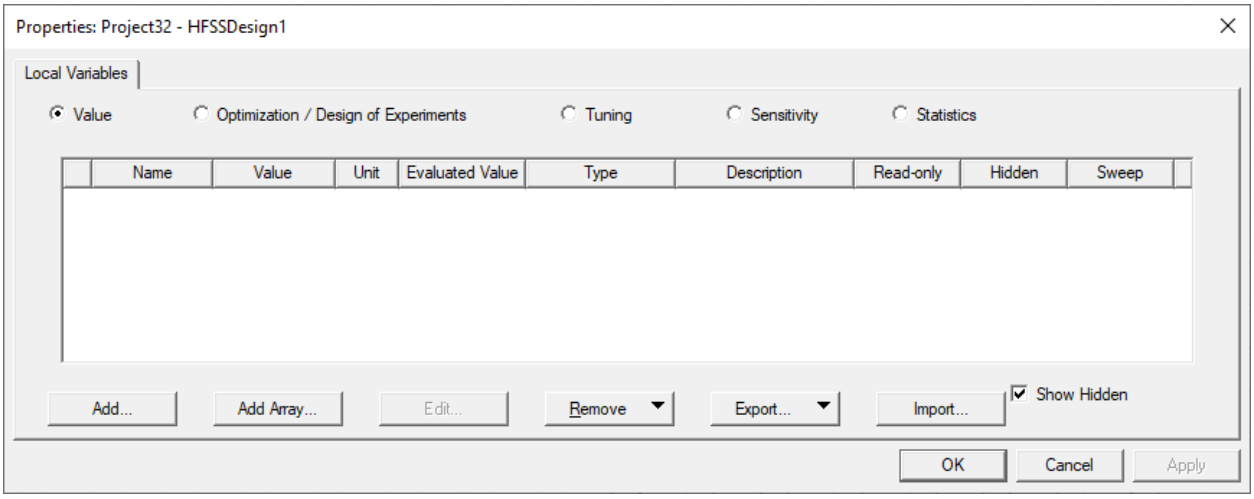
Adding a Local Variable

To define a Local Variable:

1. Access **Design Properties** one of three ways:
   - Click **Mechanical** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

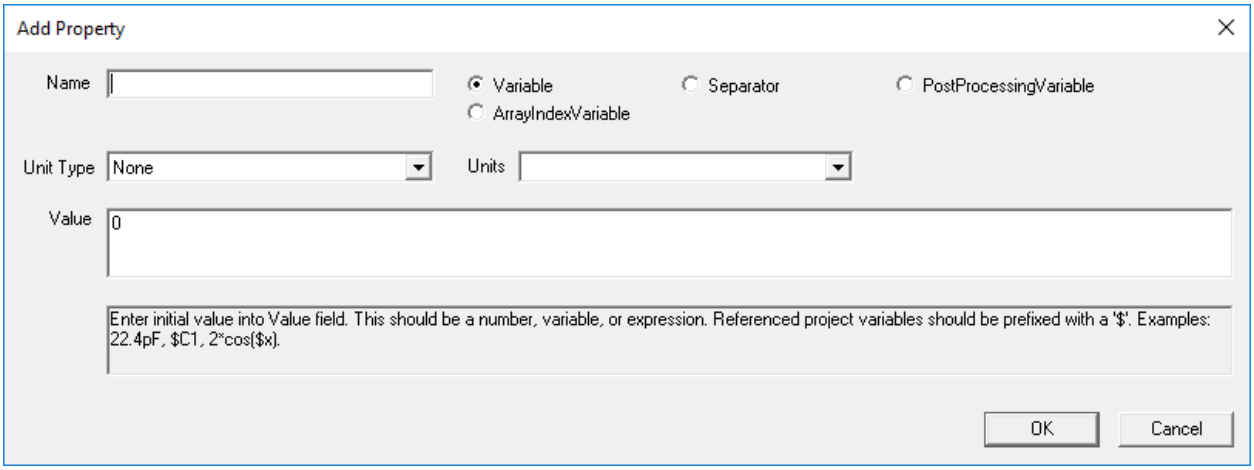
   The **Properties** window appears with the **Local Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add**.

   The **Add Property** window appears.



3. Enter information for the variable, as applicable:
   - **Name** – the name must start with a letter and can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – use the **Variable**, **Separator**, **ArrayIndexVariable**, and **PostProcessingVariable** radio buttons to select the variable type.

     > **Note:**
     >
     > Not all variable types are available for all designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value
PostProcessingVariable	Unit Type, Units, Value.

- **Unit Type** – for Variables and PostProcessingVariables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for Variables and PostProcessingVariables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, ArrayIndexVariables, and PostProcessingVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

  For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

  > **Warning:**
  >
  > If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

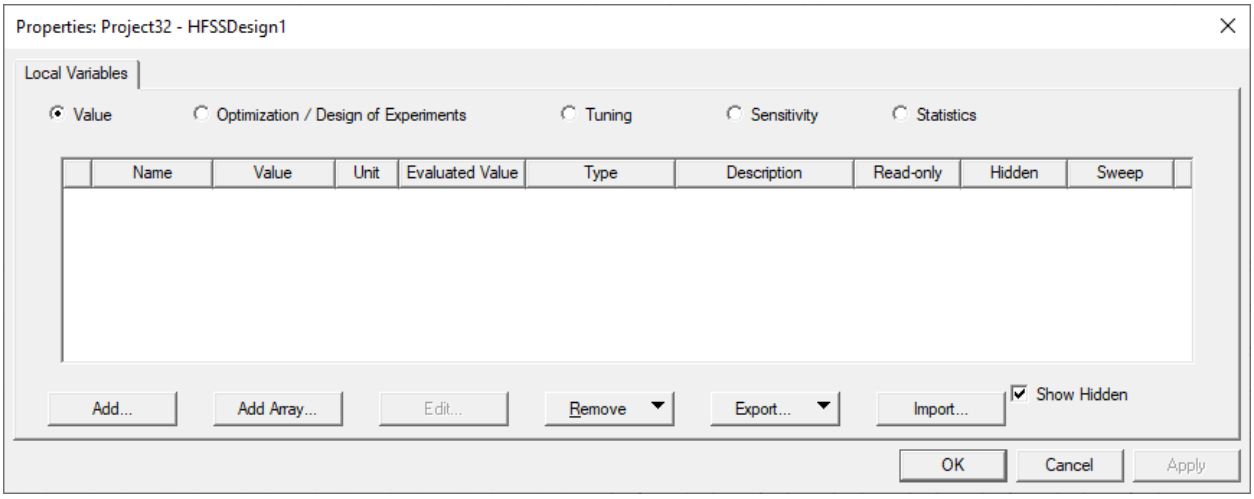
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
    - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
    - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
    - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click OK to dismiss the warning.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.



Deleting a Local Variable

To remove a Local Variable:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** as described above.
2. Remove one or more variables:
    - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
    - To remove all unused variables, click **Remove** > **Remove All Unused**.
    - To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Editing a Local Variable

To edit a Local Variable:

1. Access **Local Variables** as described above.
2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Local Variable

You can override a variable value from an analysis setup's **General** tab.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Defining Parameter Defaults at the Design Level

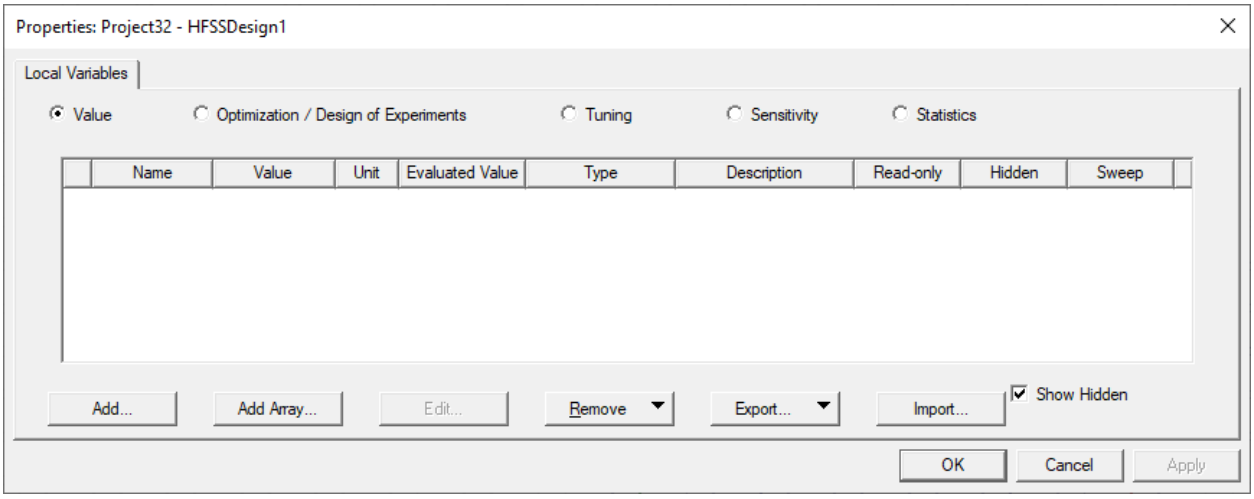
Adding a Parameter Default

To define a new parameter default at the design level:

1. Access **Design Properties** one of three ways:
    - Click **Mechanical** > **Design Properties**.
    - Right-click the design name in the Project Tree and select **Design Properties**.
    - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

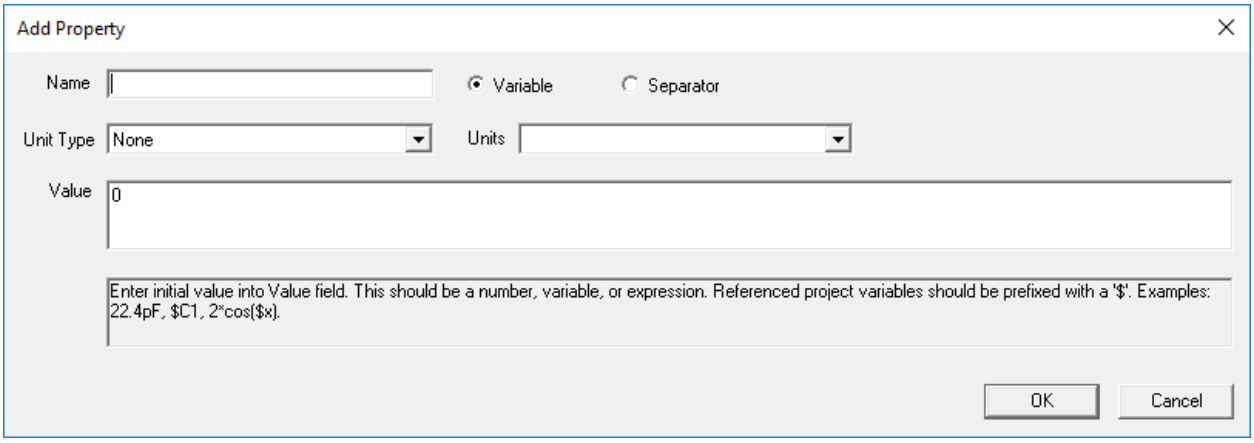
    The **Properties** window for design variables appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add...**.

   The **Add Property** window appears.



3. Enter information for the parameter default, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – select **Variable** to enter a value, or select **Separator** to provide a bolded name for a blank line.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Type	Editable Properties
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.

- **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for Variables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, enter a valid numeric quantity. Alternately, you can assign a variable to a parameter by typing a variable name or mathematical expression in place of a parameter value.

4. Click **OK**.

   The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

Deleting a Parameter Default

To remove a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select a parameter from the list and click **Remove**.

Editing a Parameter Default

To edit a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select the parameter you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Parameter Default

You can override a parameter default from an analysis setup's **General** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Adding a Design Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values. Design variable arrays are only available within the Mechanical design for which they are defined.

> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:
>
> - V_PRBS
> - V_PRBS_JITTER
> - V_PRBSD
> - V_PRBSG_JITTER
> - V_PSK
> - V_QAM
> - V_CPM

To define an array variable, you must first define a Local Variable Array:

1. Access **Local Variables** one of three ways:
    - Click **Mechanical** > **Design Properties**.
    - Right-click the design name in the Project Tree and select **Design Properties**.

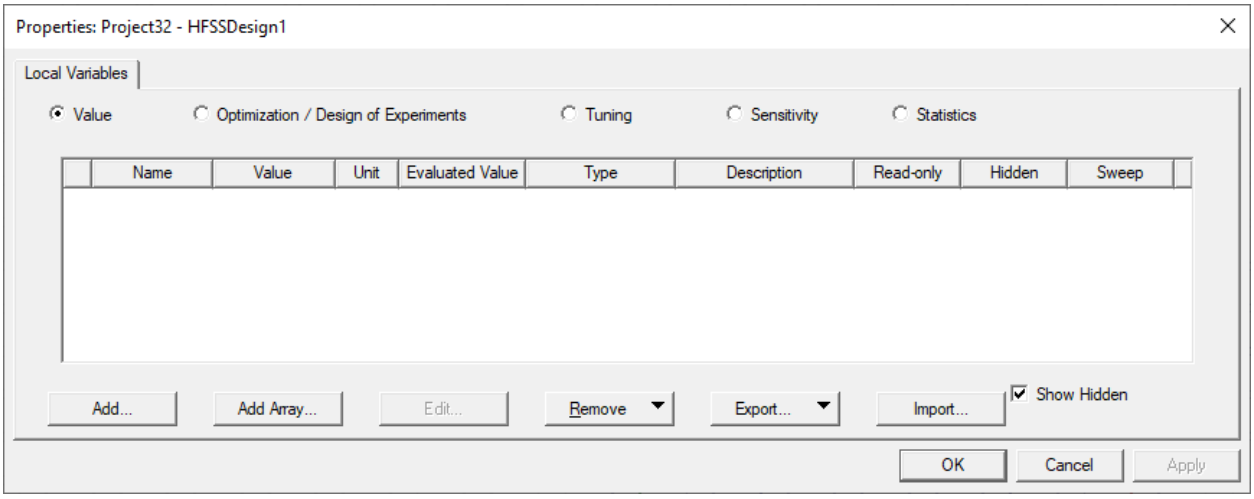
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

The **Properties** window appears.

2. Select the **Local Variables** tab.

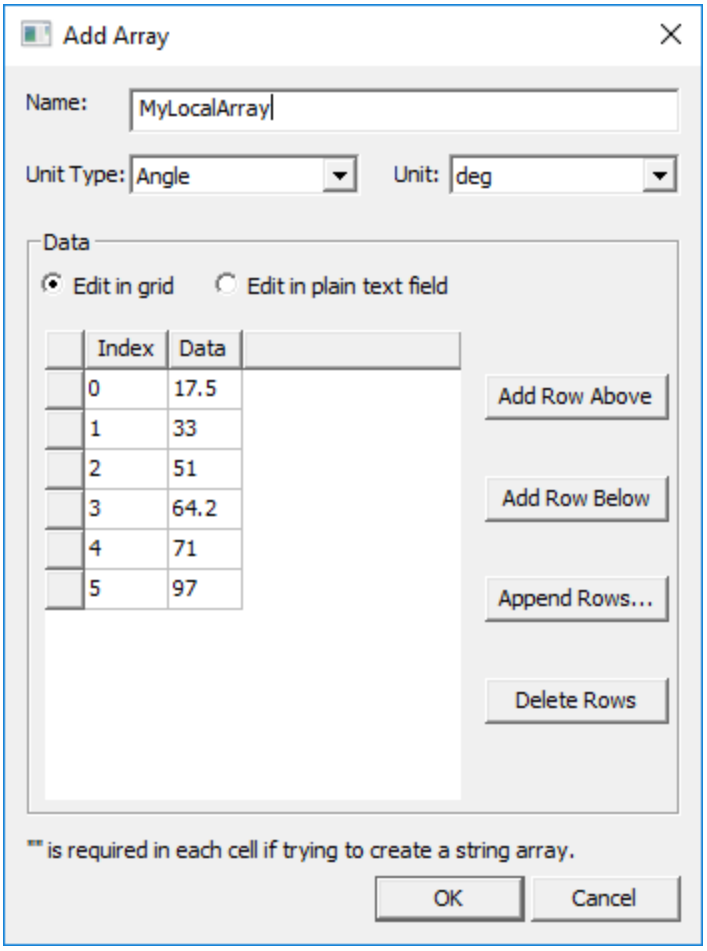
   The **Local Variables** tab displays.



3. Click **Add Array**.

   The **Add Array** window appears.

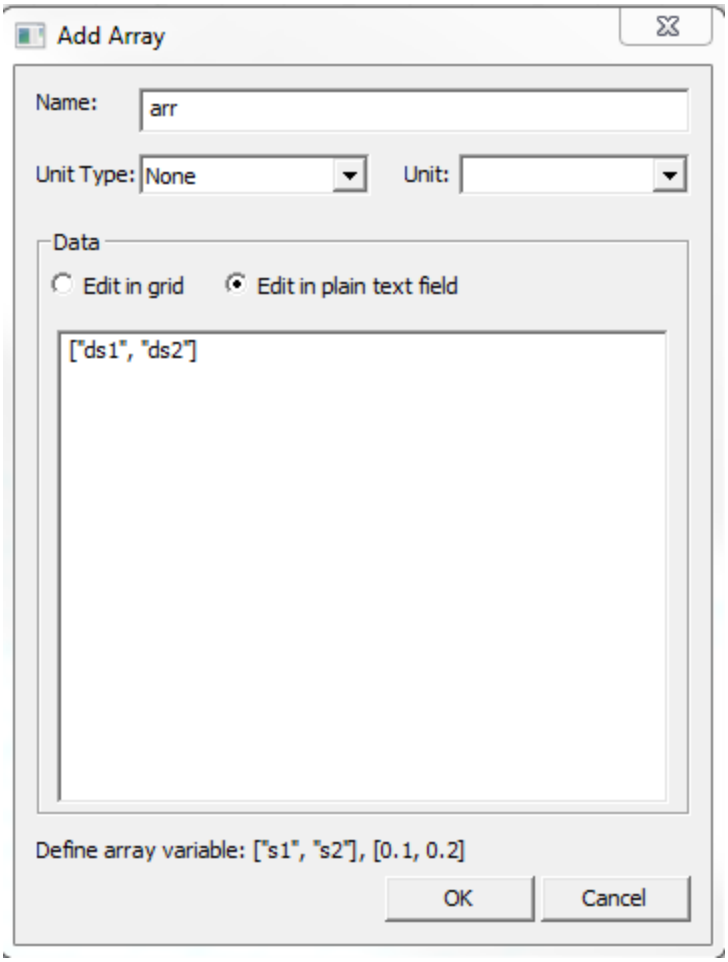
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

If you elected to **Edit in plain text field** in the Add Array dialog box, the bracketed and comma delimited format is used.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
5. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

   > **Note:**
   >
   > Quotation marks ("") are *required* as delimeters when array values are entered in either the grid or text field.

6. When you have finished entering values, click **OK**.

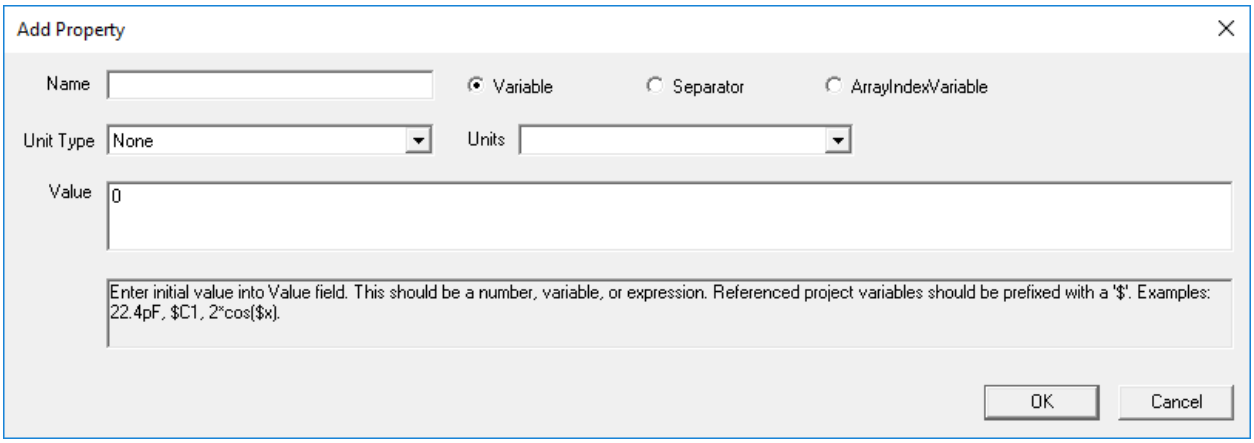
   The array appears in the **Local Variables** list.

You can add a variable to the array by adding an ArrayIndexVariable on the **Local Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** one of three ways:
   - Click **Mechanical** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.
2. From the **Local Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

Converting Variables and Parameter Defaults

Design (Local) variables can be converted to parameter defaults, and vice versa.

To convert a parameter default or local variable:

1. Access **Design Properties** one of three ways:
   - Click **Mechanical** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.

   The **Properties** window appears.
2. Select either the **Parameter Defaults** tab or the **Local Variables** tab, depending on which item you want to convert.
3. Select the variable/parameter and click **Convert to Parameter** or **Convert to Variable**.

   The item will move to the list on the appropriate tab.

Some special cases to consider:

- If a local variable references another local variable, it cannot be converted to a parameter default. You will see a pop-up message if you attempt to convert. This is because local variables can reference parameter defaults, but parameter defaults cannot reference local variables.
- Parameter defaults can be overridden and can have different values in each subcircuit, while a Local Variable cannot. If a parameter default has been overridden and you convert it to a variable, it will lose all overrides. Attempting to do so will display a pop-up message asking if you really want to do the conversion and lose the override values.

Fixed vs. Non-Fixed Variables

By default, both Project variables and Design variables are included in the list of variables that index solution data. As a result, they are visible when selecting data to be displayed in plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, if you define a variable as *fixed*, it is not swept and will not index solution data. Consequently, defining variables as fixed can speed up simulation—particularly if there are many variables to simulate.

To define a variable as fixed:

1. Open the **Properties** window for either Project or Design variables.
2. Clear the **Sweep** check box next to the variable you want to be fixed.

	Name	Value	Unit	Evaluated Value	Type	Description	Read-only	Hidden	Sweep
	wg_width	7.1	mm	7.1mm	Design		☐	☐	☑
	fillet_radius	0	mm	0mm	Design		☐	☐	☐

- If a **Sweep** check box is selected (default) the corresponding variable is NOT fixed.
- If a **Sweep** check box is cleared, the variable is fixed.

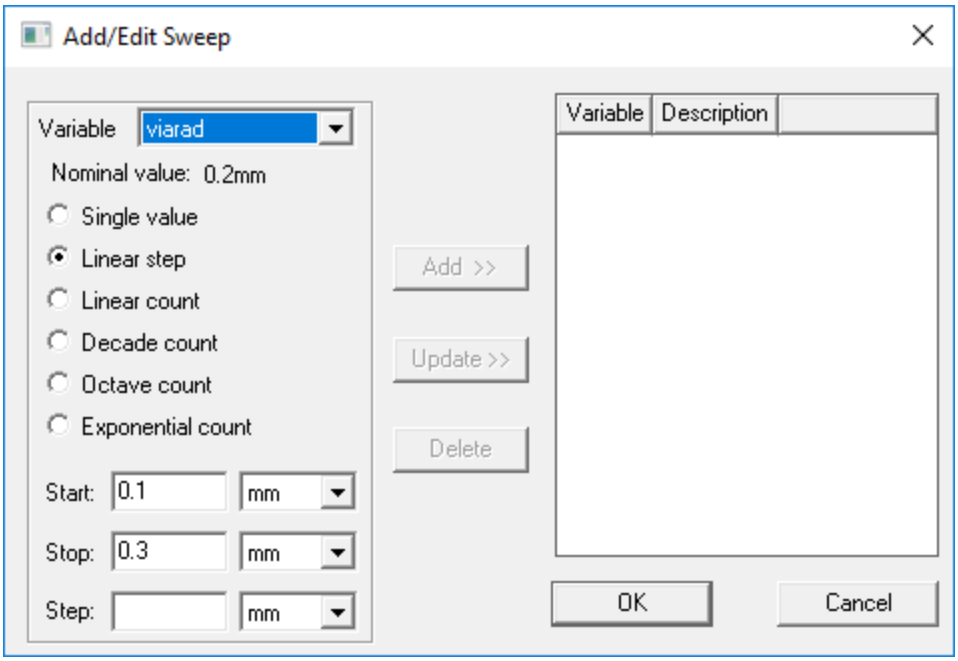
Non-Fixed Variable Sweeps

When setting up a sweep, the **Setup** window contains a list of variables being swept.

To add a non-fixed variable sweep:

1. Click **Add**.

   The **Add/Edit Sweep** window appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use the **Variable** drop-down menu to select a non-fixed variable.
3. Select the desired settings and click **Add**.
4. Click **OK** to close the **Add/Edit Sweep** window.
5. Click **OK** to exit the **Setup**.

Orphaned Sweeps

If you create a simulation setup that contains a sweep of a variable, and then subsequently clear the variable's **Sweep** check box, the variable becomes fixed and, as a result, its sweep is "orphaned." The sweep is removed from the setup and the simulation runs as if the sweep did not exist.

- When you close the **Properties** window, a warning message is added to the **Message Window** for each orphaned sweep.
- If you edit an orphaned variable in the **Properties** window and reselect its **Sweep** box, the orphaned sweep will be restored to the setup list.
- If you double-click a setup that contains orphaned sweeps, a pop-up dialog box asks if you want to delete the orphaned sweeps. If you respond **Yes**, all orphaned sweeps are deleted when the **Properties** window is closed. Even if you reselect a **Sweep** check box, its orphaned sweep will NOT be restored. This action is undoable.
- Orphaned sweeps are not written to disk when the project is saved, so once you save a project and close it, any orphaned sweeps are permanently lost.

Choosing a Variable to Optimize

Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** window.

To specify that a variable is to be used during optimization:

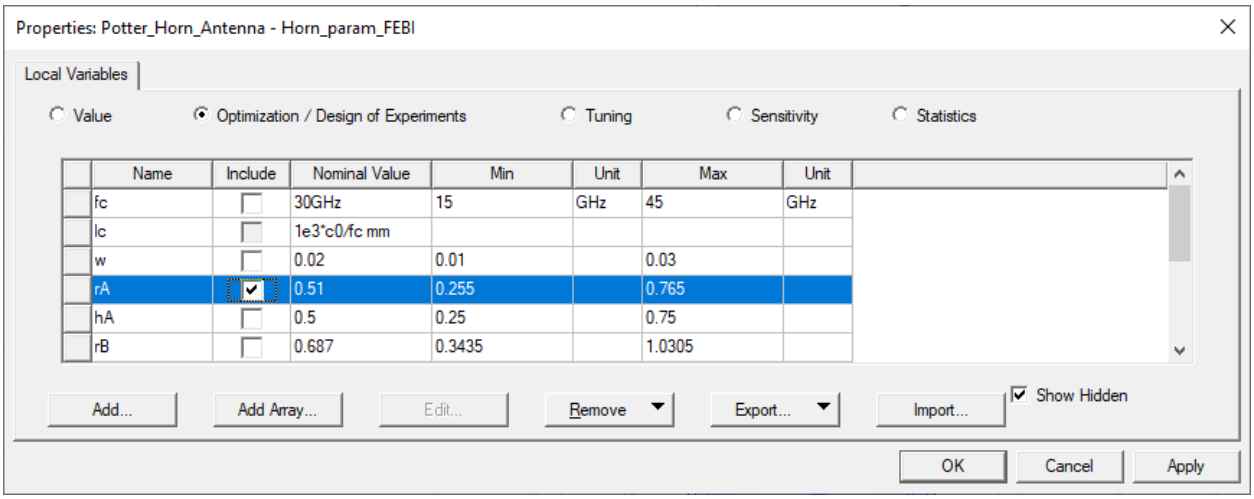
1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Mechanical** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Optimization / Design of Experiments** radio button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

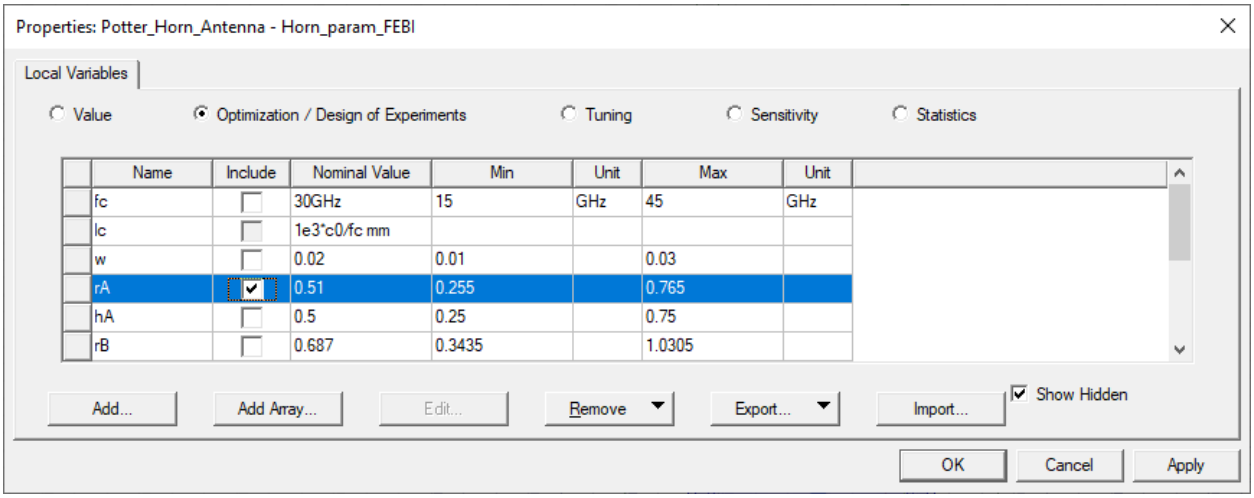
4. Use the **Include** check boxes to select variables for optimization.



Important:

Dependent variables cannot be optimized.

Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



The selected variable(s) will be available for optimization in an Optimetrics setup defined in the current design or project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every optimization analysis. During optimization, the optimizer will not consider variable values that lie outside of this range.

Choosing a Variable to Tune

Before a variable can be tuned, you must specify that you intend for it to be tuned in the **Properties** window.

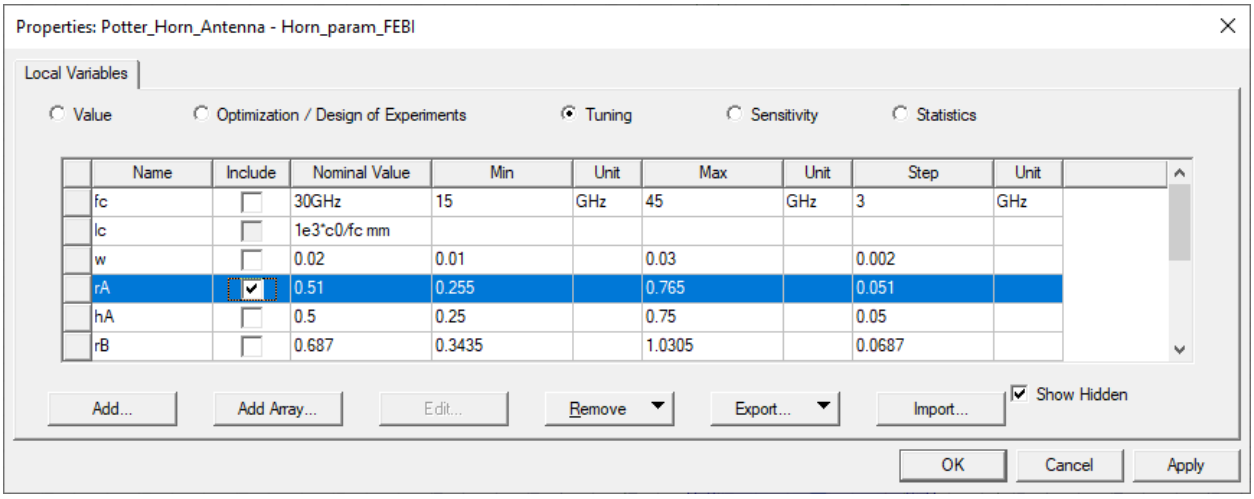
To specify that a variable is to be tuned:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Mechanical** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Tuning** radio button.
4. Use the **Include** check boxes to select variables for tuning.

> **Important:**
>
> Dependent variables cannot be tuned.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



The selected variable(s) will be available for tuning.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Including a Variable in a Sensitivity Analysis

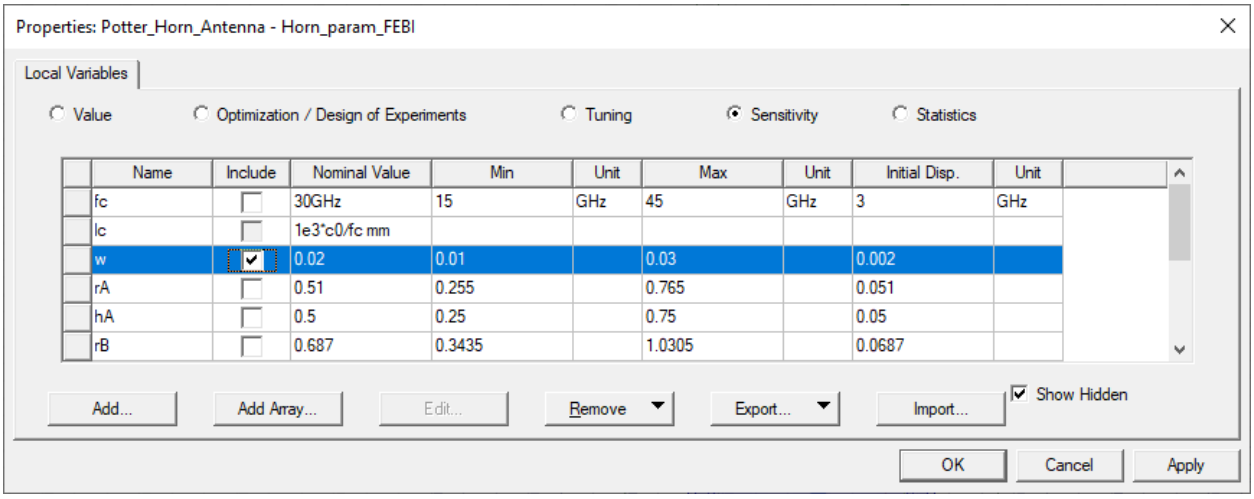
Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** window.

To specify that a variable is to be used during sensitivity analysis:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Mechanical** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to use.
3. Select the **Sensitivity** radio button.
4. Use the **Include** check boxes to select variables for sensitivity analysis.



> **Important:**
>
> Dependent variables cannot be included in a sensitivity analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

The selected variable(s) will be available for sensitivity analysis in the current design or project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every sensitivity analysis. During analysis, the optimizer will not consider variable values that lie outside of this range.
6. If desired, use the **Initial Disp.** field to override the default initial displacement value that Optimetrics will use for the variable in every sensitivity analysis. During analysis, Optimetrics will not consider a variable value for the first design variation that is greater than this step size away from the starting variable value.

Including a Variable in a Statistical Analysis

Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** window.

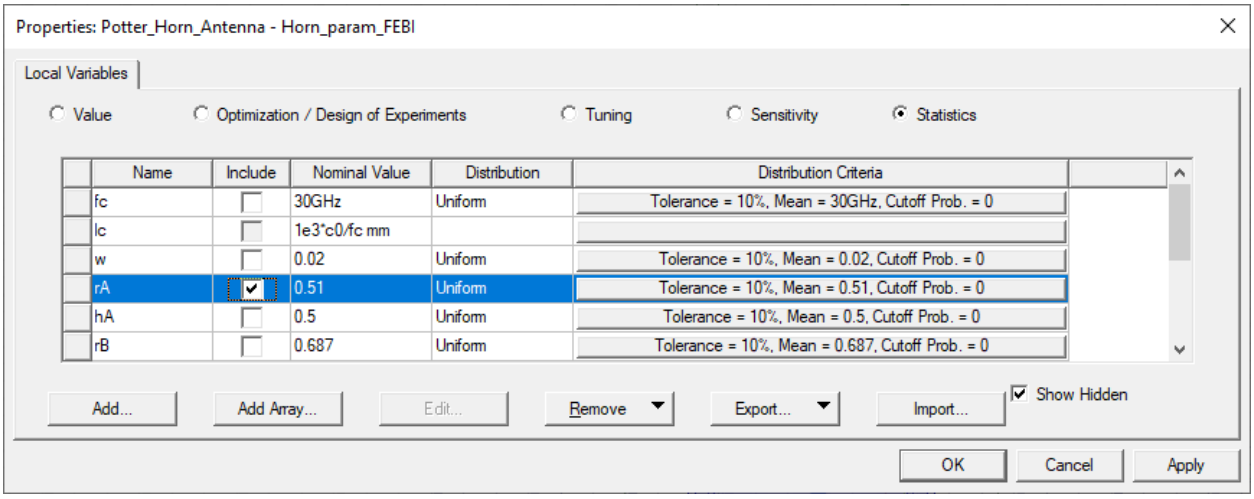
To specify that a variable is to be used during statistical analysis:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Mechanical** > **Design Properties**).

   The **Properties** window appears.
2. Use the tabs to navigate to the variable you want to use.
3. Select the **Statistics** radio button.
4. Use the **Include** check boxes to select variables for statistical analysis.

> **Important:**
>
> Dependent variables cannot be included in a statistical analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selected variable(s) will be available for statistical analysis in the current design or project.

5. If desired, click the **Distribution Criteria** button to override the distribution criteria that Optimetrics will use for the variable in every statistical analysis.

Copying and Pasting a Variables List

From either the Project or Design variables window, you can copy a list of variables and their values to the clipboard. You can then paste them in a plain text, tab-separated format. This is useful for creating documentation.

To copy and paste the variables list:

1. Navigate to the Project Variables or Design Variables **Properties** window.
2. Right-click and select **Copy to Clipboard**.
3. Open a text editor and **paste**.

The pasted variables appear in a tab separated column format. Fields that do not contain values are left blank.

Name	Value	Unit	"Evaluated Value	"Description	Read-only	Hidden
$width	14.8570192	mm	14.8570192mm		false	false
$length	7.824547736	mm	7.824547736mm		false	false
$height	0.45*$width		6.68565864mm		false	false

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Datasets

Datasets are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot and an expression is derived from the curve that best fits the segmented plot. The created expression is then used in piecewise linear Intrinsic Functions. You can add datasets at either the project level or the design level. They can be for various purposes, including to define frequency-dependent port impedances or frequency-dependent global variables.

Project-level datasets are typically used for defining material properties at the project level (applicable to all designs in the project).

Design-level datasets can be used in geometry entities like part commands, coordinate systems, points, and planes. Design-level datasets do not work with equation-based surfaces or curves. Design-level datasets can be used directly with piecewise linear functions in expressions or indirectly through variables that can refer to the dataset.

Design-level datasets can also be used in the following operations:

- **Creating or editing geometry** – when a geometry uses a dataset directly, edit dataset invalidates the solution. When a geometry uses a variable that is defined by dataset, editing the dataset does not invalidate the solution.
- **Creating an Animation** – based on a variable which can index in datasets.
- **Copying and Pasting Geometry** – if a part refers to a design dataset, it will be pasted to destination design.

Datasets are provided for Mechanical designs.

The **Datasets** dialog box provides a browsable listing of all datasets currently defined for the project or design. A preview window displays a plot of the currently selected dataset. Controls allow you to **Add**, **Edit**, **Remove**, and **Clone** datasets; to **Import** and **Export** characteristic data; and to launch the **SheetScan** tool to extract data from graphics such as data sheets.

To access the **Datasets** window:

- For project-level datasets, including 3D datasets, click **Project** > **Datasets**.
- For design-level datasets, click **Mechanical** > **Design Datasets**.

Adding Datasets

The following procedure describes how to add a dataset manually.

You can also add datasets:

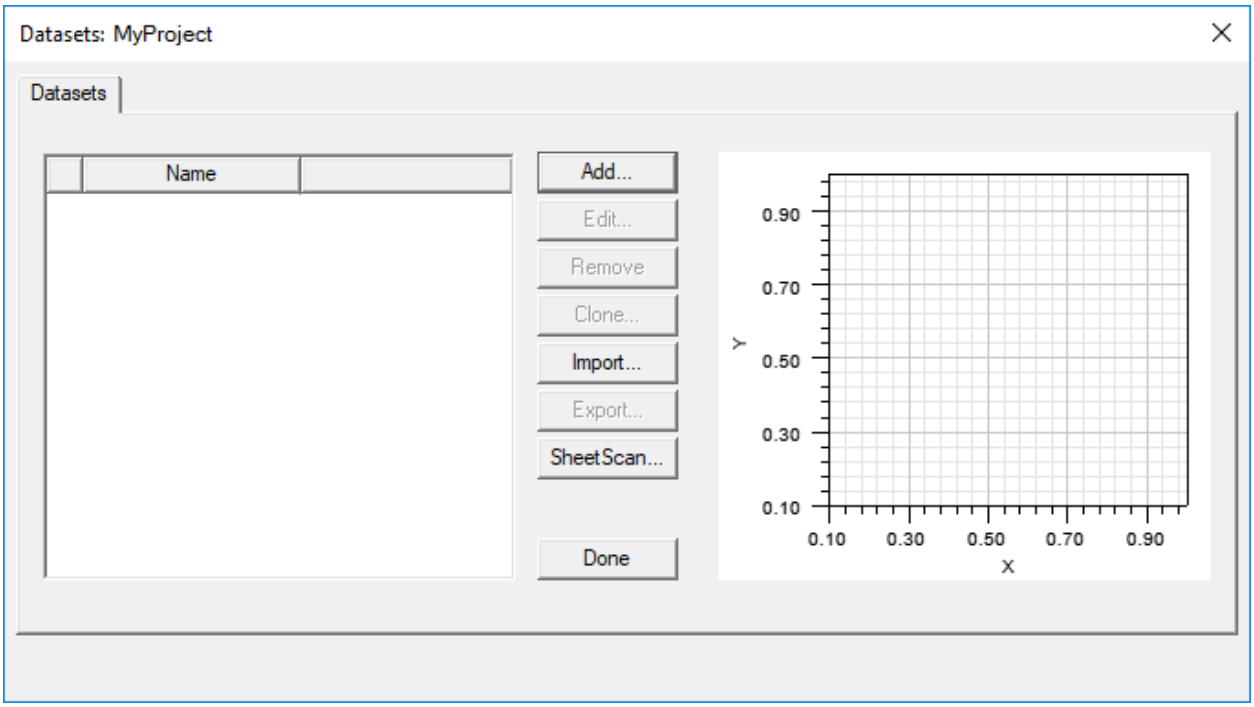
- By importing data from an external tab delimited file using the **Import...** button.
- Using the Sheetscan tool.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To add a dataset manually:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets** from the menu bar, or click **Datasets** on the **Desktop** ribbon tab.
   - For design-level datasets, click **Mechanical** > **Design Datasets** from the menu bar.

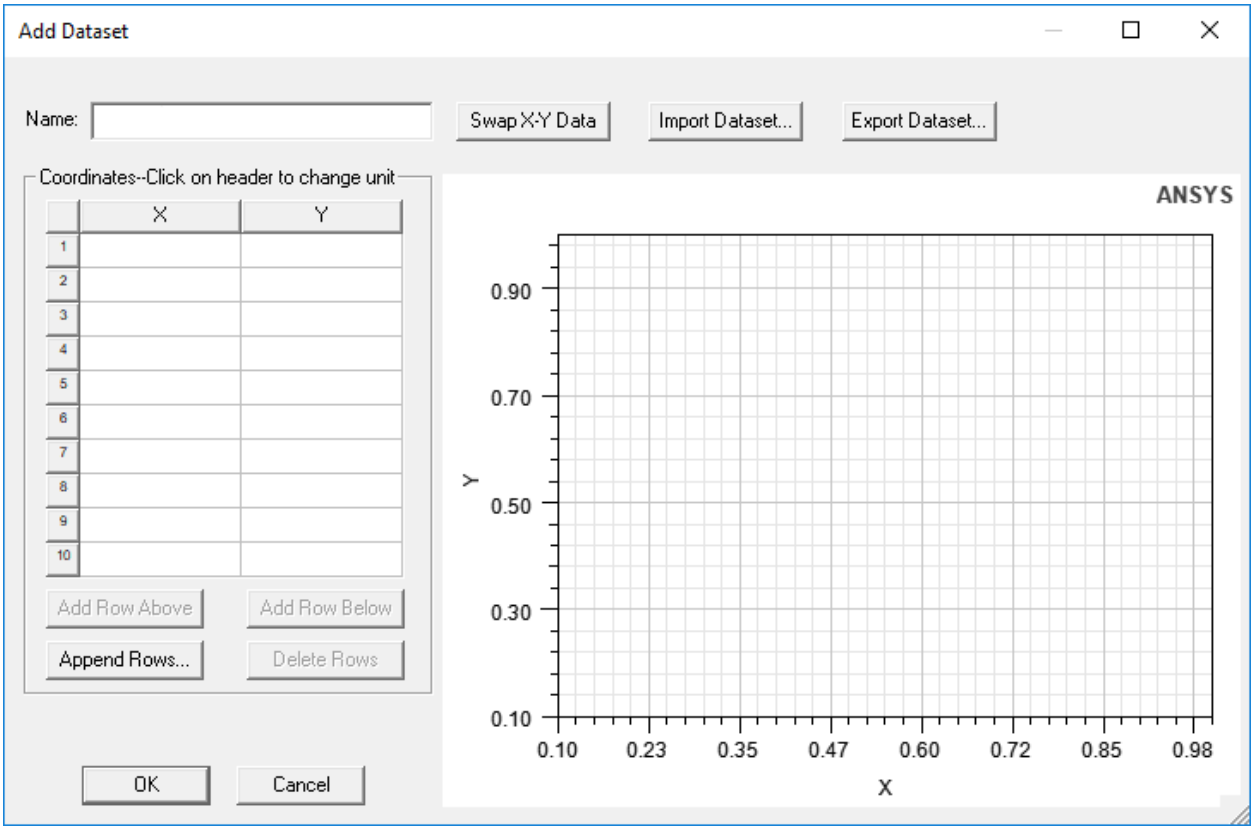
   The **Datasets** window appears, listing any project or design-level datasets.



2. Click **Add**.

   The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left side of the **Add Dataset** window contains fields for the dataset name and a table for X- and Y- coordinates. The right side contains a graphic display that draws a line for the coordinates you add. It also includes buttons for the following functions:

- **Swap X-Y Data** – this swaps the X- and Y- coordinates and adjusts the graphical display.
- **Import Dataset** – this provides a way to import data sets from an external source. The format is a tab-separated points file. Clicking the button opens a file browser window.
- **Export Dataset** – this provides a way to export the current dataset to a tab separated points file. Clicking the button opens a file browser window.
- **Add Row Above** – adds a new row to the table above the selected row.
- **Add Row Below** – adds a new row to the table below the selected row.
- **Append Row** – opens a dialog box that lets you specify a number of rows to add to the table.
- **Delete Row** – deletes the selected row or rows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

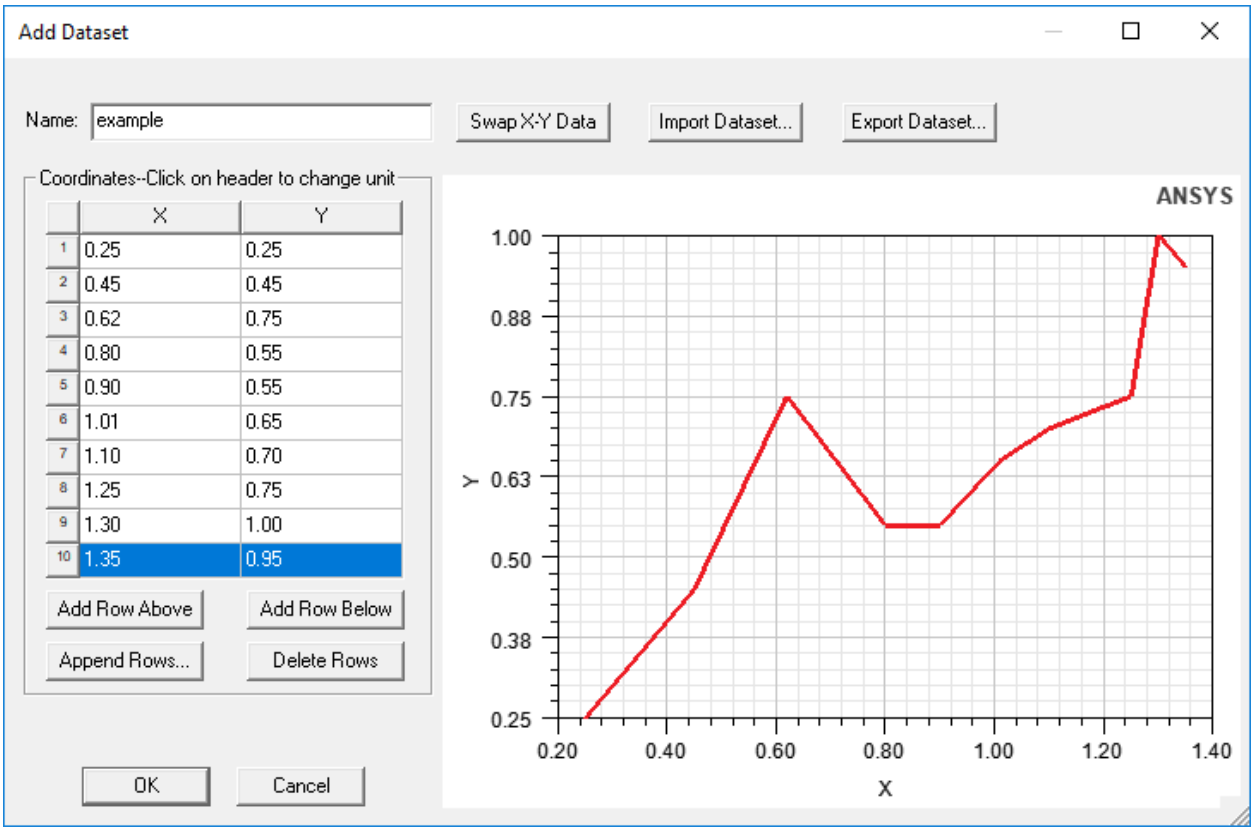
3. Enter a **Name** for the dataset.

> **Note:**
>
> Project-level datasets must begin with a dollar sign character ($). If you do not type one, it will be added for you.

4. Enter X/Y coordinates, either by manually typing or by clicking **Import Dataset**.

   If desired, click the **X** and **Y** column headers to change the unit.

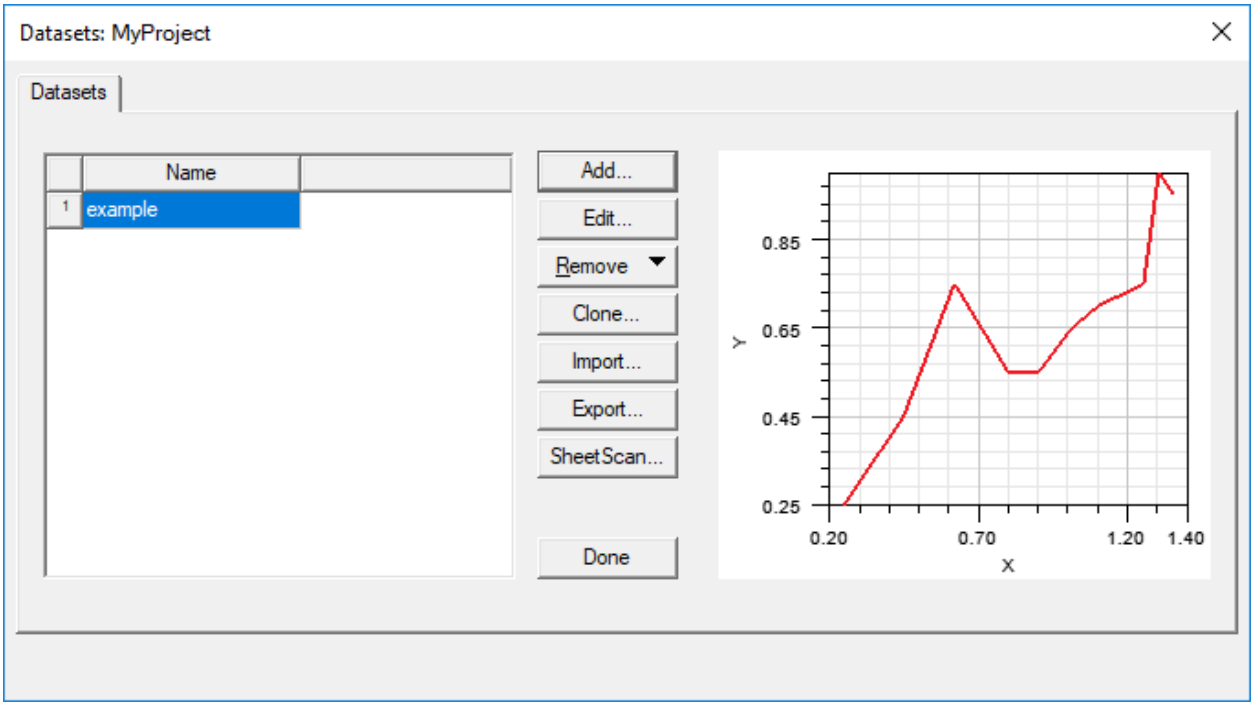
   The display updates to show your coordinates.



5. When you are finished, click **OK** to exit the **Add Datasets** window.

   The new dataset appears in the **Datasets** list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dataset plot is extrapolated into an expression that can be used in parametric analyses, boundary definitions, or assigned to a material property value.

You can now edit, remove, clone, or export the dataset. You can also change the plot display properties.

6. Click **Done** to exit the **Datasets** window.

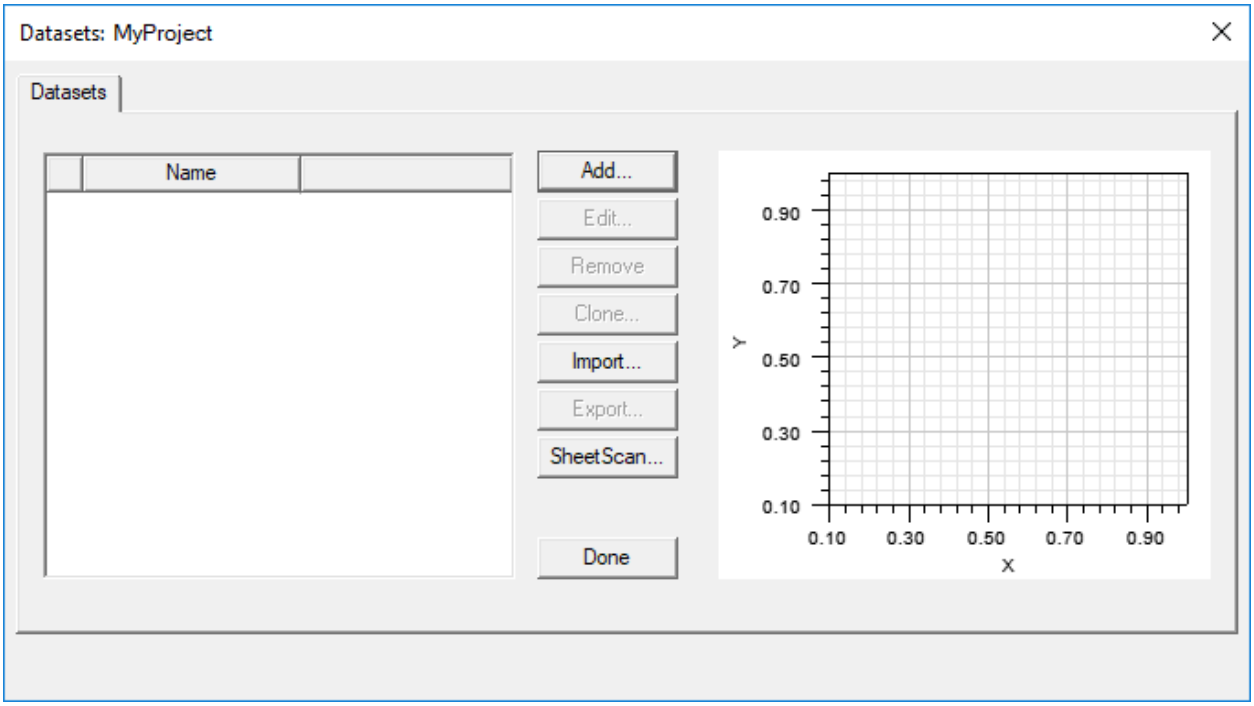
Importing Datasets

To import a dataset from a file:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Mechanical** > **Design Datasets**.

   The **Datasets** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Import**.
3. In the file browser that appears, use the drop-down menu to select the file format of the file you wish to import.

> **Note:**
>
> The table below lists the file types supported for *Project level datasets*. For *Design level datasets*, only **.tab** files are supported.

.mdx, .mda	Twin Builder Characteristic format
.xls, .xlsx	Microsoft Excel
.txt	text file
.csv	Comma-separated value
.out	Maxwell SPICE (read-only - reads data inside the KW_ DATA section)
.cfg	Comtrade (IEEE Std C37-111-1999)
.dat	TEK Oscilloscope
.tab	tab delimited data files

   a. If you select an **.mdx**, **.mda**, **.out**, **.cfg**, or **.tab** file, the data is imported immediately into **Datasets** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If you select an **.xls** or **.xlsx** file containing multiple sheets, a **Table Properties** dialog box appears, where you can choose the desired sheet from a drop-down menu. Otherwise, selecting an .xls or .xlsx file imports the data immediately.

> **Note:**
>
> Microsoft Excel is required to import .xls or .xlsx files.
>
> Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
>
> The first row of data is assumed to contain column headings and is ignored.

c. Selecting a .**txt** or .**csv** file opens an **Import** dialog box where you can specify settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

4. When satisfied with the import settings, click **OK** to import the data.

   Imported datasets take the name of the imported file. If a dataset of the same name already exists, a number will be appended to the end of the new dataset (for example, a duplicate 'dataset' becomes 'dataset1').

   Once the data has been imported, you can edit, remove, clone, or export the dataset. You can also change plot display properties.

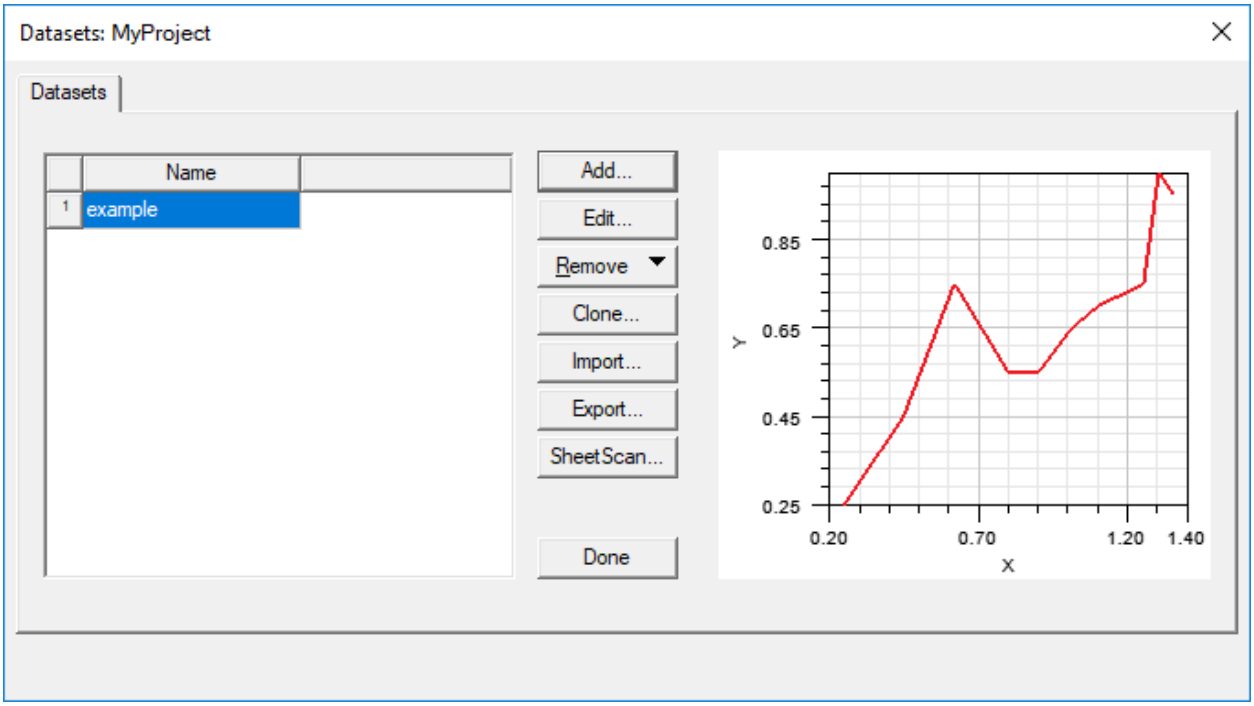
Editing Datasets

To modify an existing dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Mechanical** > **Design Datasets**.

   The **Datasets** window appears, listing any project or design-level datasets.

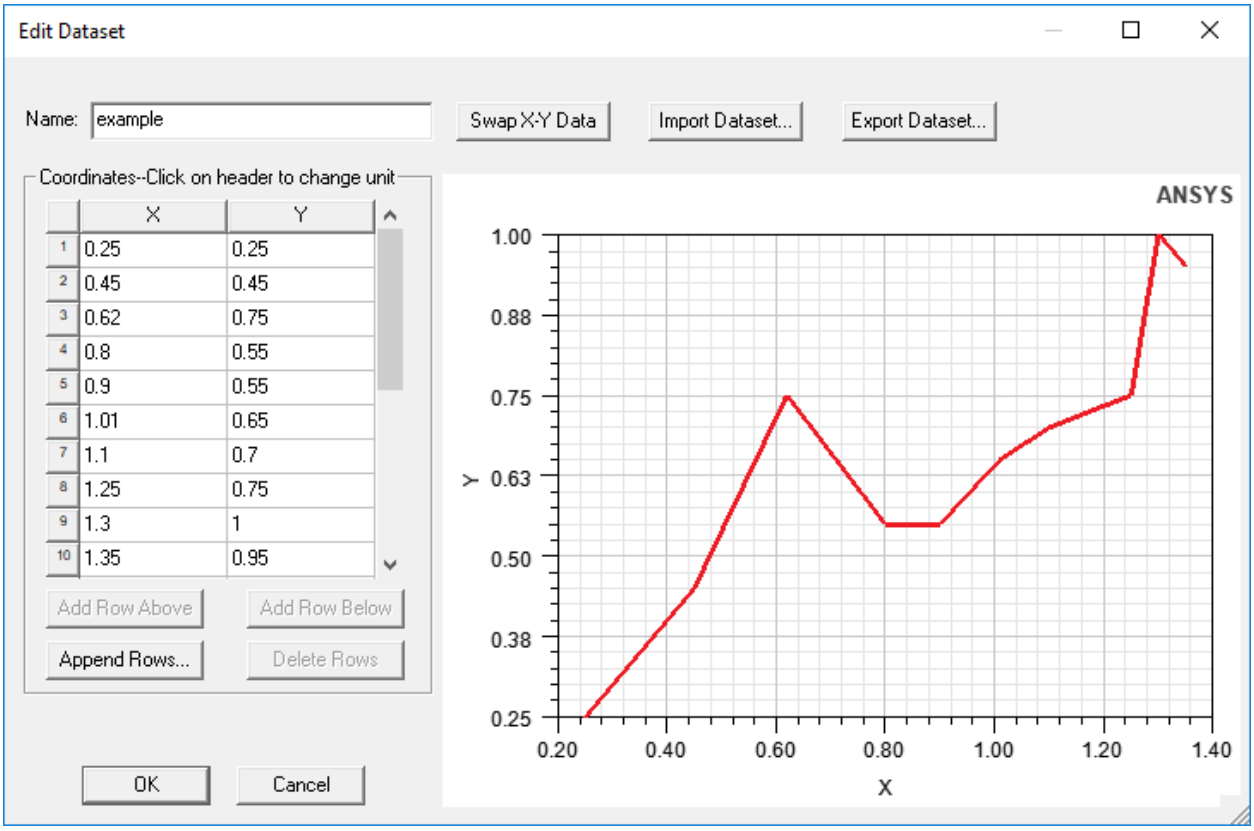
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Edit**.

   The **Edit Dataset** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Edit Dataset** window.
5. Click **Done** to exit the **Datasets** window.

Cloning Datasets

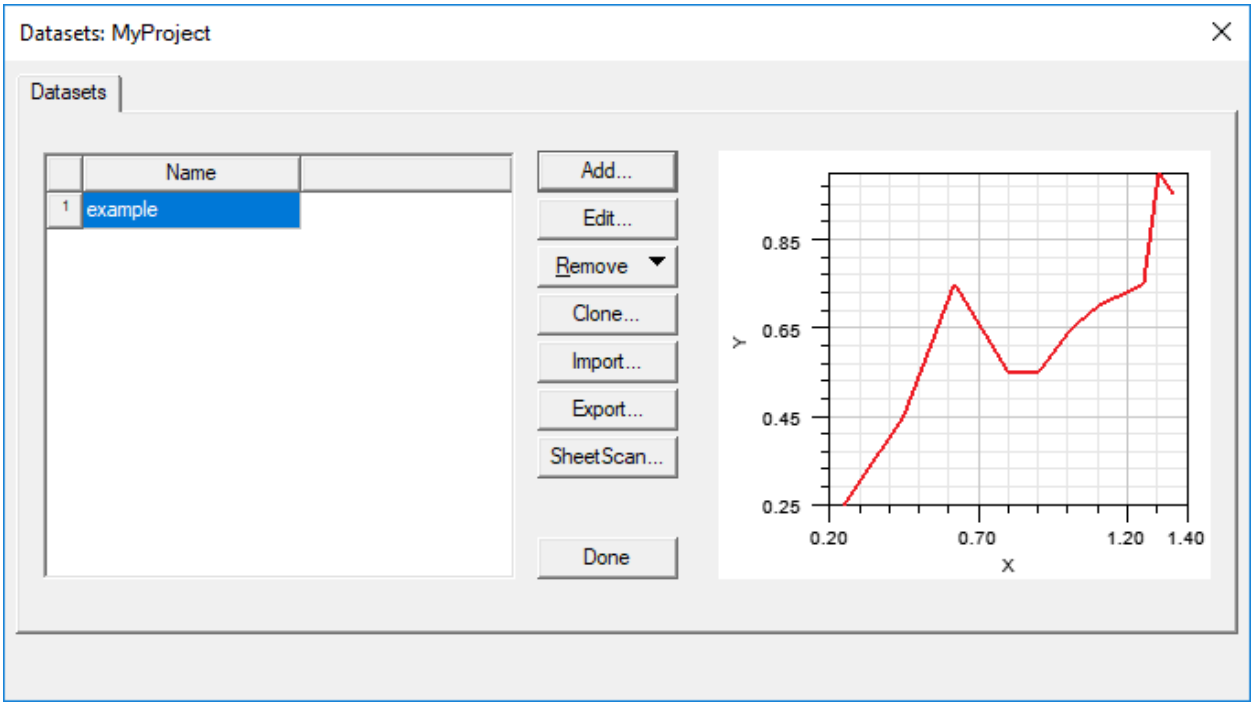
Cloning a dataset creates an identical copy of an existing dataset. The clone can then be modified as necessary.

To clone a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Mechanical** > **Design Datasets**.

   The **Datasets** window appears, listing any project or design-level datasets.

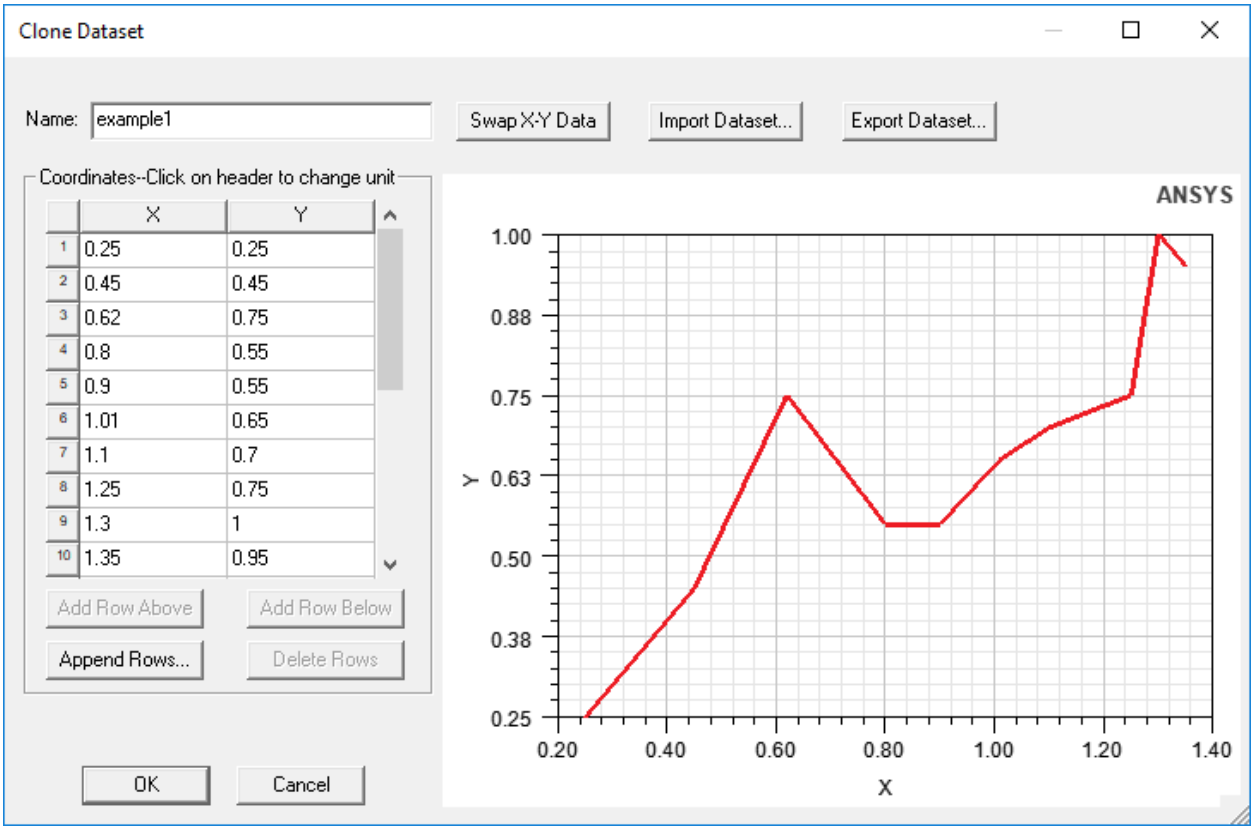
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Clone**.

   The **Clone Dataset** window appears. The default name is the original dataset name with a number appended to the end.

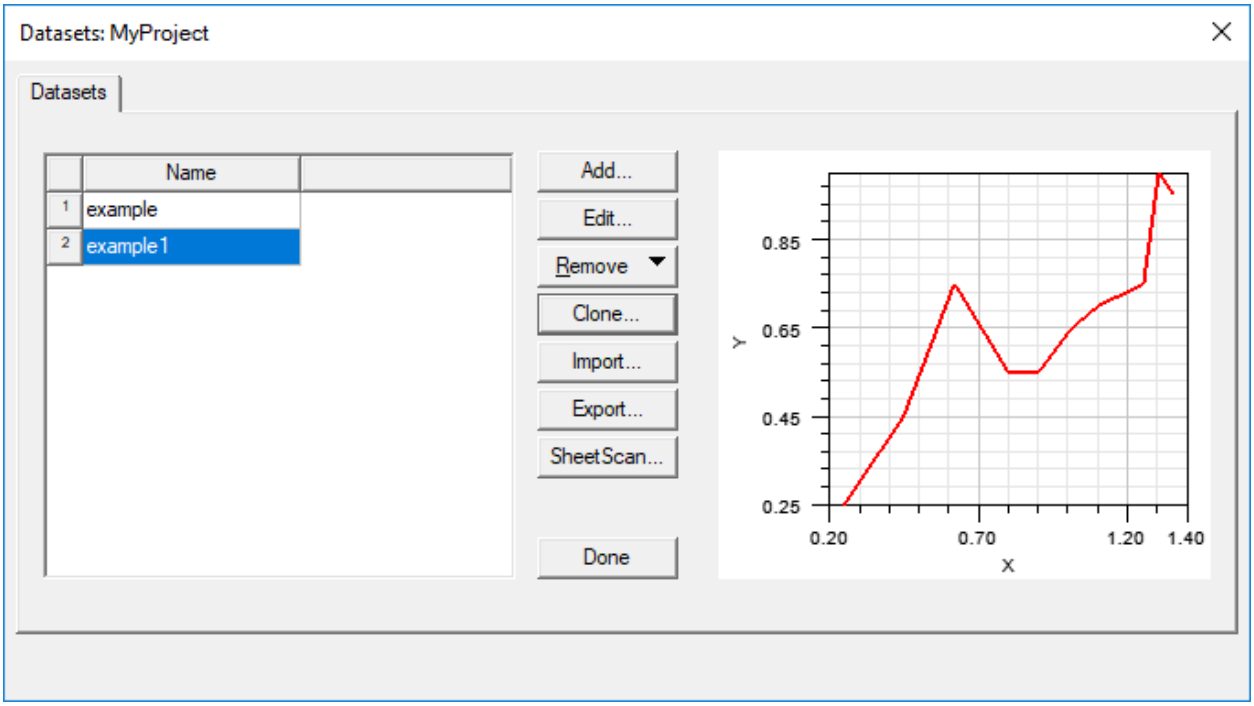
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Clone Dataset** window.

   The cloned dataset appears in the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Done** to exit the **Datasets** window.

Exporting Datasets

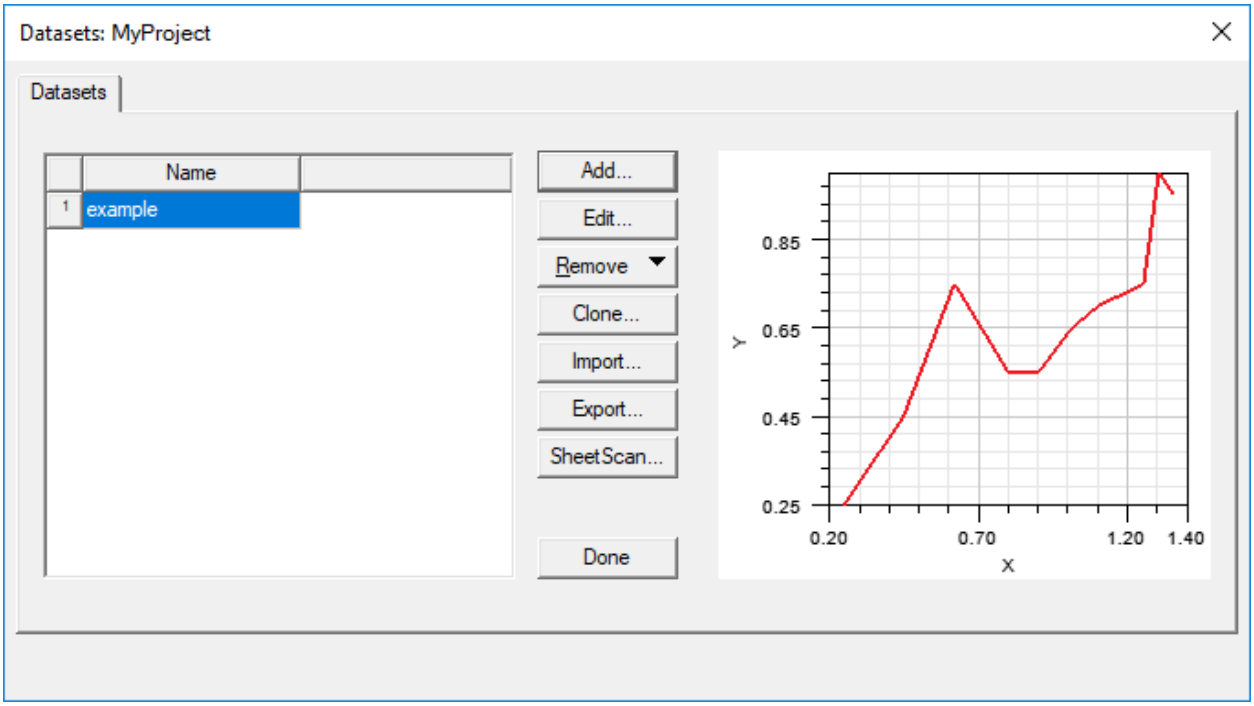
You can export datasets in **.tab** format.

To export a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Mechanical** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Export**.
3. Browse to the location where you would like to save the *.tab file.
4. Enter a name for the file.
5. Click **Save**.

   The dataset file is saved to the specified location for later import.

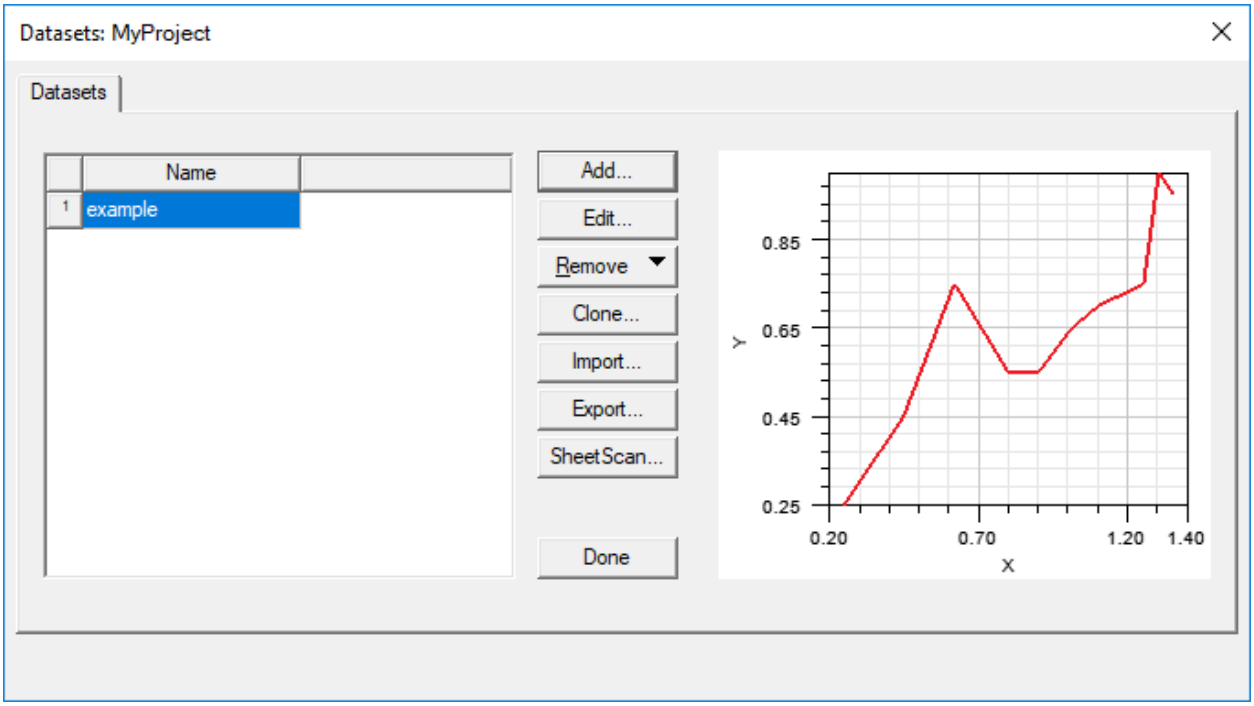
Removing Datasets

To remove a previously added dataset:

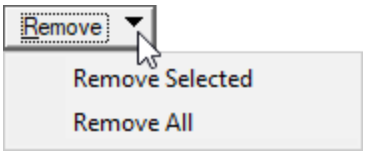
1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Mechanical** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Remove**.
3. Select either **Remove Selected** to remove the selected dataset or **Remove All** to remove all project or design-level datasets.



The selected dataset(s) are deleted from the list.

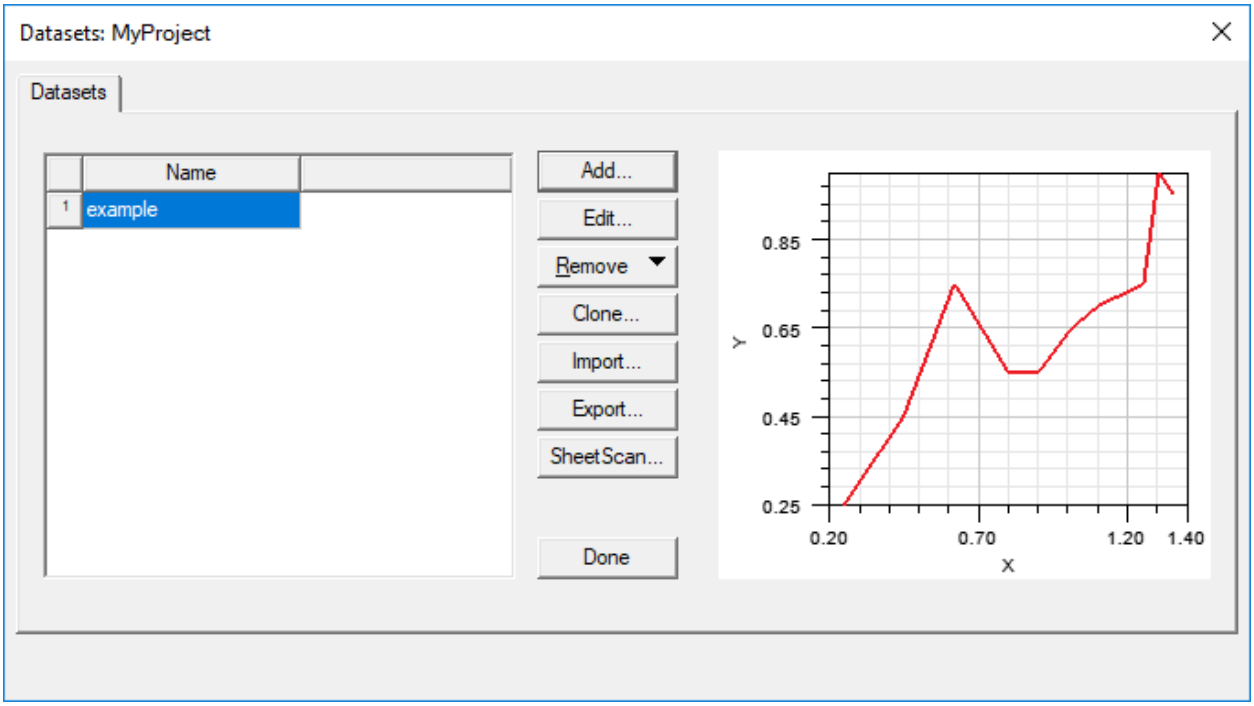
> **Note:**
>
> The **Remove All** option does not work across dataset types. That is, selecting **Remove All** from the **Project Datasets** window does not remove design-level datasets, and vice versa.

4. Click **Done** to exit the **Datasets** window.

Changing Dataset Plot Properties

From the **Datasets** window, you can change the appearance of a dataset's plot.

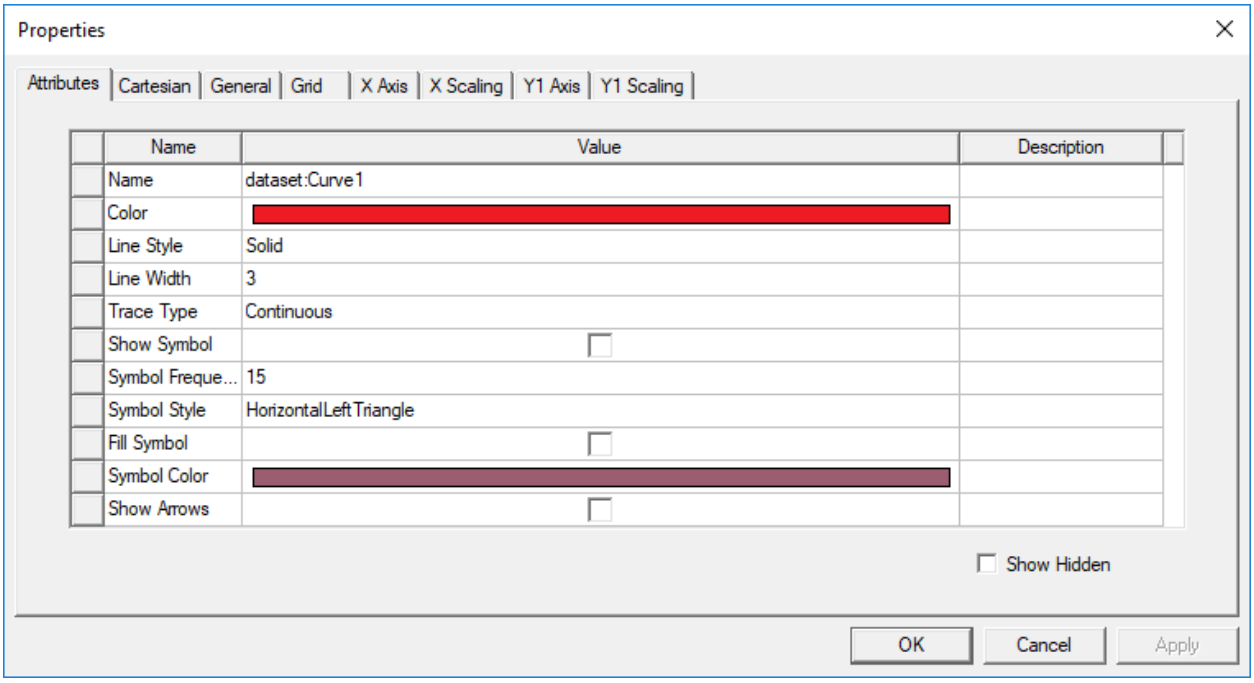
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To change a plot's appearance:

1. Double-click a plot element (for example, a line, axis, or title) to open that element's **Properties** window.

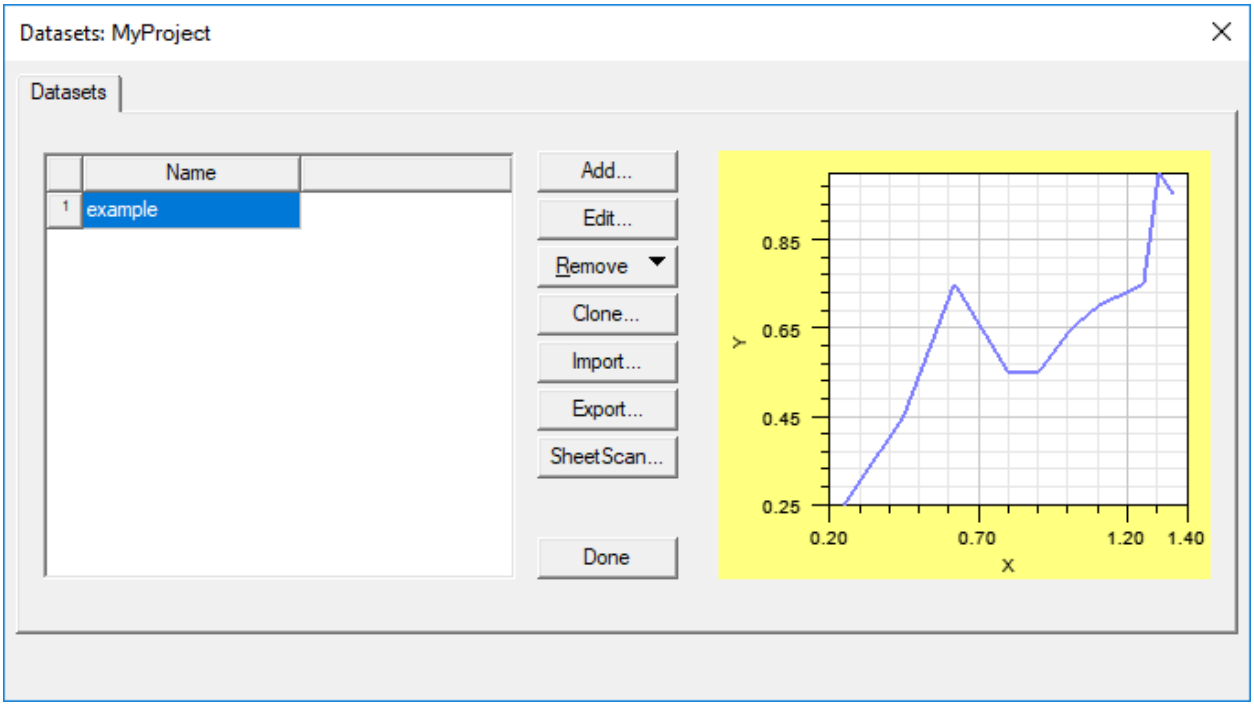
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Set the colors, fonts, and styles as applicable and as desired.
3. Click **Apply**.
4. Click **OK** to exit the **Properties** window.

   The plot now displays as specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using SheetScan

SheetScan allows you to extract characteristics data from graphics such as data sheets which have been scanned and saved in any of the following formats: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, or .html.

Access SheetScan from the **Datasets** window:

- For project-level datasets, click **Project** > **Datasets**.
- For design-level datasets, click **Mechanical** > **Design Datasets**.

In addition to importing graphic files directly, SheetScan also can be used to browse the Internet for datasheet information and transfer a snapshot of the web page to the SheetScan editor where you can map axes on the image as an overlay. You can then manually add datapoints to approximate the characteristic curve(s) on the datasheet.The sampled data can then be converted to Ansys Electronics Desktop format, and the extracted data exported to an Ansys Electronics Desktop dataset or saved to a tab-delimited file.

The process for creating a dataset using SheetScan involves four basic operations:

- Loading a datasheet into SheetScan.
- Defining a coordinate system for the imported datasheet picture.
- Defining a characteristic curve using the datasheet picture as reference.
- Exporting the characteristic curve data to a file or to a dataset.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SheetScan Menus and Settings

In the SheetScan window, various menus and toolbars allow you to access SheetScan functionality.

The top menu bar contains the following submenus:

- **File** – allows you to open, close, save, and print files.
- **Edit** – provides access to cut, copy, and paste functions.
- **View** – allows you to select which toolbars appear in the SheetScan window (see below).
- **Options** – provides access to document settings.
- **Picture** – provides options for loading a picture or HTML file into SheetScan, and for deleting a picture.

> **Note:**
>
> The **Delete** action cannot be undone. If you delete a picture from the SheetScan editor, you must reload it from the source file or Internet web page.

- **Coordinate System** – provides options for creating and editing a coordinate system.
- **Curve** – provides options that allow you to define a curve.
- **Window** – provides options for rearranging the currently open SheetScan windows (for example, overlapping and tiling).
- **Help** – accesses the Electronics Desktop online help.

SheetScan View Options

You can toggle the following toolbars via **View** > **Toolbar**:

- **Standard** – provides access to basic functions such as file Open and Save, Cut, Copy, Paste, Print, and Help.

- **Curve** – contains a drop-down menu allowing you to select a curve and tools for working with curve values.

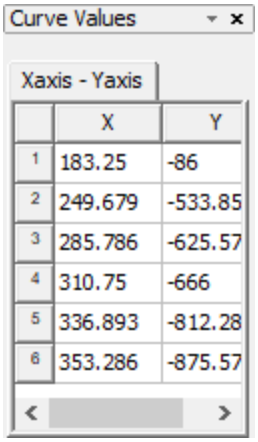
- **Zoom** – provides tools for scaling the current view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **View** menu, you can also toggle the following:

- **Status Bar** – displays, at the bottom of the screen, the current cursor coordinates.

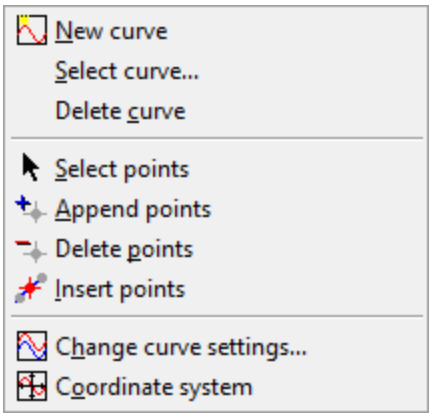
- **Curve Values** – dockable window that displays the data points used while creating a characteristic curve.

- **Grid** – displays the coordinate grid you created over the sheet.

SheetScan Right-click Menu

Once you have loaded a sheet, you can right-click in the SheetScan window to access curve and coordinate system options:



SheetScan Settings

Default settings can be changed from **Options** > **Settings**.

The **Settings** dialog box contains three tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Document** – allows you to set the width and height of the sheet created when a picture is imported. You can either enter the dimensions manually, or allow SheetScan to adapt the dimensions to the picture being loaded.
- **Axis** – allows you to set the name, units, scaling, and offset for the X and Y axis. You can also select **Monotonicity in X** to prevent adding consecutive data points whose X-values are not increasing.
- **Representation** – allows you to choose whether to connect points on the characteristic curve and to choose the connecting line color. You can also decide whether to display markers and choose marker colors.

> **Note:**
>
> You can also override the default settings on the **Axis** and **Representation** tabs for individual curves (See: Defining a Characteristic Curve in SheetScan).

Loading a Datasheet Picture into SheetScan

By default, SheetScan opens a new, blank datasheet editing window.

There are two ways to load a datasheet picture into the editor: directly, or by using the HTML Viewer.

Loading a Datasheet Picture Directly

1. Browse directly to the datasheet picture file by choosing **Picture** > **Load Picture** to open a file browser window.
2. When you have located the desired file, click **OK** to load the image into the SheetScan editor. Supported file types include: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, and .html.

Loading a Datasheet Picture Using the HTML Viewer

1. Choose **Picture** > **Internet** to open the HTML Viewer.
2. Browse the Internet for the desired datasheet.
3. Resize the HTML Viewer window and adjust its scrollbars until the desired portion of the datasheet is in view.
4. Click the **To SheetScan** button ( ) to copy the visible contents into the SheetScan editor window.

> **Note:**
>
> To hide the datasheet picture, select **View** > **Picture**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

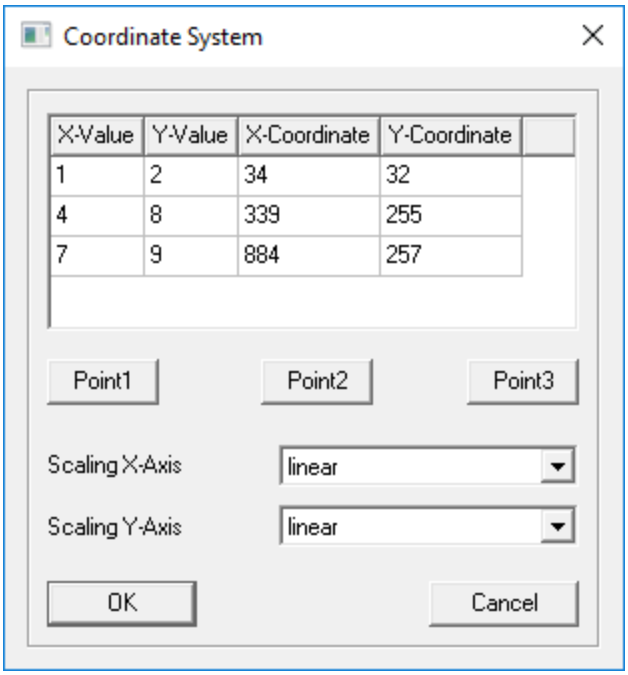
After loading a datasheet picture into the SheetScan editor, the next step is defining a coordinate system for the imported picture.

Defining a SheetScan Coordinate System

You can define a coordinate system for a graph on a datasheet picture that you have previously loaded into the SheetScan editor.

To define the coordinate system:

1. Select **Coordinate System** > **New** to open the **Coordinate System** dialog box.



   You can enter points manually or select them using a crosshair.

2. To activate the crosshair cursor, click **Point1**.

   The cursor transforms into a crosshair.

3. Position the cursor over a corner of the datasheet graph and click the left mouse button.

   The **Coordinate System** dialog box reappears, displaying the X- and Y-Coordinate values for the chosen point.

4. Enter the X-Value and Y-Value for this point. Typically, these values will correspond to the values taken from the axis scale values on the datasheet.
5. Select the desired scaling (linear, logarithmic, or decibel) for both the X and Y axes.
6. Repeat the above steps for **Point2** and **Point3**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **OK**.

   The grid is placed over the graphic.

   > **Note:**
   >
   > - You can edit the grid after placement by selecting **Coordinate System** > **Properties**, by clicking the coordinate system icon on the Curve toolbar ( ), or by right-clicking in the SheetScan editing window and selecting **Coordinate System**.
   > - You can hide the grid by selecting **View** > **Grid**.

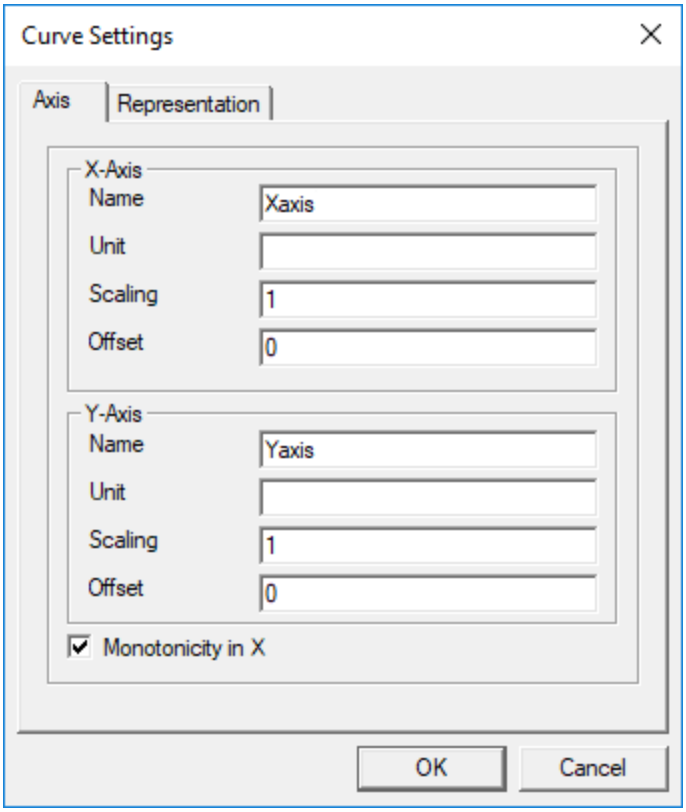
8. You can now define a Characteristic Curve.

Defining a Characteristic Curve in SheetScan

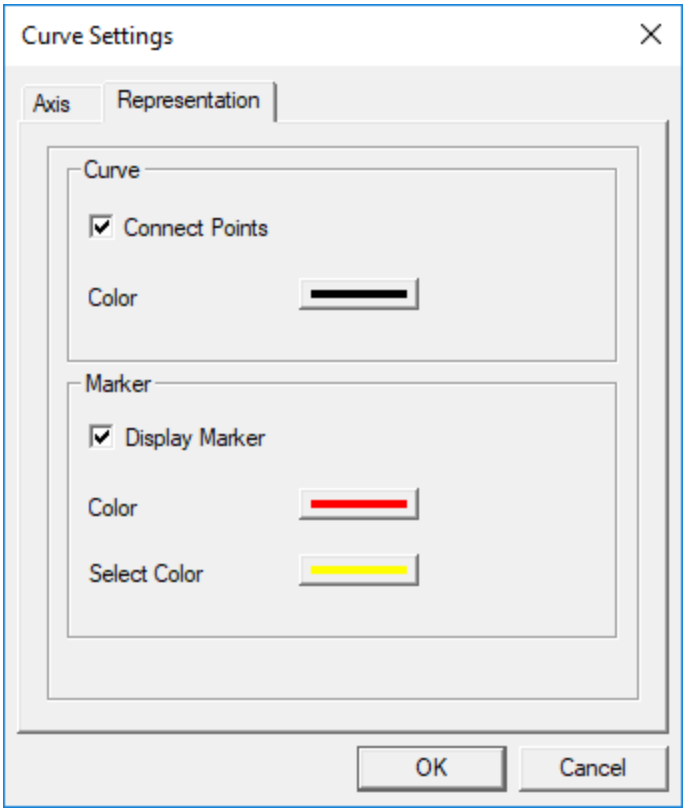
Once you have loaded a datasheet picture in the editor and have defined a coordinate system, you can define one or more characteristic curves as follows:

1. Choose **Curve** > **New**.

   The **Curve Settings** dialog box opens, containing the **Axis** and **Representation** tabs.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the fields on the **Axis** tab to define the curve's properties. Use the buttons on the **Representation** tab to change the curve's appearance.
3. Click **OK**.

   The cursor changes to a crosshair.

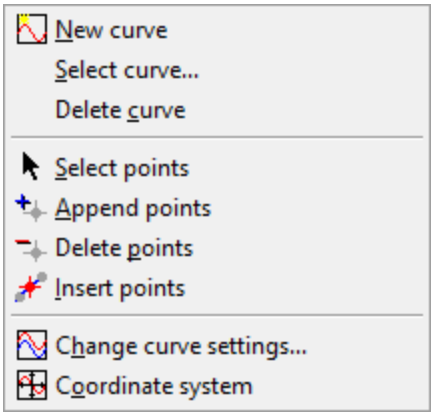
4. Click the points of the characteristic which you want to capture for the dataset. The points are connected automatically.
5. Repeat steps 1 through 4 for each additional characteristic curve you wish to define.

After characteristic curves have been defined, you can perform various operations on them. See: *Performing Operations on SheetScan Curves.*

Performing Operations on SheetScan Curves

Use the **Curve** menu (from the toolbar or by right-clicking in the sheet) to perform actions on SheetScan curves.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can perform the following actions:

- **New Curve** – allows you to define a new characteristic curve.
- **Select Curve** – opens the **Select Curve** dialog box, listing all active curves. Select the desired curve and click **OK**.
- **Delete Curve** – deletes the selected curve.

  > **Warning:**
  >
  > You cannot undo this action. If you delete a curve and its data points from the SheetScan editor, you must reconstruct it manually.

- **Select Points** – select this option, then click a point to select it. **Ctrl+click** allows you to select multiple points.
- **Append Points** – select this option, then click to add data points to the end of a curve.
- **Delete Points** – select this option, then click a data point to remove it from the curve.
- **Insert Points** – select this option, then click to insert new data points between existing data points.
- **Change Curve Settings** – opens the **Curve Settings** dialog box, where you can edit the initial curve settings.
- **Coordinate System** – opens the Coordinate System dialog box, where you can edit the coordinate system settings.

Checking for Monotonicity in X

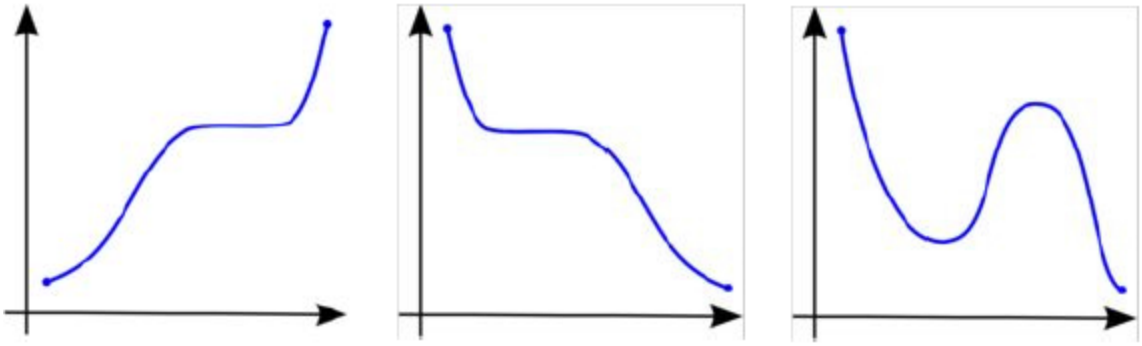
Mechanical requires that characteristics be monotonically increasing along the X-axis. In other words, successive data points must have increasing X-values, while Y-values may both increase and decrease.

To check for monotonicity in X from SheetScan:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Curve** > **Check Monotonicity**.

   If the characteristic curve is monotonically increasing in X-value, the check completes without notice. Typical examples of curves that meet monotonicity criteria are shown below.



2. If the characteristic curve is not monotonically increasing in X-value, a dialog box displays informing you that errors were found. Click **Yes** to have SheetScan automatically correct the errors.

Importing Characteristic Data into SheetScan

SheetScan supports data import from the following file types:

- Twin Builder Characteristic (*.mdx, *.mda)
- Microsoft Excel (*.xls, *.xlsx)
- Text (*.txt)
- Comma Separated Value (*.csv)
- Spice (*.out)
- Comtrade (*.cfg)
- TEK Oscilloscope (*.dat)

> **Note:**
>
> Before importing characteristic data, you must define a coordinate system.

To import characteristic curve data into SheetScan:

1. Click **File** > **Import**.
2. In the file **Open** dialog box, select the desired data file and click **Open**.
   a. Selecting an **.xls** or **.xlsx** file containing multiple sheets opens a **Table Properties** dialog box where you can choose the desired sheet from a drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

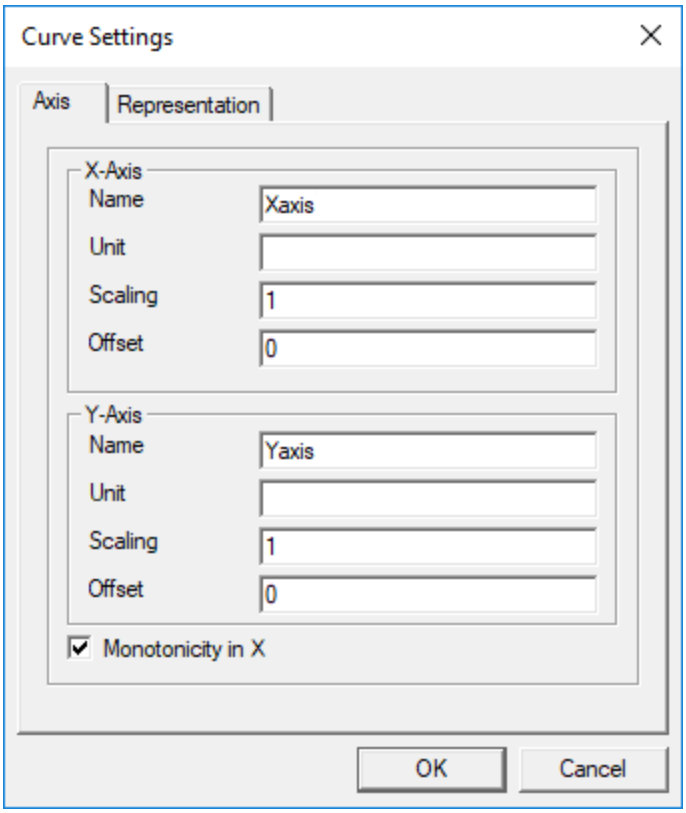
Otherwise, selecting an **.xls** or **.xlsx** file imports the data immediately into the **Add Dataset** dialog box.

> **Note:**
>
> - Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
> - The first row of data is assumed to contain column headings and is ignored.

b. Selecting a **.txt** or **.csv** file opens an **Import** dialog box in which you can specify how to settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

When satisfied with the import settings, click **OK** to import the data.

The **Curve Settings** dialog box opens.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Change **Curve Settings** as needed and click **OK** to complete the data import.

   The new characteristic curve is added to the current SheetScan sheet.

Exporting SheetScan Data

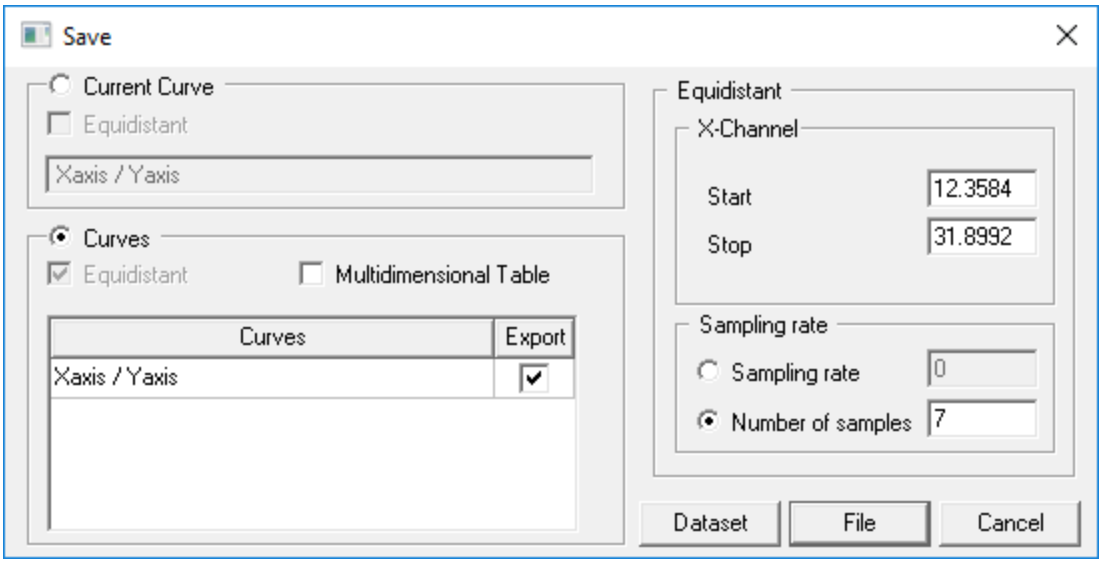
After defining a coordinate system and adding a curve, you can export SheetScan curve data directly to:

- Electronics Desktop Dataset
- Twin Builder Characteristic (*.mdx) file
- Comma Separated Value (*.csv) file
- Comtrade (*.cfg) file

To export curve data:

1. Click **File** > **Export**.

   The **Save** dialog box appears.



2. In the **Save** dialog box, choose **Current Curve** (default) to export current curve data, or **Curves** to choose the curve(s) whose data you wish to export. Choosing **Curves** reveals a list box showing all of the curves available for export.

> **Note:**
>
> The **Multidimensional Table** option is not currently supported.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose **Equidistant** if you want to set the **Start** and **Stop X-Channel** values and a **Sample Rate** or **Number of samples** for the exported dataset(s).
4. Click **Dataset** to export curve data directly to the project's dataset file, or click **File** to export as one of the supported file types.

Defining an Expression

Expressions are mathematical descriptions that typically contain Intrinsic Functions, such as sin (x), and arithmetic operators, such as +, -, *, and /, as well as defined variables. For example, you could define: x_size = 1mm, y_size = x_size + sin(x_size). Defining one variable in terms of another makes it a dependent variable. Dependent variables, though useful in many situations, cannot be the subject of optimization, sensitivity analysis, tuning, or statistical analysis.

The Constants tab of the **Project Variables** window lists all available pre-defined constants. These may not be reassigned a new value.

Numerical values may be entered in Ansys shorthand for scientific notation. For example, $5x10^7$ could be entered as **5e7**.

Valid Operators for Expressions

The operators that can be used to define an expression or function are performed in a sequence.

The following table lists valid operators and the sequence in which they are accepted (listed in decreasing precedence):

()	Parenthesis	1
!	Not	2
^ (or **)	Exponentiation (If you use "**" for exponentiation, as in previous software versions, it is automatically changed to "^".)	3
-	Unary minus	4
*****	Multiplication	5
/	Division	5
+	Addition	6
-	Subtraction	6
==	Equals	7
!=	Not equals	7
>	Greater than	7
<	Less than	7
>=	Greater than or equal to	7
<=	Less than or equal to	7
&&	Logical and	8

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

||	Logical or	8

Using Intrinsic Functions in Expressions

Mechanical recognizes a set of intrinsic trigonometric and mathematical functions that can be used to define expressions. Intrinsic function names are reserved and may not be used as variable names.

> **Note:**
>
> If units are not specified, all trigonometric functions interpret their arguments as radians. Likewise, inverse trigonometric functions' return values are in given in radians. When the argument to a trigonometric expression is a variable, the units are assumed to be radians. To have values interpreted in degrees, supply the argument with the unit name **deg**.

The following intrinsic functions may be used to define expressions:

Function	Description	Syntax
abs	Absolute value (|x|)	abs(x)
sin	Sine	sin(x)
cos	Cosine	cos(x)
tan	Tangent	tan(x)
asin	Arcsine	asin(x)
acos	Arccosine	acos(x)
atan	Arctangent (in range of -90 to 90 degrees)	atan(x)
atan2	Arctangent (in range of -180 to 180 degrees)	atan2(y,x)
asinh	Hyperbolic Arcsine	asinh(x)
atanh	Hyperbolic Arctangent	atanh(x)
sinh	Hyperbolic Sine	sinh(x)
cosh	Hyperbolic Cosine	cosh(x)
tanh	Hyperbolic Tangent	tanh(x)
even	Returns 1 if integer part of the number is even; returns 0 otherwise.	even(x)
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.	odd(x)
sgn	Sign extraction	sgn(x)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

exp	Exponential (e^x)	exp(x)
pow	Raise to power (x^y)	pow(x,y)
if	If	if(cond_exp,true_exp,false_exp)
pwl	Piecewise Linear. (**pwl** can be used with datasets for Design Variables but not for Project variables).	pwl(dataset_exp,variable)
pwl_periodic	Piecewise Linear for periodic extrapolation on x.	pwl_periodic (dataset_exp,variable)
sqrt	Square Root	sqrt(x)
ln	Natural Logarithm (The "log" function has been discontinued. If you use "log(x)" in an expression, the software automatically changes it to "ln (x)".)	ln(x)
log10	Logarithm base 10	log10(x)
int	Truncated integer function	int(x)
nint	Nearest integer	nint(x)
max	Maximum value of two parameters	max(x,y)
min	Minimum value of two parameters	min(x,y)
mod	Modulus	mod(x,y)
rem	Returns the fractional part of a decimal number such that rem (x) = x-int(x)	rem(x)
clp	Implements smooth interpolation employing weighted impact of all points of the dataset (not just the closest one). See formula below. **Note:** If used with a large 3D dataset, clp function will degrade.	clp (*datasetName* , X,Y,Z)

clp Formula

$$clp\left(Dataset,X,Y,Z\right)=\begin{cases} DatasetValue_i \text{ if } r_i\left(X,Y,Z\right) < r_{rounding} \\ \Sigma_i \frac{DatasetValue_i}{r_i^4} \Big/ \Sigma_i \frac{1}{r_i^4} \quad otherwise \end{cases}$$

$$r_i\left(X,Y,Z\right)=\sqrt{\left(X-DatasetX_i\right)^2+\left(Y-DatasetY_i\right)^2+\left(Z-DatasetZ_i\right)^2}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Piecewise Linear Functions in Expressions

The following piecewise linear intrinsic functions are accepted in expressions:

- **`pwl (dataset_expression, variable)`** – interpolates along the X-axis and returns a corresponding Y value.
- **`pwlx (dataset_expression, variable)`** – interpolates along the X-axis and returns a corresponding Y value.
- **`pwl_periodic (dataset_expression, variable)`** – also interpolates along the X-axis, but periodically.

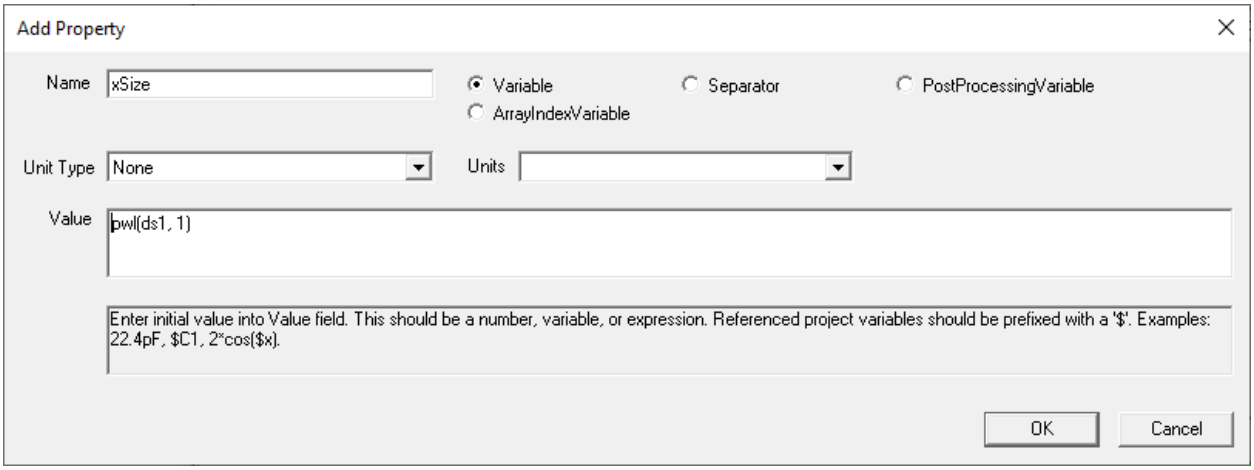
You can use **pwl** in an expression that uses datasets for such things as a frequency dependent material property (See: Adding Datasets).

For example, you can specify BulkConductivity as:

```
pwlx($ds1, Freq)
```

You can create a design variable representing a dimension `xSize` as `pwl(ds1, 1)` where `ds1` is a dataset.

Doing so looks like this:

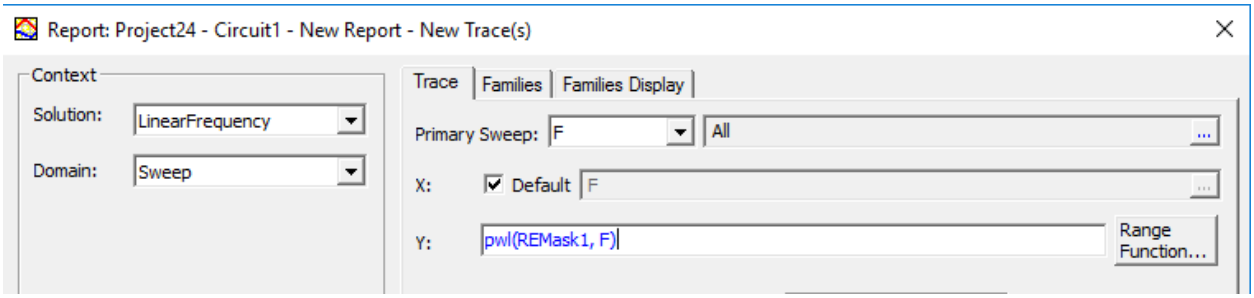


After the dataset is configured, the **Properties** window shows the new variable.

To use datasets in reports, create a report using **pwl(dataset_expression,**`variable`**)** where `variable` is the primary sweep in the report.

For example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> pwl can be used with datasets for Design Variables but not for Project variables.

Using Dataset Expressions

A dataset is a collection of data. It can take the following form:

$ds1((x0,y0), ..., (xn,yn))

Once created, a dataset (such as $ds1) may be used as the first parameter to piecewise linear (**pwl**, **pwlx**, and **pwl_periodic**) functions, and may also be assigned to variables, in which case the variable may be used as the second parameter to **pwl**, **pwlx**, and **pwl_periodic** functions.

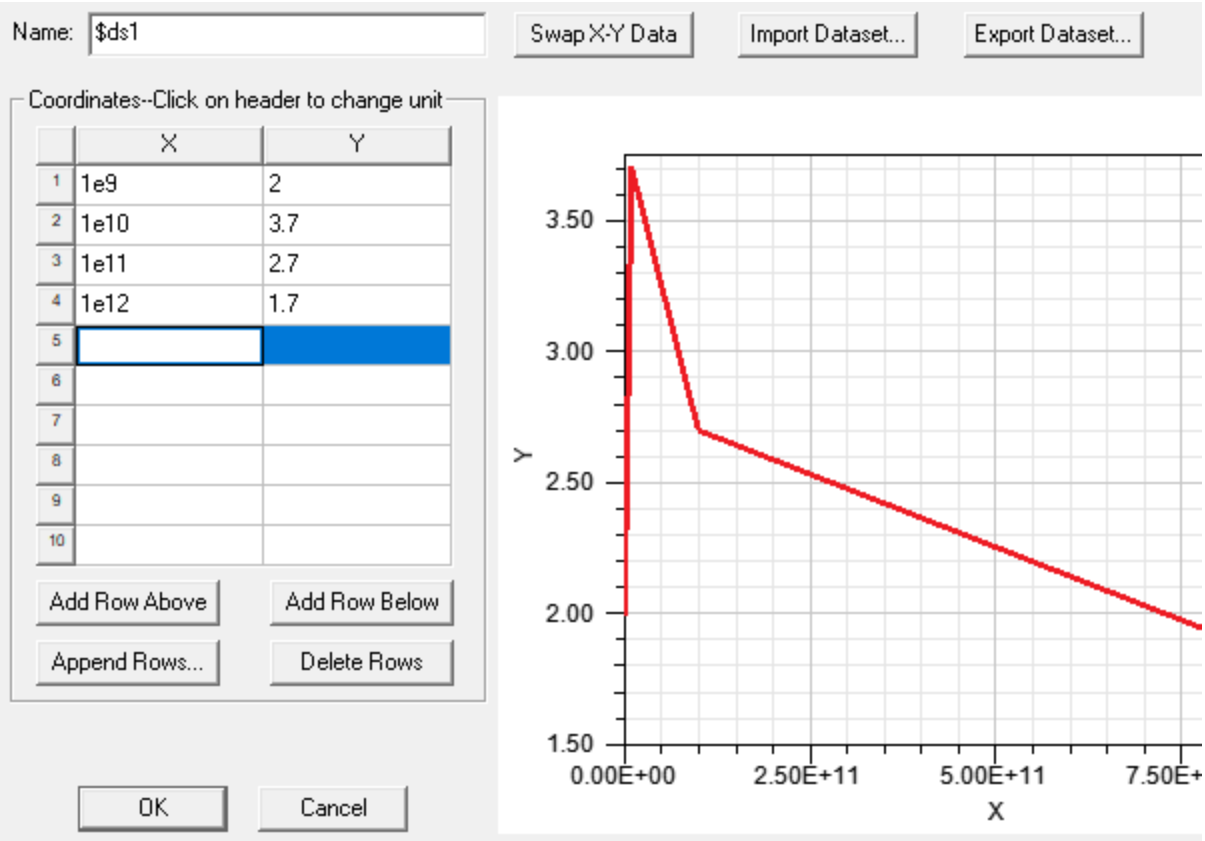
You can generate a dataset using a series of points in a plot on the **Datasets** dialog box (See: Adding Datasets). Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot. This curve, which best fits the segmented plot, consists of the co-ordinates used in the creation of the dataset. The dataset may then be used in the piecewise linear intrinsic functions.

> **Note:**
>
> The following is an example showing how to make a material property frequency dependent using a dataset as the first parameter to a **pwl** function. The values used are arbitrary.

1. For a project-level dataset, click **Project** > **Datasets**. For a design-level dataset, click **Mechanical** > **Design Datasets**.
2. Click **Add**.
3. Set the **Name** field and **Coordinates** as desired and click **OK**.

   The dataset is created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

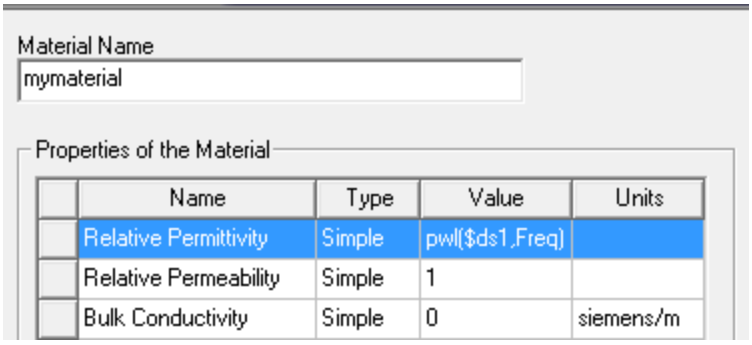


Note:

By default, the dollar sign ($) is assigned to a project dataset even if you do not use one while naming it.

4. Go to **Tools** > **Edit Libraries** > **Materials** to open the **Edit Libraries** dialog box.
5. Click **Add Material**.
6. Type in the piecewise linear function and use the dataset $ds1, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Click **OK**.

Handling Delta Temperature Units in Expressions

When Temperature and Delta Temperature quantities are used as operands to plus or minus operations in an expression, they are handled specially. The biggest difference is an automatic unit change of the resulting values based on the units of the operands.

- When two Temperature quantities are being subtracted (either the first or both operands is/are temperature quantities), the result value has Delta Temperature units.
- When a Delta Temperature quantity is added or subtracted from a Temperature quantity, the result value has Temperature units.
- Adding or subtracting two Delta Temperature quantities results in a quantity with Delta Temperature units.
- Subtracting two temperature quantities is an observable behavior change when compared to earlier releases of Ansys Electronics Desktop.

Temperature Units

Celsius Family

- cel, delta_cel (Legacy name for delta_cel, celdiff continues to be supported)

Kelvin Family

- mkel, delta_mkel (milli Kelvin) (Legacy name for delta_mkel, mkeldiff continues to be supported)
- ckel, delta_ckel (centi Kelvin)
- dkel, delta_dkel (deci Kelvin)
- kel, delta_kel (Legacy name for delta_kel, keldiff continues to be supported)

Farenheit Family

- fah, delta_fah

Temperature-related Use Cases

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The high-level legal use cases (in unit types) are:

- Temperature - Temperature = DeltaTemperature
- Temperature +/- DeltaTemperature = Temperature
- DeltaTemperature +/- DeltaTemperature = DeltaTemperature

The minor variations in these use cases specify how the actual units (Celsius is a unit of the Temperature unit type) are handled.

Temperature - Temperature

If they have the same unit, then the Delta Temperature unit will be a matching one.

For example:

- 10 cel - 5 cel = 5 delta_cel
- 100 kel - 90kel = 10 delta_kel
- 10 fah - 1 fah = 9 delta_fah

If they have different units, they are converted to the default units for temperature difference.

For example:

- 10cel - 1 kel = 9 delta_kel

Temperature +/- DeltaTemperature

The resulting temperature quantity will retain the units of the Temperature quantity (first operand)

DeltaTemperature +/- DeltaTemperature

- If they have the same unit, the resulting Delta Temperature quantity will retain that.
- If they have different units, the resulting Delta Temperature quantity will have the default Delta Temperature units
- 5 delta_cel + 10 delta_cel = 15 delta_cel
- 10 delta_cel + 1 delta_kel = 11 delta_kel

All other use cases in a plus or minus arithmetic operation are physically meaningless and revert to previous bahavior, where Electronics Desktop converts quantities to their SI values and then operates on the plain numbers.

- Temperature + Temperature
- Temperature +/- non-Temperature/non-Delta Temperature
- Delta Temperature +/- non-Temperature/non-Delta Temperature
- non-Temperature/non-Delta Temperature +/- Delta Temperature
- non-Temperature/non-Delta Temperature +/- Temperature

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Mathematical Functions

A mathematical function is an expression that references another defined variable. A function's definition can include both expressions and variables.

The following mathematical functions may be used to define expressions:

Basic Functions	**/**, **+**, **-**, *, mod (modulus), ** (exponentiation), **-** (Unary minus), **==** (equals), **!** (not), **!=** (not equals), **>** (greater than), **<** (less than), **>=** (greater than equals), **<=** (less than equals), **&&** (logical and), **||** (logical or)
Intrinsic Functions	**if**, **abs**, **exp**, **pow**, **ln** (natural log), **log10** (log to the base 10), **sqrt**
Trigonometric Expressions	**sin**, **cos**, **tan**, **asin**, **acos**, **atan**, **sinh**, **cosh**, **tanh**

The predefined variables X, Y, Z, Phi, Theta, R, and Rho must be entered as such. X, Y, and Z are the rectangular (Cartesian) coordinates. Phi, Theta, and Rho are the spherical coordinates. R is the cylindrical radius, and Rho is the spherical radius.

If you do not specify units, all trigonometric expressions expect their arguments to be in radians, and the inverse trigonometric functions' return values are in radians. If you want to use degrees, you must supply the unit name **deg**. When the argument to a trigonometric expression is a variable, the units are assumed to be radians. These function names are reserved and may not be used as variable names.

As far as expression evaluation is concerned: units are conversion factors (that is, from the given unit to SI). Note also that the evaluated value of an expression) is always interpreted as in SI units.

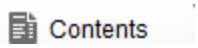
Meshing Quick Links

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

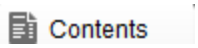
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that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

Modeling Quick Links

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

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

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

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shortcut Keys

Various commands in the Ansys Electronics Desktop software may be accessed through keyboard shortcuts or “hot keys”. These keystroke combinations allow you to bypass the menu system and directly execute commands. In addition, you may use the **Keyboard Shortcuts** dialog box to customize the hotkeys associated with various Ansys Electronics Desktop menu commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Mechanical Licensing Requirements

Mechanical–Modal and Mechanical–Structural solutions have a unique licensing requirement, different from other Ansys Electronics Desktop (AEDT) products:

- The **License for Structural/Modal solution type** that you specify must match the type of license purchased by you or your organization. Instructions for doing so follow.

Choose Structural/Modal License

There are three levels of PPE license available for purchase:

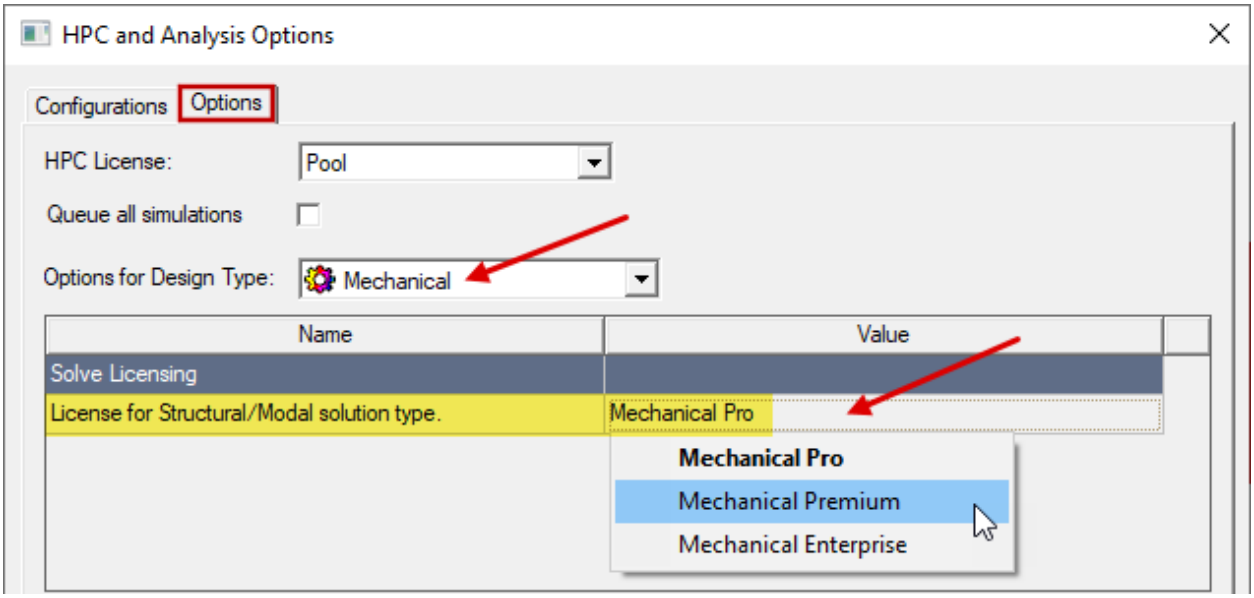
- Mechanical Pro (default selection)
- Mechanical Premium
- Mechanical Enterprise

> **Important:**
>
> Though all three of these licenses support the full scope of modal and structural solutions available for Mechanical designs in AEDT, you still **must** choose the specific type of license that has been purchased for your installation. Otherwise, structural and model solutions will fail to run.

Use the following procedure to specify the appropriate Structural/Modal license:

1. Use one of the following two methods of accessing the HPC and Analysis Options dialog box:
   - On the **Simulation** ribbon tab, click **HPC Options**.
   - From the menu bar, click **Tools** > **Options** > **HPC and Analysis Options**.
2. Select the **Options** tab.
3. Choose **Mechanical** from the **Options for Design Type** drop-down menu.
4. Click the **Value** cell to the right of **License for Structural/Modal solution type** and select the license type that matches the type purchased for your use:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Supported Geometry

A Mechanical design can contain any type of 3D geometry that you can draw in the Modeler window or import from a CAD source (solids, sheets, and lines). However, the solver will ignore line and sheet objects. That is, they will not participate in the solution.

For Thermal solutions only, shell objects are also supported by the solver. These objects are sheets with the optional **Shell Element** attribute selected in the object properties. Sheets without this option selected will be ignored in Thermal solutions in the same way that they are for Modal and Structural solutions.

Imported geometry may contain sheet objects. For example, it is not uncommon for HFSS and Maxwell 3D models to include both sheet and solid objects. Again, the solid objects will be solved and the sheet objects ignored.

Model Objects

The following types of geometry are supported by the Mechanical solvers and participate in the solution:

- Modal and Structural Solutions:
  - Solid objects
- Thermal Solutions:
  - Solid objects
  - Shell objects

Non-Model Objects

Despite line and sheets being ignored by the solver, they can be included in models for a couple of useful purposes, as follows:

- **Lines:** You can use lines to specify a path along which solid element results can be plotted or tabulated in reports. This feature is covered for each solution type in the *Results* sections of the Mechanical help.
- **Sheets:** You can use sheet objects as tools for imprinting onto the face of a solid object, splitting the original face into smaller subfaces. This technique is useful when you need to assign a face-based boundary or excitation to only a portion of an object face. For more information, see the topic, Assigning a Boundary or Excitation to a Portion of a Face. This technique is applicable to all Mechanical solution types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- You cannot apply boundaries or excitations to sheet objects in any Mechanical solution type. If you attempt to do so, you will encounter an error message stating that the boundary or excitation can only be applied to sheets modeled as shell elements. However, the *Shell Element* attribute is not available for Modal or Structural solutions; it is only available for Thermal solutions.
- You must use the *Tau* mesher when you have performed imprinting and have **not** kept (that is, *Cloned*) the sheet objects used as the *Tool Parts*. The *Classic* mesher will simplify coplanar and fully tangential subfaces by merging them back into the original, larger face (unless there is an adjacent sheet or solid body at the imprinted subface). The model geometry will not be altered, but the Classic mesher will ignore the subfaces when generating the solid mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 - Modal Solutions

Use the *Modal* solution type to determine vibrational natural frequencies of structures, along with the shapes of the various vibration modes. It is often desirable to avoid exciting structures at frequencies at or near their natural frequencies, due to resulting resonance. Where certain excitation frequencies are unavoidable, you can alter the design of the structure (mass, stiffness, support locations) to change the natural frequencies. In other cases, you might provide a means of damping to limit the severity of unavoidable resonant frequencies.

The results of a modal analysis include the vibrational natural frequencies and mass participation factors, which reflect the amount of a structure involved in a particular vibration mode. You can sum these factors to determine the cumulative mass participation from multiple vibration modes. Displacements are also calculated, but they are not scaled to any specific excitation (such as ground displacement or acceleration). Rather, the displacement magnitudes are arbitrary in nature, and the raw numeric values are meaningless. However, the relative displacement values (between one area of the structure and another area) provide a visual indication of the vibration mode shape. These deformed shapes typically must be scaled, as the arbitrary displacements can be orders of magnitude greater than the size of the structure or, in some cases, too small to be clearly seen without exaggeration.

> **Important:**
>
> Product licensing for Structural solutions differs from other design and solution types within the Ansys Electronics Desktop application. See the Mechanical Licensing Requirements topic for details.

This section contains information regarding the material assignment, setup options, and results that are available for a *Mechanical* design, *Modal* solution type. Specifically, the following topics are included:

- Assigning Materials
- Boundaries
- Solution Setup
- Results

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For a valid modal solution, the following model characteristics are required:

- The *Mass Density, Young's Modulus,* and *Poisson's Ratio* must be defined for the selected materials.

Regarding Boundaries and Contact:

- The model does not have to be statically stable. That is, the model can be free to experience rigid-body motion in any or all six directions (X, Y, and Z translations and X, Y, and Z rotations). Rigid-body modes are supported by the solvers. When a body is free to move in any X, Y, and direction, the first six calculated modes will be the rigid-body modes, which do not involve distortion of the geometry but only rigid-body translation and rotation. For two unconstrained bodies, there would be twelve rigid-body vibration modes, eighteen rigid-body modes for three unconstrained objects, and so on. After all possible rigid-body modes are accounted for, the remaining modes will involve deformation of the geometry (bending, stretching/compression, and torsion).
- Bonding occurs between parts with coplanar faces and also occurs by default between parts that overlap each other. Depending on the options selected in the *Mechanical Design Settings*, implicit subtraction of material may occur where solids overlap (eliminating the overlap from the solution). Alternatively, an error condition may be reported during model validation. Parts that do not touch each other are separate and do not interact.

Regarding Damping:

- All modal solutions in the current release of Ansys Electronics Desktop are undamped. Damping is not currently supported.

Assigning Modal Materials

When choosing a material for a modal analysis, you will see a very different set of materials and material properties compared to an electronic circuit or electromagnetics analysis. Instead of properties such as Electrical Conductivity and Relative Permittivity, the physical properties of interest for modal analyses are the material's Young's Modulus, Mass Density, Poisson's Ratio, and Shear Modulus. Only materials with the first three of these four properties defined will appear in the *Select Definition* dialog box, where you choose the default material or assign an object material.

For isotropic materials, the *Shear Modulus* (G) is calculated from the Young's Modulus (E) and Poisson's Ratio (ν), according to the following equation:

$$G = E / [2 * (1 + \nu)]$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, the shear modulus is listed as *Undefined* for the library materials, which are isotropic solids. You can specify shear moduli in three directions for *anisotropic* solid materials.

Anisotropic Properties

Materials for modal solutions can have simple isotropic properties (that is, behaving quantitatively the same way in every direction), or anisotropic properties (exhibiting different behavior in three orthogonal directions). Specifically, the Young's Modulus, Poisson's Ratio, and Shear Modulus can vary in each of the three orthogonal directions. These directions are defined by the **Orientation** property of each object that uses the material (indicating a global or user-defined coordinate system).

All predefined library materials are of the simple (isotropic) type. You must define your own custom material to use anisotropic properties.

See the Defining Anisotropic Tensors topic for more information.

Default Material and Applicable Libraries

The default material for modal analyses is *steel_1008*, which is a standard AISI low-carbon grade of steel. The following libraries contain materials suitable for Modal solutions:

- *[sys] GRANTA Materials Data for Simulation*
- *[sys] GRANTA Producers Materials Data for Simulation*
- *[sys] Materials*
- *[sys] Vacuumschmelze*

Non-Applicable Materials

Fluid materials are not applicable to modal analyses and are filtered from the libraries. Additionally, Modal solutions do not currently support anisotropic or temperature-dependent material properties.

Material Selection

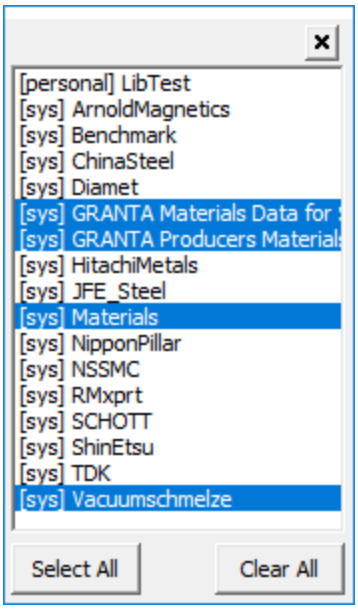
To set the default material for all objects you subsequently create:

1. At the far right end of the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.

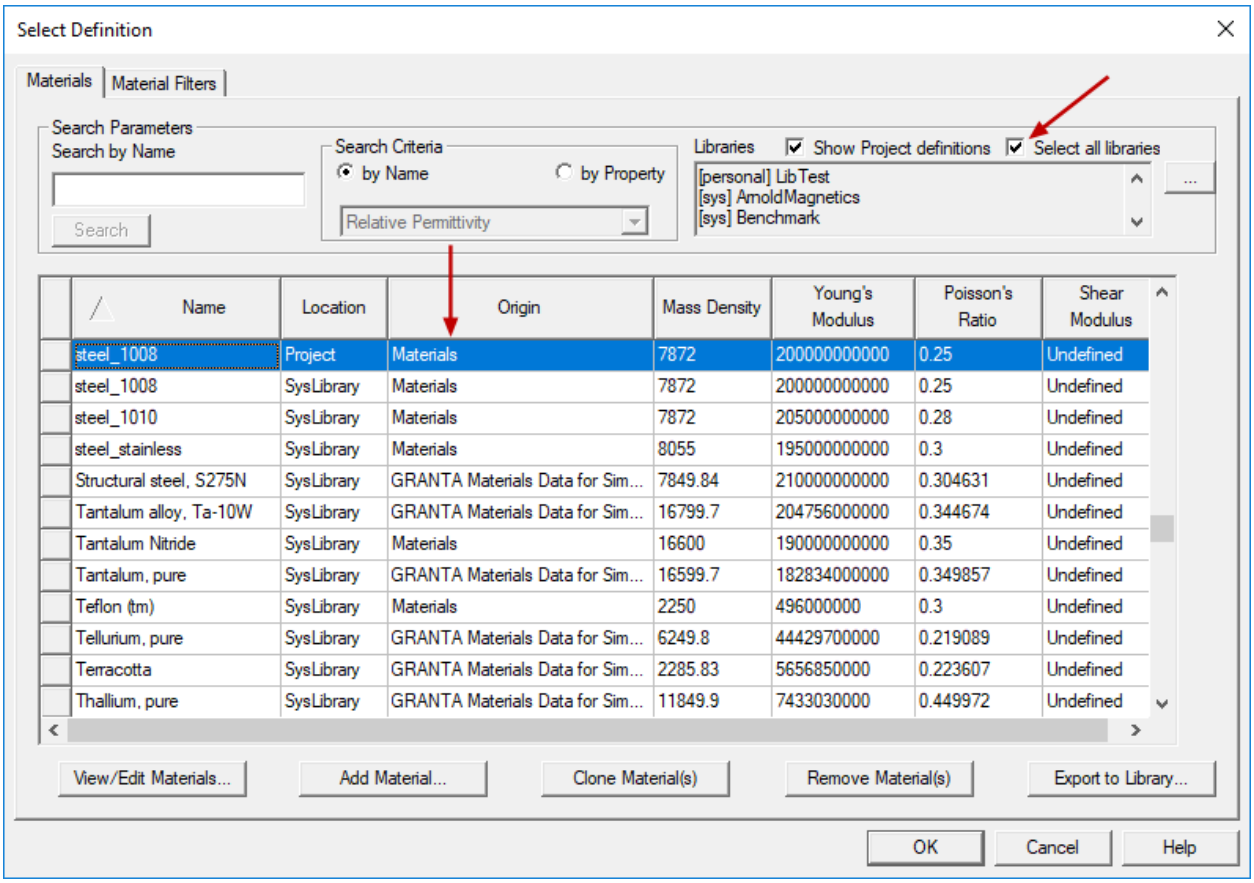
   The *Select Definition* dialog box appears.

2. In the *Libraries* list, ensure that one or more of the previously listed suitable libraries are selected, or a user library with mechanical material properties. Click the ellipsis button (**...**) to the right of the *Libraries* list to access a pop-up dialog box in which you can select which libraries to include in the *Materials* list:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Alternatively, select the **Select all libraries** option:

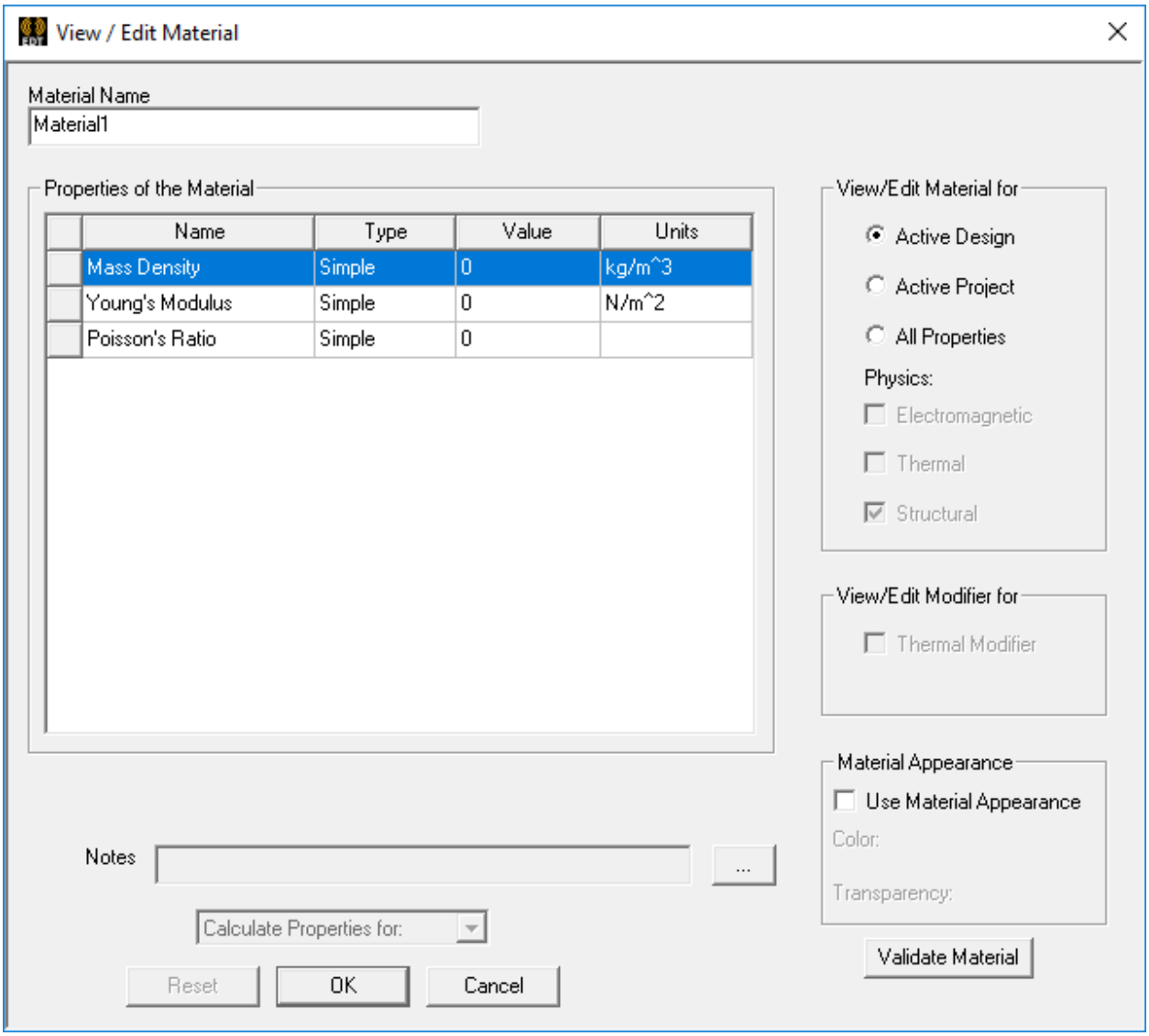


This option is selected by default for new projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Optionally, type a portion of the material name in the **Search by Name** text box. The list will automatically scroll as needed and the first material matching the search string will be selected. If you scroll the selected material off screen, click **Search** to quickly bring the searched material back into view.
4. Select the desired material, if present.
5. Optionally, to define your own material properties, click **Add Material**.

   The *View / Edit Material* dialog box appears:



6. Optionally, to specify anisotropic properties for the **Young's Modulus, Poisson's Ratio,** or **Shear Modulus** (when properties by direction), select **Anisotropic** from the **Type** drop-down menu. Otherwise, the *Simple* option is for isotropic properties (when the properties are the same in every direction).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If editing properties, or defining a new material, specify the desired **Material Name, Mass Density, Young's Modulus,** and **Poisson's Ratio**. The following material validation rules apply:
   - For simple (isotropic) materials:
     - Mass Density (ρ) must be greater than zero: ($\rho > 0$)
     - Young's Modulus (E) must be greater than zero: ($E > 0$)
     - Poisson's Ratio (v) must be in the range: ($-1 < v < 0.5$, non-inclusively)
   - For anisotropic materials:
     - Mass Density (ρ) must be greater than zero: ($\rho > 0$)
     - Young's Modulus (E) must be greater than zero in all three directions: [for E, T(1,1) > 0, T(2,2) > 0, and T(3,3) > 0]

       For *Young's Modulus*, T(1,1), T(2,2) and T(3,3) respectively correspond to E_x, E_y, and E_z (the moduli in the three axis directions, whether the coordinate system is global or user-defined).

     - Shear Modulus (G) must be greater than zero in all three directions: [for G, T(1,2) > 0, T(1,3) > 0, and T(2,3) > 0]

       For *Shear Modulus*, T(1,2), T(1,3) and T(2,3) respectively correspond to G_{xy}, G_{xz}, and G_{yz} (the moduli based on the three axis directions, whether the coordinate system is global or user-defined).

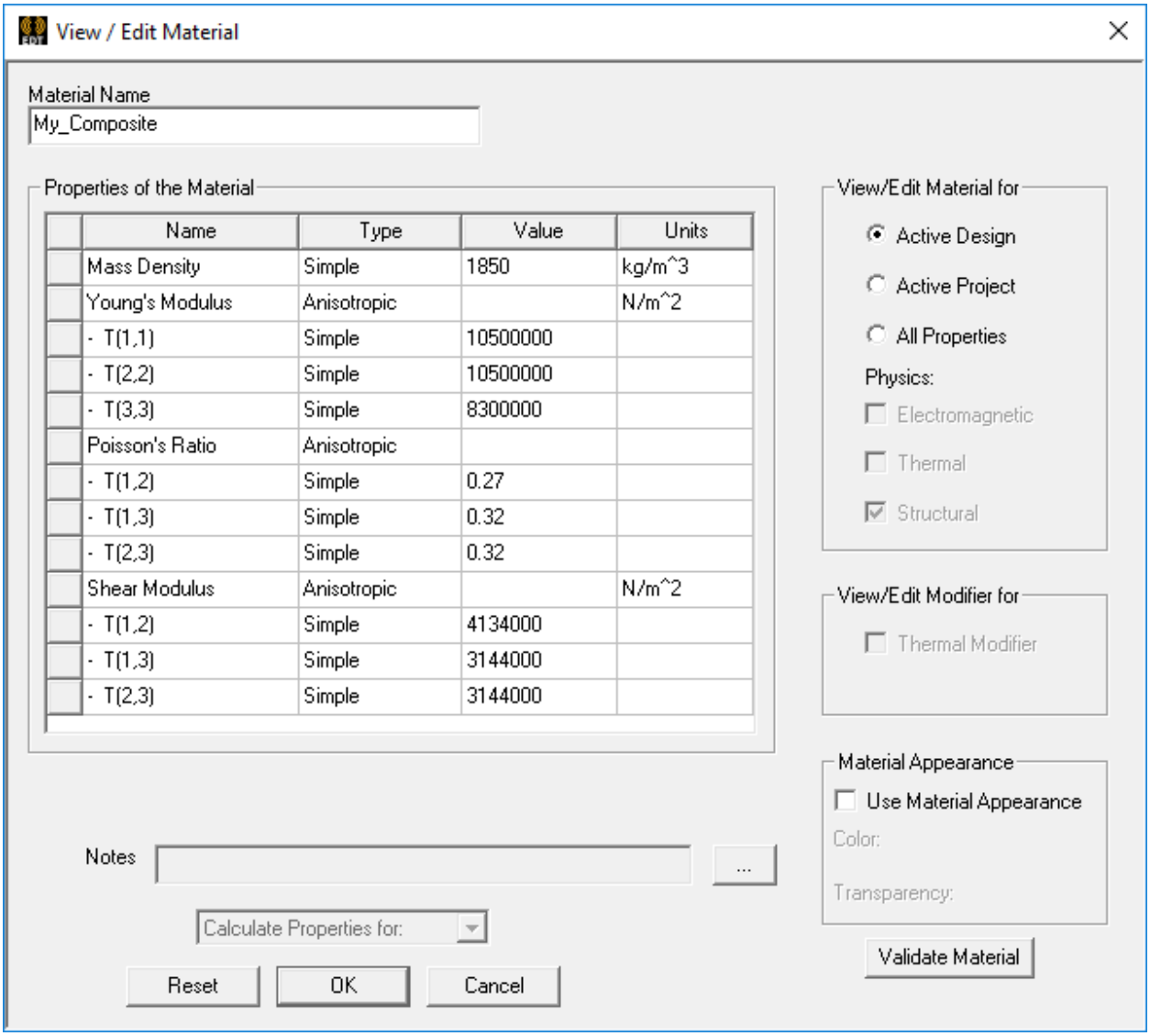
     - Material must be positive definite as defined by the following function of Young's Modulus (E) and Poisson's Ratio (v):

       $(1 - v_{12}^2 * E_2/E_1 - v_{23}^2 * E_3/E_2 - 2 * v_{12} * v_{23} * v_{13} * E_3/E_1) > 0$

       For *Poisson's Ratio*, T(1,2), T(1,3) and T(2,3) respectively correspond to v_{xy}, v_{xz}, and v_{yz} (the ratios based on the three axis directions, whether the coordinate system is global or user-defined).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is an example of the properties for an anisotropic material:



8. Optionally, select the **Use Material Appearance** option and specify the desired **Color** and **Transparency** attributes.
9. Optionally, click **Validate Material** to check for errors in the specified properties.

   If the properties pass the validation check, a green check mark appears to the right of the *Validate Material* button. Otherwise, a red X appears, and a pop-up message provides the error details.
10. Click **OK** to close the *View / Edit Material* dialog box.
11. Click **OK** to close the *Select Definition* dialog box.

To assign a different material to a selected solid object:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object to which you want to assign a material.
2. Use one of the following five methods of accessing the *Select Definition* dialog box:
   - On the **Draw** ribbon tab, click **Material**.
   - Using the menu bar, click **Modeler** > **Assign Material**.
   - Right-click in the **Modeler** window and choose **Assign Material** from the shortcut menu.
   - In the **Attribute** tab of the docked *Properties* window, choose **Edit** from the **Material** *Value* drop-down menu.
   - If the *Properties* dialog box appears automatically when you create an object, select the **Attribute** tab. Then, choose **Edit** from the **Material** *Value* drop-down menu.
3. Complete steps 2 through 6 of the previous procedure.

To add, remove, or edit materials:

1. Use one of the following two methods of accessing the *Edit Libraries* dialog box:
   - Using the menu bar, click **Tools** > **Edit Libraries** > **Materials**.
   - Under *Definitions* in the Project Manager, right-click **Materials** and choose **Edit Library** from the shortcut menu.
2. Select the desired material library from the *Libraries* list, or select **Show all libraries**.
3. Select an existing material or click **Add Material** to define a new one.
4. Optionally, you can click the following buttons once you've selected one or more existing materials:
   - **View / Edit Materials:** To change the name or properties.

     > **Note:**
     >
     > When you change the properties of a *SysLibrary* material, a new *Project* library material is created.

   - **Clone Material(s):** Make copies of the selected materials as a starting point for defining new ones.
   - **Remove Material(s):** Remove the selected materials from the library.

     > **Note:**
     >
     > You cannot remove predefined *SysLibrary* materials. You can only delete materials that you've added to a the project library or a user library (and only if the material is not being used in the current design).

5. After making the desired changes, click **OK** to close the *View / Edit Material* and/or *Edit Library* dialog boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundaries (Modal)

In modal analyses, boundaries are used to represent areas where the model is supported and therefore constrained against motion in one or more directions. The words "support" and "constraint" are frequently used interchangeably.

The *Mechanical – Modal* solution type implemented within Ansys Electronics Desktop can determine rigid-body vibration modes. In a rigid-body mode, the shapes of the objects comprising the model are not distorted as they vibrate. Instead, the body or bodies simply rotate or translate to a different location in space. There will be six rigid body modes for each physically separate body or group of connected bodies (assuming each is fully unconstrained).

A partially constrained body will be forced to distort rather than experiencing rigid-body motion in the directions that are constrained. The number of rigid-body modes will therefore be less than six for each physically separate body that is partially constrained.

For most real world conditions, a model will be statically stable (that is, not able to freely translate in any direction or rotate about any axis because of applied boundaries). In such cases, all vibration modes will involve distortion of the object shape (bending, twisting, or stretching and compressing).

The following three types of supports are available in Mechanical – Modal analyses. Click the links for more information on each type:

- **Fixed Support**
- **Frictionless Support**
- **Cylindrical Support**

> **Note:**
>
> Solid elements only have translational degrees of freedom (DOF). Individual nodes along a constrained face are prevented from translating in the X, Y, or Z direction but are free to rotate. However, applying a support to a face also indirectly prevents rotation in one or more directions. The reason is that the face cannot rotate without its nodes translating in the X, Y, and/or Z direction.

General Usage Notes Common to all Boundaries

- You can include different boundary types on different faces to best represent the behavior of the structure being modeled. For example, you could have a fixed support on one face, cylindrical supports on other faces (with various radial, tangential, or axial fixity), and a frictionless support on yet another face. The goal is to support the model without overconstraining it, which would prevent the discovery of vibration modes that will actually

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

occur. Additionally, overconstraint would stiffen the model and erroneously increase the calculated natural frequencies.

- While you are permitted to combine two or more boundary types on the same face, there are currently no supports for which it would make sense to do so.
- Any of the available boundary types can be included in the definition of a 3D component.

Solution Setup (Modal)

You must add at least one solution setup to each Mechanical design. The solution setup for a modal analysis includes the following settings:

- **Name:** The default name is Setup*x*, where *x* is a numeric value incremented for each solution setup that you add to the design. You can customize the name of each setup as desired.
- **Enabled:** This option is selected by default. Deselect the option to prevent the setup from being solved when you click the *Analyze All* command.
- **Solver type:** Specify the solver to use, or let the program choose the solver. The choices are:
  - **Program Controlled:** This is the default setting, which is recommended for most cases. The application selects the optimal solver.
  - **Direct**
  - **Iterative**
  - **Subspace**
  - **Supermode**

  Note:

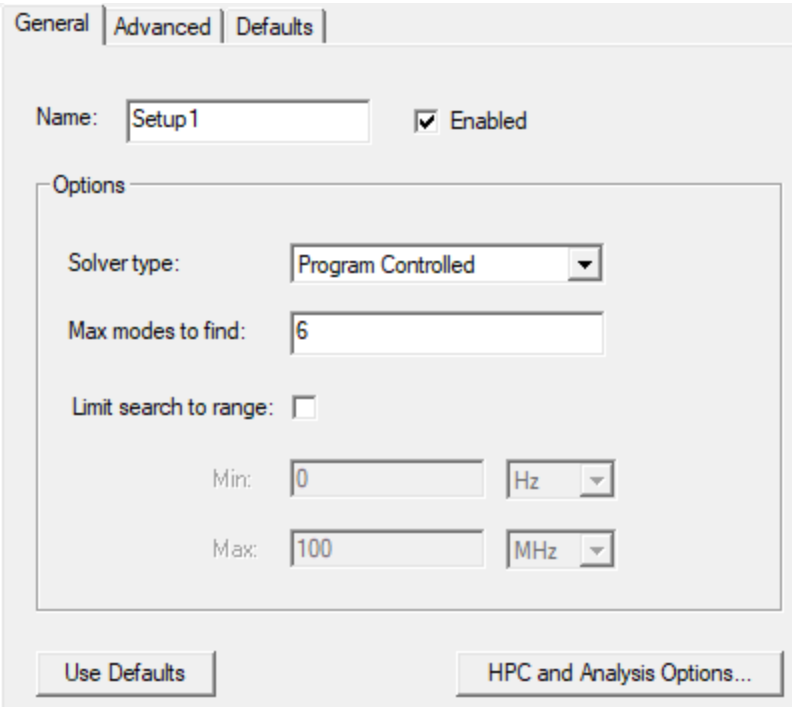
  - Consult the Ansys *Mechanical User's Guide* (specifically, *Configuring Analysis Settings* > *Analysis Settings for Most Analysis Types* > *Solver Controls*) for more information about the available solver choices.
  - The modal solution is undamped regardless of the selected solver. Damping is not currently available.
- **Max modes to find:** Specify the maximum number of natural vibration modes to include in the solution results. The default number of modes to solve is 6. The settings that follow may override this setting, causing fewer modes than specified here to be output by the solver.
- **Limit search to range:** Select this option to limit modal results to a specified frequency range. The following additional settings become available when this option is selected:
  - **Min:** The minimum frequency for which you want modal results. The solver will skip over modes at frequencies below this value without outputting associated results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **Max:** The solver will stop executing the solution at the first natural frequency that is above this value and will not produce any output for this frequency.
- Consult the Ansys *Mechanical User's Guide* (specifically, *Configuring Analysis Settings* > *Analysis Settings for Most Analysis Types* > *Solver Controls*) for more information about the available solver choices.

How to Set Up a Modal Solution:

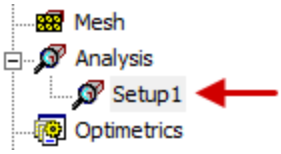
1. Use *one* of the following three methods of accessing the modal solution settings:
    - On the **Simulation** ribbon tab, click **Setup**.
    - Right-click **Analysis** in the Project Manager and choose **Add Solution Setup**.
    - Using the menu bar, click **Mechanical** > **Analysis** > **Add Solution Setup**.
2. In the dialog box that appears, specify the **Number of modes to find**.



3. Optionally, change the **Name** and any of the other default settings as needed (see the descriptions above).
4. Optionally, click **HPC and Analysis Options** to change any of your high performance computing configurations or options.
5. Optionally, specify settings for importing a mesh from another design or project. (See Mesh Linking.)
6. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The setup appears under *Analysis* in the Project Manager:



Results (Modal)

Modal analysis results presentations fall into three basic categories: *Modal Reports, Fields Reports,* and *Field Overlays*:

- **Modal Report:**

  There are several different display types available for fields reports, including 2D (Rectangular) Plots, 3D Plots, Data Tables, and more. The results that follow can be plotted or tabulated. However, by their nature, modal results are typically more suitable for tabulation than plotting. When presenting graphically, a discrete or bar-graph type of plot is best, since continuous curves interpolate values between calculated points. For modal analyses, resonant frequencies do not exist between vibration modes. Therefore the interpolated values are invalid.

  - **Frequency:** The natural vibration frequency in Hertz (cycles/second) for each mode.
  - **Participation Factor:** A scalar variable that indicates how strongly motion in the specified rotational or translational direction is represented in the eigenvector of the mode. These values are intended to measure the relative contribution of system modes to system states, and of system states to system modes, for linear systems.
  - **Effective Mass:** The mass that is participating in the vibration mode in the specified direction. The higher this resultant value, the more significant the particular vibration mode (that is, the more likely that external excitation will produce the resonant vibration mode).
  - **Effective Mass Ratio:** The *Effective Mass* result for the given mode and direction divided by the total mass of the design. The equation for this result is as follows:

    $$\frac{Me_i}{M}$$

    where Me_i is the effective mass participating in a particular direction for mode number i, and M is the total mass of the structure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sum of this result for all calculated modes represents the true cumulative effective mass participation that has been achieved by the number of modes calculated. You can report the sum of this result by using the **sum** *Range Function*.

> **Note:**
>
> The *Data Table Example* outlined in the *Creating a Modal Report* subtopic shows how to sum the *Effective Mass Ratio* modal results. Specifically, refer to steps 4 and 5 of that procedure.

The equation for the **sum** of the *Effective Mass Ratio* results in a given direction is as follows:

$$\frac{\sum_{i=1}^{n} Me_i}{M}$$

where *i* is the mode number, *n* is the total number of modes calculated, and the other variables are as previously defined.

- **Cumulative Effective Mass Fraction:** The cumulative sum of the *Effective Mass* values divided by the sum of the effective mass *that has participated for the specified number of calculated modes*. This results will always be equal to 1.0 at the last calculated mode, regardless of the number of modes calculated. (That is, it is a normalized result.) As such, it is only useful for showing the relative significance of modes within a limited frequency bandwidth.

CAUTION: This result does *not* represent the true cumulative effective mass participation, since only the participating mass through the last calculated mode is considered (not the full mass of the design). For a more meaningful evaluation of total effective mass participation, sum the *Effective Mass Ratio* values for the calculated modes and directions. (See the note under *Effective Mass Ratio* above.) Ensure that these sums satisfy any applicable design code or standard. For example, common guidelines state that 85% or 90% (0.85 or 0.9) cumulative effective mass participation must be achieved. If the applicable standard has not been satisfied, increase the number of modes specified in the setup and rerun the solution to capture additional mass participation.

Example: The following table shows the equations used for the *Cumulative Effective Mass Fraction* based on a total of 6 calculated modes. The third and sixth mode are tabulated:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mode Number	Cumulative Effective Mass Fraction Equation	Result
3	$\frac{\sum_{i=1}^{3} Me_i}{\sum_{i=1}^{6} Me_i}$	< 1
6	$\frac{\sum_{i=1}^{6} Me_i}{\sum_{i=1}^{6} Me_i}$	1.0

where *i* is the mode number and Me_i is the effective mass for mode number *i*.

The following example illustrates the difference between a cumulative sum of the *Effective Mass Ratios* and the *Cumulative Effective Mass Fraction* for a six-mode analysis and a total mass of 10 kg:

Mode Number	Effective $Mass_X$ (kg)	Effective Mass $Ratio_X$	Cumulative Eff. Mass $Fraction_X$
1	6.00	0.600	0.7186
2	0.50	0.050	0.7784
3	1.50	0.150	0.9581
4	0.00	0.000	0.9581
5	0.25	0.025	0.9880
6	0.10	0.010	**1.0000**
Totals:	**8.35**	**0.835**	

- **Design Volume:** The volume of each object comprising the design. Though you can plot volume points on a Cartesian grid, they are really only suitable for tabulation.

- **Fields Report:**

  There are several different display types available for fields reports, including 2D (Rectangular) Plots, 3D Plots, Data Tables, and more. One convenient way to generate a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fields report for modal solutions is to draw a polyline running along the faces and/or through the interior volume of your model. This polyline, which can consist of multiple straight and curved segments, defines a path along which you wish to plot a particular modal result.

You can also plot user-defined *Named Expressions* along a polyline, or create named expressions that do not need a polyline for plotting.

Regardless of the display type, the following modal field results are available for reporting.

- **Mag_Displacement:** The magnitude of the displacement for the specified vibration mode. This quantity is listed under the category *Calculator Expressions* but will not appear until a polyline is selected from the *Geometry* drop-down menu of the *Report* dialog box.

  > **Note:** Regarding Modal Displacement Results
  >
  > In a modal analysis, the magnitude of the displacement is *arbitrary*. Displacement results are not scaled to any specific excitation force, acceleration, or power spectral density. Therefore, the absolute displacement values are meaningless. Only the relative values are meaningful in that they demonstrate the shape of the deformed structure for each vibration mode. Do not be concerned if the displacement magnitudes are out of proportion with the size of the model.

- **Design Volume:** The volumes of the solid objects comprising the model. Though you can plot volume points on a Cartesian grid, they are really only suitable for tabulation.
- **Named Expression:** A custom result previously defined using the *Fields Calculator* (see the Fields Calculator Note on the *Creating a Fields Report* page).

- **Field Overlay:**

  You can overlay fields on selected solid or sheet objects, faces, or polylines. The following three results are available:

  - **Mag_Displacement:** A color contour plot of the displacement magnitude overlaid on a deformed representation of the model. This result indicates the vibration mode shape for the specified model. (See preceding note Regarding Modal Displacement Results.)
  - **Displacement_Vector:** This overlay has the same reported values as the Displacement Magnitude result, but it is presented as a vector plot. Arrows of different colors and lengths demonstrate the relative displacement magnitudes and directions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Named Expression:** A custom result previously defined using the *Fields Calculator* (see the Fields Calculator Note on the *Creating a Fields Report* page).

These reports and overlays are described in the subsections that follow along with the instructions for creating them.

Modal Report

The following display types are available for modal reports in Mechanical – Modal solutions:

- 2D (Rectangular) Plot
- Stacked Plot
- Data Table
- 3D Plot
- 2D Contour Plot

In this topic, we will focus on the display types that are most likely to be used for modal reports, specifically, Data Tables, 2D (Rectangular) plots, and Stacked plots. The procedure for creating other types of plots is similar. Data tables and discrete or bar graphs are best suited for plotting modal frequency results. It is best to avoid continuous curve plots because the interpolated values between calculated vibration modes would be invalid.

The available modal results are listed and described in the Results introduction. Examples of Data Tables, 2D plots, and Stacked plots follow:

- **Data Table:** Modal results are particularly well suited for this display type. The same information is available as for curve plots, but the information is presented as tabulated numerical results.

  The following image is an example of a data table listing natural vibration frequencies and effective mass ratio results in all degrees of freedom for each mode. A total of fifteen modes were calculated. The *Number Format* for the Mode column was changed to integer, the *Field Precision* of the ModeFreq column was reduced to 2 (decimal places), and the column widths were customized:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency Table 1 MechanicalDesign1 Ansys

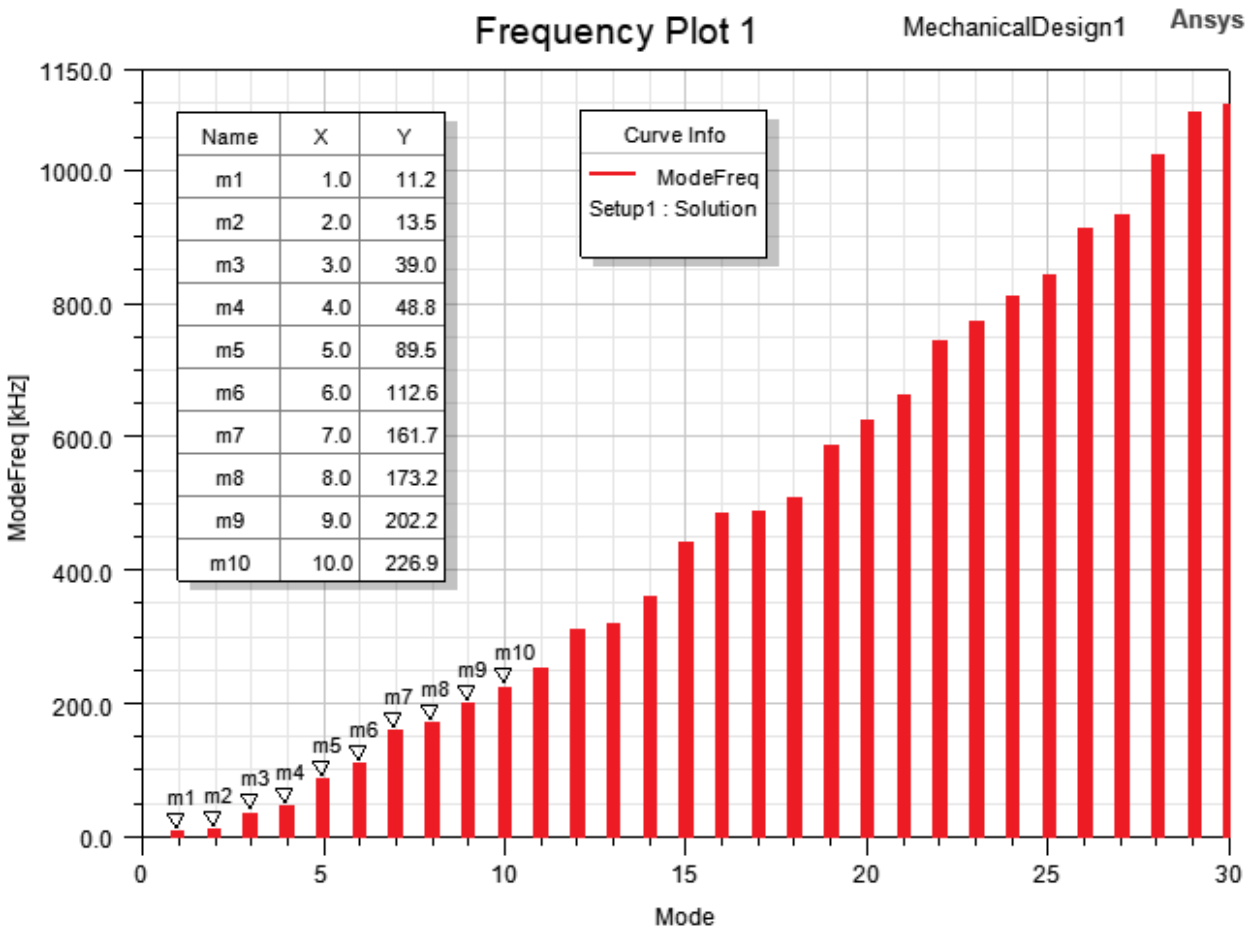
	Mode	ModeFreq [Hz] Setup1 : Solution	EffectiveMassRatio_DirX Setup1 : Solution	EffectiveMassRatio_DirY Setup1 : Solution	EffectiveMassRatio_DirZ Setup1 : Solution	EffectiveMassRatio_RotX Setup1 : Solution	EffectiveMassRatio_RotY Setup1 : Solution	EffectiveMassRatio_RotZ Setup1 : Solution
1	1	97.74	0.000000	0.671589	0.003000	0.000021	0.002410	0.000004
2	2	118.64	0.000000	0.002202	0.002400	0.000017	0.805815	0.007098
3	3	152.29	0.564001	0.000000	0.000011	0.016567	0.000000	0.004046
4	4	153.47	0.000010	0.001795	0.595922	0.004339	0.000152	0.001054
5	5	186.62	0.012208	0.000000	0.000000	0.182957	0.000000	0.000084
6	6	192.56	0.000000	0.000068	0.011385	0.000063	0.002063	0.451014
7	7	196.45	0.015114	0.000000	0.000000	0.280469	0.000000	0.000085
8	8	232.02	0.001346	0.000000	0.000000	0.004594	0.000000	0.000009
9	9	271.13	0.000000	0.063709	0.008376	0.000058	0.000473	0.006106
10	10	299.72	0.000007	0.000091	0.064333	0.000542	0.004013	0.013097
11	11	300.38	0.056241	0.000000	0.000008	0.026355	0.000000	0.000463
12	12	311.04	0.000000	0.000276	0.006907	0.000061	0.018961	0.047244
13	13	315.47	0.000548	0.000000	0.000000	0.052889	0.000000	0.000007
14	14	341.43	0.000000	0.000850	0.001355	0.000011	0.043045	0.010618
15	15	343.16	0.004275	0.000000	0.000000	0.000020	0.000000	0.000028

Tables are also well suited for presenting results on which you have performed mathematical operations (such as summing a quantity or reporting the maximum, minimum, or average values).

- **2D (Rectangular) Plots:** This display type produces continuous traces by default. After creating the plot, change the *Trace Type* attribute to a discrete or bar type. You can plot one quantity or multiple quantities using this format, with all of the results sharing the same Cartesian grid and axes.

  The following image is an example of a *Frequency vs. Mode* 2D (rectangular) plot for a 30 mode solution with the *Trace Type* attribute set to *Bar Infinity*. Additionally, the X axis *Number Format* was defined as *Integer*, and the Y axis scale and units were customized. Finally, markers were added to tabulate the exact frequencies of the first ten modes:

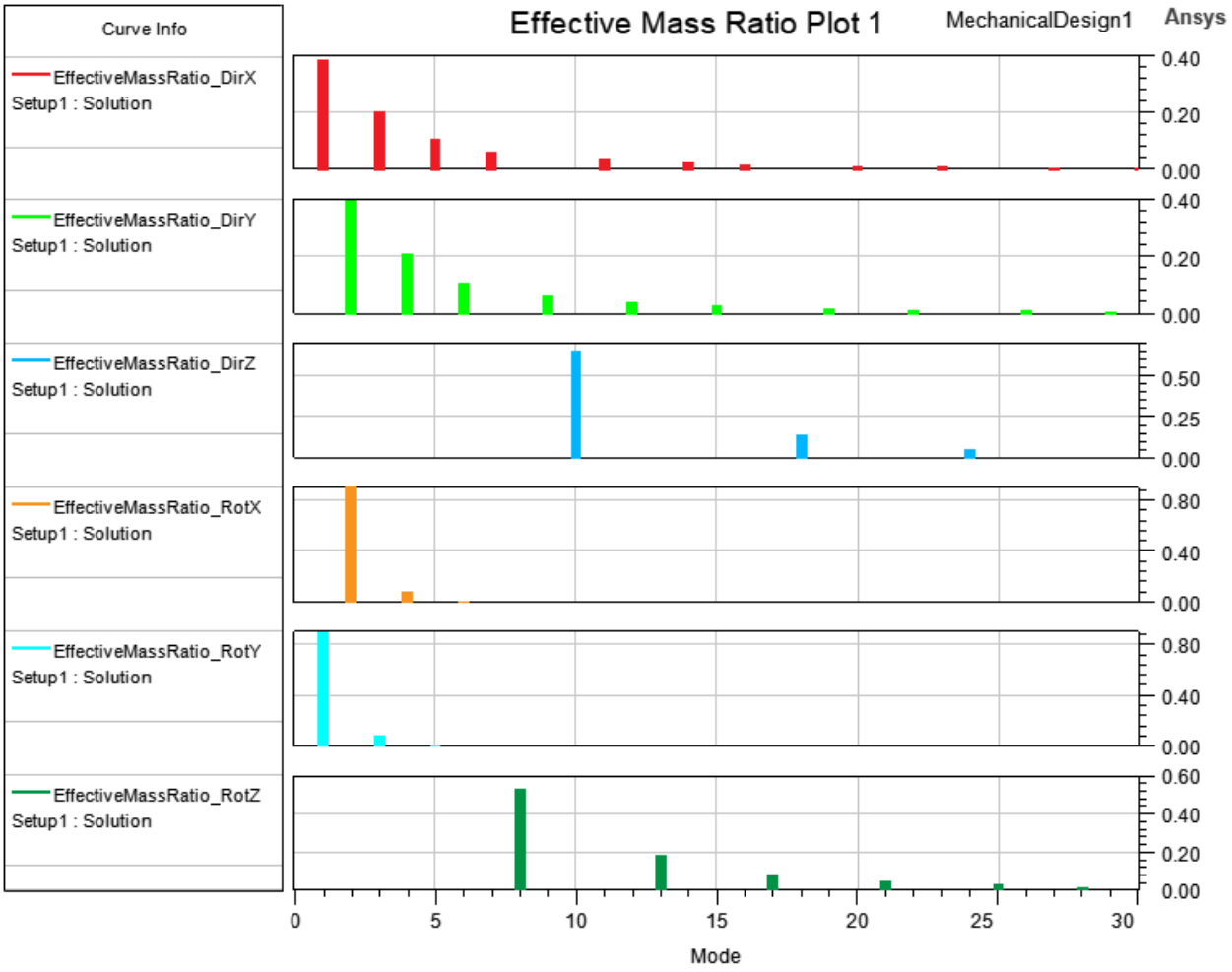
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Stacked Rectangular Plots:** This display type is intended for multiple traces. Unlike the 2D plot previously described, this type puts every individual trace in a separate Cartesian grid stacked vertically. The X scale is shared by all traces, but the Y scale is adjusted to suit each individual trace.

  The following image is an example of an Effective Mass Ratio vs. Mode stacked plot showing all six degrees of freedom:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

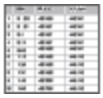


This plot provides a good visual indication of the most significant vibration modes for each degree of freedom.

See the subtopic, Creating a Modal Report, for step by step instructions on creating and customizing modal reports.

Creating a Modal Report

Data Table Example:

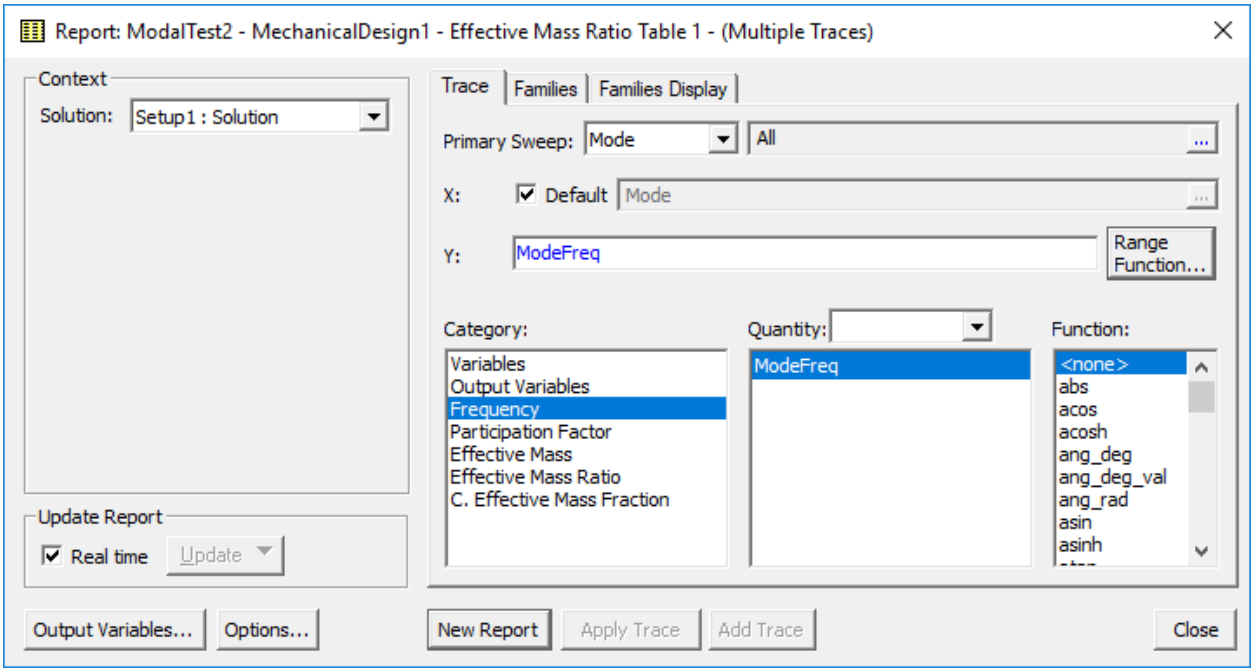
1. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Modal Report** >  **Data Table**.
   - Using the menu bar, click **Mechanical** > **Results** > **Create Modal Report** > **Data Table**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click **Results** in the Project Manager and choose **Create Modal Report** > **Data Table** from the shortcut menu.

2. Select the desired **Solution, Category,** and **Quantity** or **Quantities** to tabulate.

   The following report setup example shows the settings to tabulate the *Mode Frequencies*.



3. Click **New Report** but keep the *Report* dialog box open.

   The table appears in a new window. The following data table example is consistent with the *Report* dialog box settings shown under step 2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

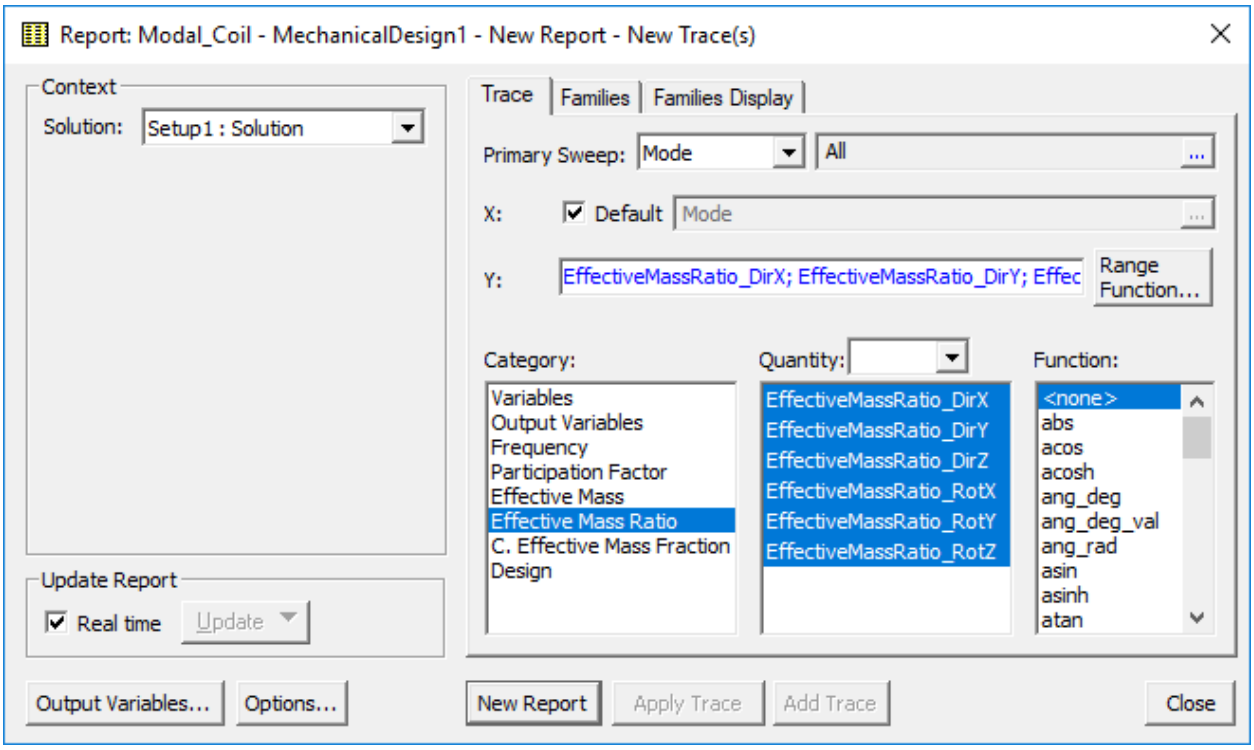
Frequency Table 1 Ansys

	Mode	ModeFreq Setup1 : Solution
1	1.000000	50040.904571
2	2.000000	78153.399161
3	3.000000	213051.742382
4	4.000000	337422.217223
5	5.000000	488888.599159
6	6.000000	515625.576735
7	7.000000	785880.132333
8	8.000000	790595.736632
9	9.000000	934037.901836
10	10.000000	1049026.203000
11	11.000000	1367122.021745
12	12.000000	1445237.090999
13	13.000000	1671432.619382
14	14.000000	1957441.163799
15	15.000000	2022774.020078

4. To add additional results to the current report, select the desired **Category** and **Quantities** in the *Report* dialog box.

   The following sample report setup will add effective mass ratio results to the current report:

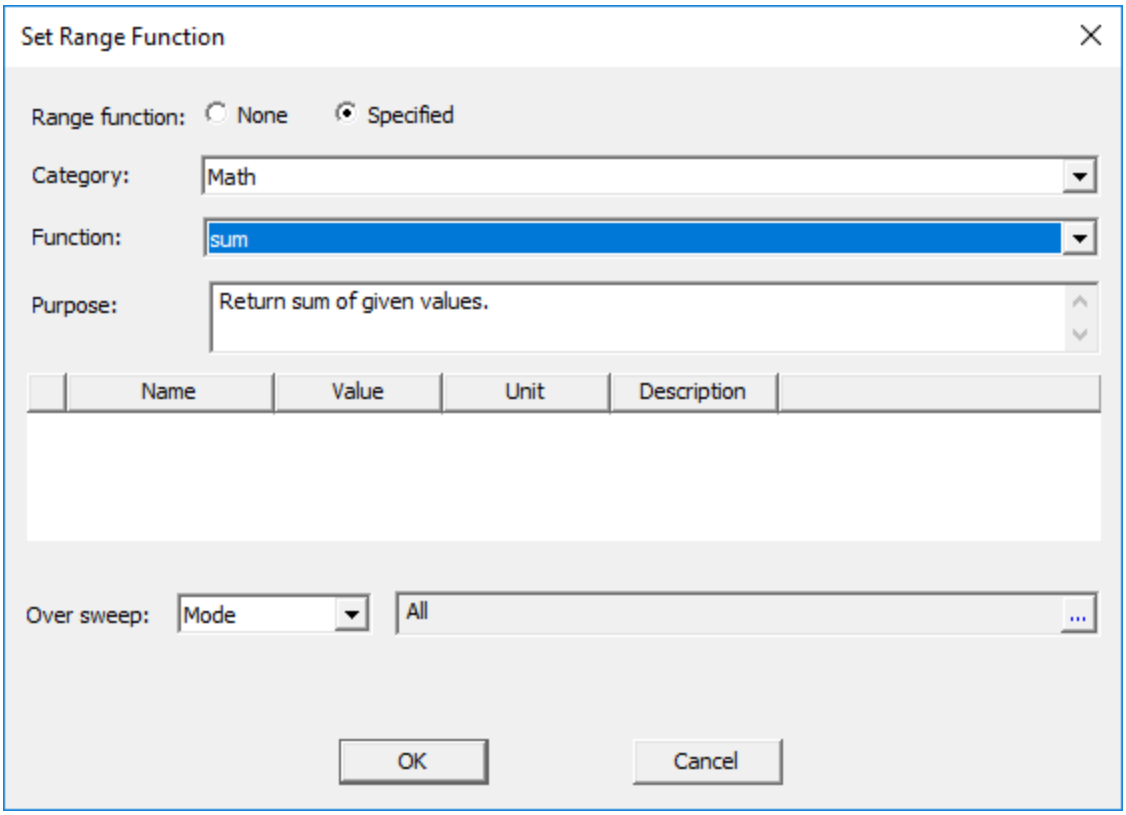
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Optionally, click **Range Function** to define a mathematical operation to perform on the specified results. Then:
   a. Choose the **Category** and **Function** to apply to the selected quantities from the drop-down menus.

      The following example shows the mathematical **sum** function being applied to the specified quantities. The sum will include the contribution of all modes and will be applied separately to each degree of freedom:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Click **OK** to apply the operation and close the *Set Range Function* dialog box.

6. Click **Add Trace** and **Close**.

   Six columns are added to the Frequency Table. Though the effective mass ratio sums are listed in the Mode 1 row, the numbers are the total for all calculated modes.

7. Customize the table as follows:
   a. Click on the **Mode** column header.
   b. In the **Data Filter** tab of the docked *Properties* window, change the **Number Format** to **Integer**.
   c. Click on the **ModeFreq[Hz]** column header.
   d. In the **Data Filter** tab of the docked *Properties* window, change the **Field Precision** to **2**.
   e. Click and drag the column borders to customize the column widths.

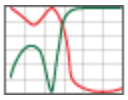
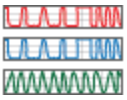
   The sample table with the described customizations is shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency Table 1 MechanicalDesign1 Ansys

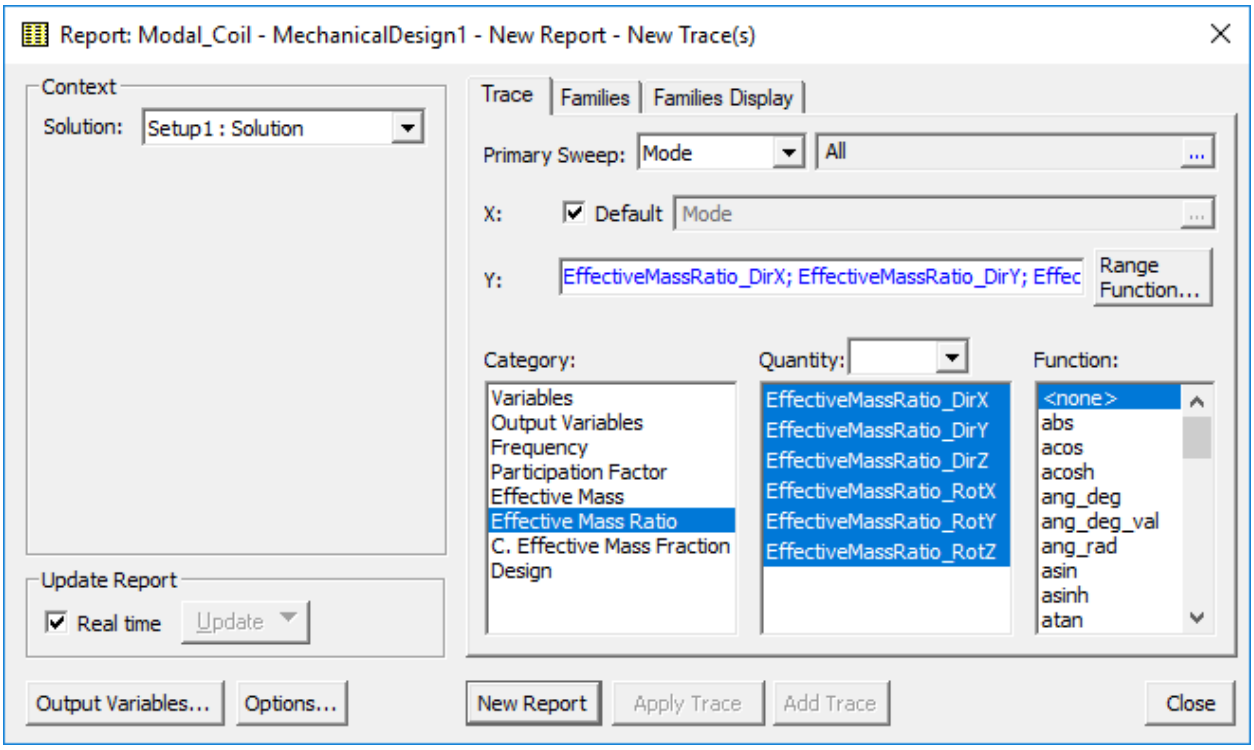
	Mode	ModeFreq [Hz] Setup1 : Solution	sum(EffectiveMassRatio_DirX) Setup1 : Solution	sum(EffectiveMassRatio_DirY) Setup1 : Solution	sum(EffectiveMassRatio_DirZ) Setup1 : Solution	sum(EffectiveMassRatio_RotX) Setup1 : Solution	sum(EffectiveMassRatio_RotY) Setup1 : Solution	sum(EffectiveMassRatio_RotZ) Setup1 : Solution
1	1	97.74	0.653751	0.740580	0.693697	0.568961	0.876932	0.540954
2	2	118.64						
3	3	152.29						
4	4	153.47						
5	5	186.62						
6	6	192.56						
7	7	196.45						
8	8	232.02						
9	9	271.13						
10	10	299.72						
11	11	300.38						
12	12	311.04						
13	13	315.47						
14	14	341.43						
15	15	343.16						

2D (Rectangular) or Stacked Plot Example:

1. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Modal Report** >  **2D** or  **Stacked**.
   - Using the menu bar, click **Mechanical** > **Results** > **Create Modal Report** > **Rectangular Plot** or **Rectangular Stacked Plot**.
   - Right-click **Results** in the Project Manager and choose **Create Modal Report** > **Rectangular Plot** or **Rectangular Stacked Plot** from the shortcut menu.
2. Select the desired **Solution, Category,** and **Quantity** or **Quantities** plot.

   The following report setup example shows the settings to plot the *Effective Mass Ratios* for all six degrees of freedom:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



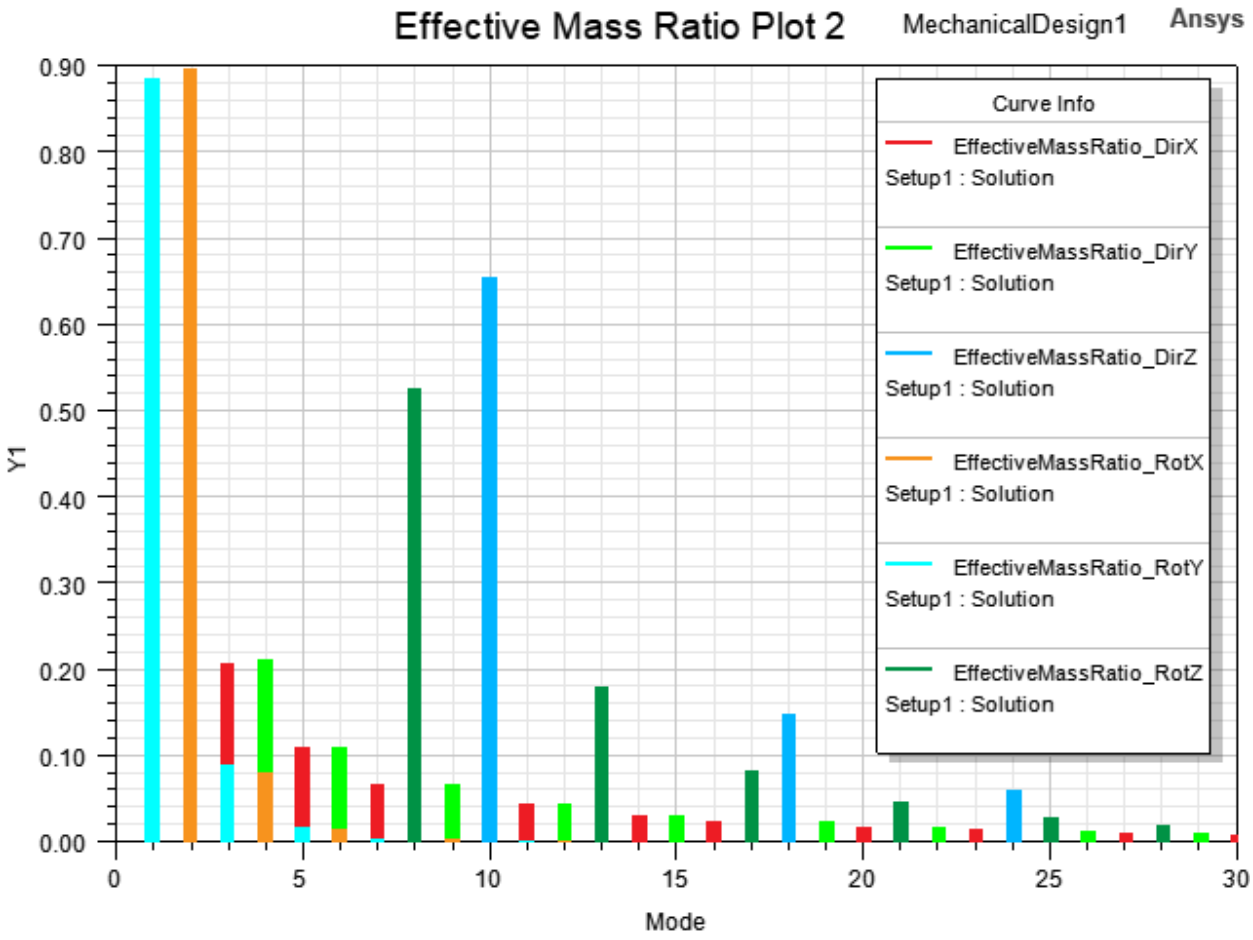
3. Click **New Report**.
4. Optionally, repeat steps 2 and 3 to add more traces to the current plot. Otherwise, click **Close**.
5. Your plot will initially appear as one or more continuous traces. Make the following changes to produce a plot more suited to modal results:
    a. In the plot window, click one of the traces to select it. Also, if there are multiple traces, press **Ctrl+A** to select all of the remaining traces too.

       The trace settings appear in the docked *Properties* window.

    b. In the **Attribute** tab of the docked *Properties* window, change the **Trace Type** to one of the following options that are more suitable for plotting modal results:
        - Discrete
        - Stick Zero
        - Stick Infinity
        - Bar Zero
        - Bar Infinity
        - Step
    c. In the plot window, click the **X axis** to adjust its settings.
    d. In the **Axis** tab of the docked *Properties* window, change the **Number Format** to **Integer**.
    e. Click in the plot window background area to deselect the axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following example is a 2D plot of the Effective Mass Ratio in all directions for 30 modes. The *Trace Type* was set to *Bar Infinity*:



This plot gives you a quick visual indication of which translational or rotational mass participation is the most significant for each mode. For example, mode 2 has the greatest effective mass ratio in the X rotational direction, mode 1 in the Y rotational direction, mode 10 in the Z translational direction, and mode 8 in the Z rotational direction. Those are the top four modes most likely to be excited by ground vibration at the corresponding mode frequencies.

Fields Report

The following display types are available for fields reports in Mechanical – Modal solutions:

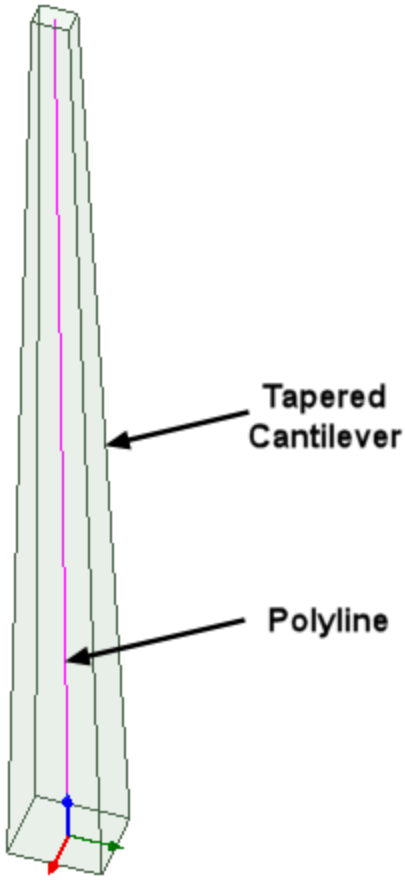
- 2D (Rectangular) Plot
- Stacked Plot
- Data Table
- 3D Plot
- 2D Contour Plot

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this topic, we will focus on the display types that are most likely to be used for modal fields reports, specifically, 2D (Rectangular) plots, Stacked plots, and Data Tables. The procedure for creating other types of plots is similar. The subtopic, Creating a Fields Report, gives step-by-step instructions for generating modal field plots and data tables.

With the exception of *Design Volume*, all other *predefined* field results require polyline geometry defining the path along which the results are to be plotted or tabulated. This polyline can be comprised of a single segment or multiple segments, and can include any combination of straight, arc, or spline segments. You can manually draw the polyline along the faces and/or through the interior of the model's objects, or you can select edges of your model and convert them to line objects.

For example, the following image is a simple cantilever with a tapered rectangular cross section. A single-segment line has been drawn from the centroid of the base to the centroid of the top:



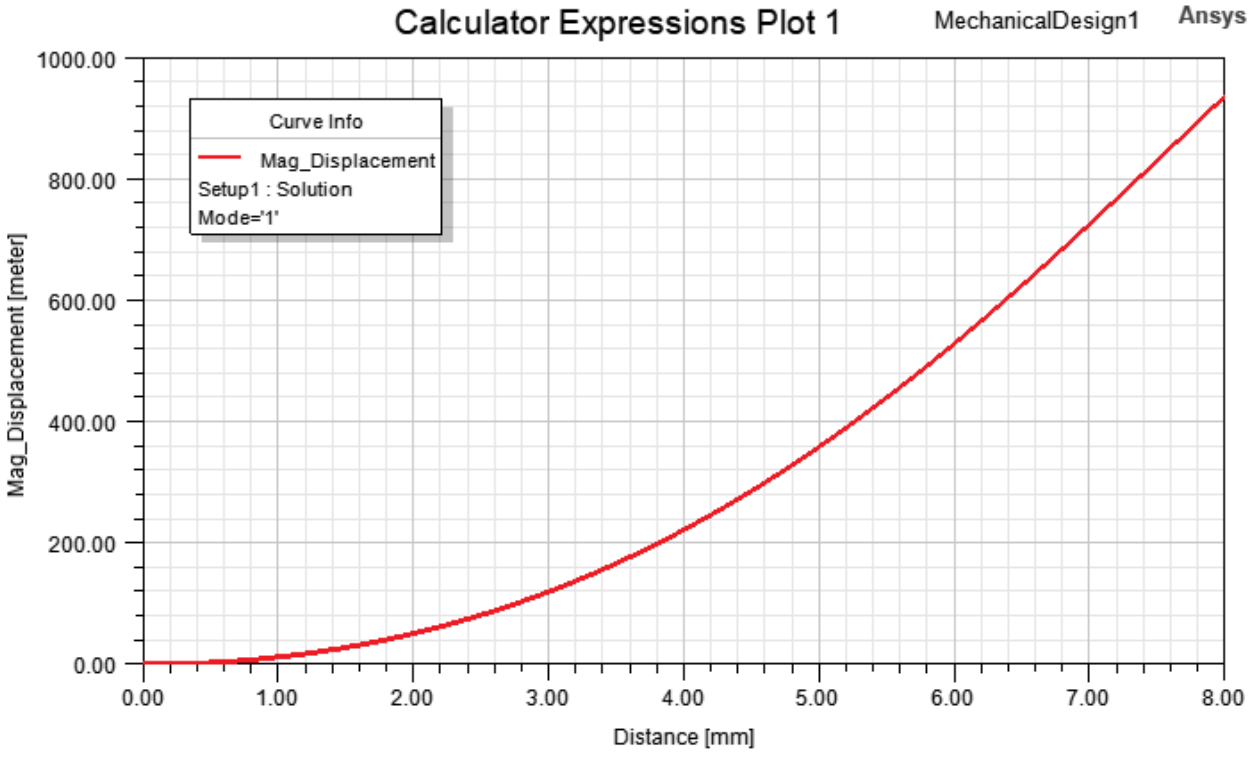
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Whether you draw the polyline before or after running the solution, it's a good idea to create it as a **non-model** part. However, the solver will ignore line and sheet objects even if the **Model** option is selected in the object properties. If you create a polyline after solving, you will be prompted to create a non-model object, or the solution results will be invalidated.

The available fields report results for Mechanical – Modal analyses are listed under Fields Report in the *Results* topic. Examples of 2D plots, Stacked plots, and Data Tables follow:

- **2D (Rectangular) Plot:** This type of plot can contain one trace or multiple traces, each of which is plotted within the same Cartesian grid.

  The following plot shows *Mag_Displacement* versus the *Distance* along the polyline for Mode 1. It is based on the preceding tapered cantilever and polyline example:

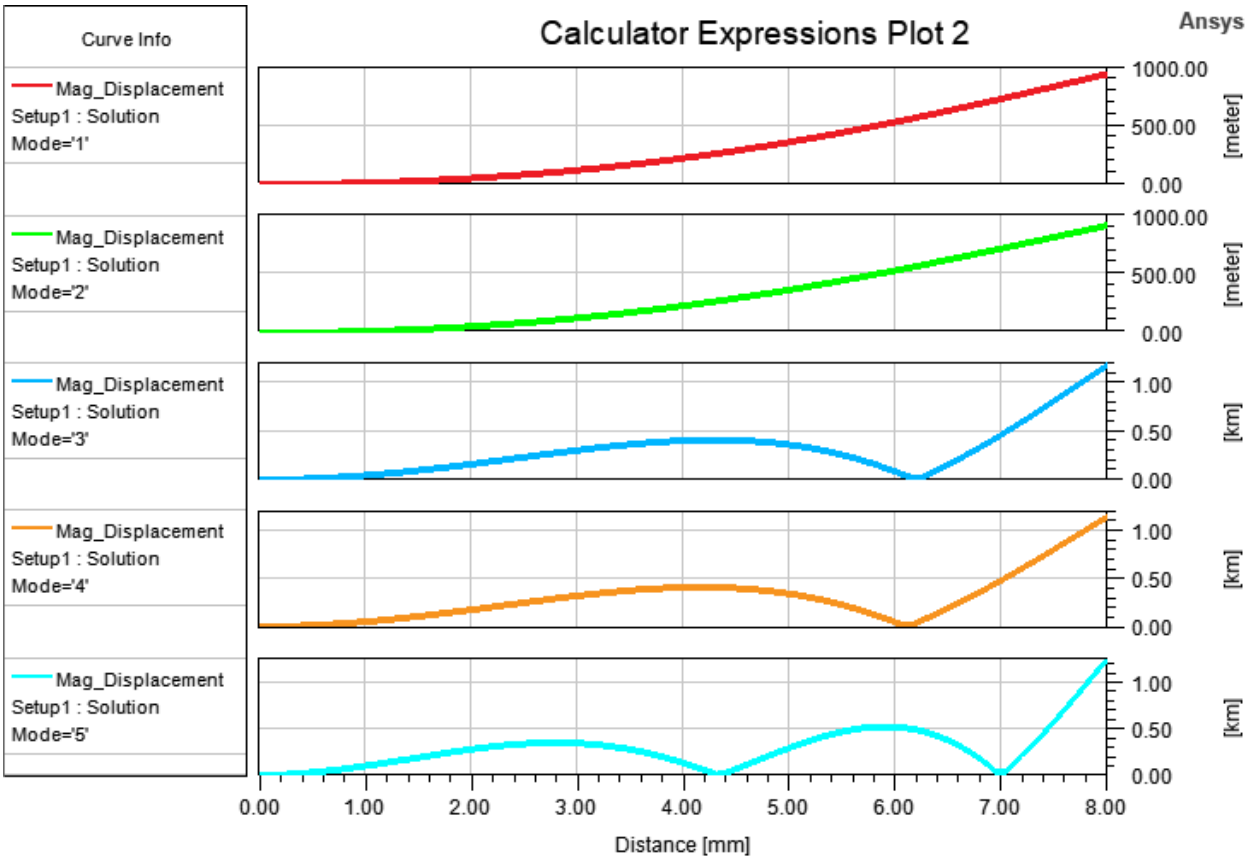


  Only the relative displacement along the length of the cantilever is significant. For modal analyses, the absolute displacement magnitudes are unscaled and arbitrary.

- **Stacked Rectangular Plot:** This type of plot is intended for multiple traces. Unlike the 2D plot previously described, this type puts every individual trace in a separate Cartesian grid stacked vertically. The X scale is shared by all traces, but the Y scale is adjusted to suit each individual trace.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following plot shows *Mag_Displacement* versus the *Distance* along the polyline for the first five vibration modes. It is based on the preceding tapered cantilever and polyline example:



Only the relative displacement values within each individual trace are relevant.

- **Data Table:** Tabulation of any of the available fields report results, as an alternative to a plotted curve.

  The following example contains exactly the same results as presented in the preceding stacked plot, except in tabular form:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Expressions Table 1 MechanicalDesign1 Ansys

	Distance [mm]	Mag_Displacement [meter] Setup1 : Solution Mode='1'	Mag_Displacement [meter] Setup1 : Solution Mode='2'	Mag_Displacement [km] Setup1 : Solution Mode='3'	Mag_Displacement [km] Setup1 : Solution Mode='4'	Mag_Displacement [km] Setup1 : Solution Mode='5'
32	4.960025	352.401235	359.537129	0.362369	0.341242	0.272535
33	5.120026	377.247371	383.876302	0.337672	0.312955	0.337970
34	5.280026	402.947445	408.942463	0.306661	0.278628	0.396950
35	5.440027	429.482074	434.710362	0.269043	0.238105	0.446745
36	5.600028	456.834601	461.158801	0.224617	0.191292	0.485220
37	5.760029	484.974181	488.248164	0.173159	0.138119	0.509149
38	5.920030	513.873287	515.951788	0.114588	0.078604	0.516376
39	6.080030	543.499003	544.231160	0.051119	0.010862	0.504570
40	6.240031	573.803640	573.038527	0.019746	0.061507	0.471469
41	6.400032	604.754140	602.343472	0.103665	0.136535	0.415875
42	6.560033	636.283011	632.080159	0.190049	0.219543	0.336487
43	6.720034	668.358398	662.218439	0.282758	0.307655	0.233141
44	6.880034	700.898272	692.688757	0.381192	0.400202	0.106325
45	7.040035	733.845129	723.443574	0.484778	0.496653	0.041609
46	7.200036	767.128481	754.427540	0.592772	0.596356	0.212231
47	7.360037	800.671796	785.584203	0.704288	0.698590	0.398348
48	7.520038	834.403550	816.860921	0.818433	0.802636	0.597661
49	7.680038	868.255448	848.205990	0.934300	0.907752	0.805900
50	7.840039	902.166653	879.581775	1.051044	1.013369	1.018859
51	8.000040	936.091181	910.954323	1.167992	1.118951	1.232976

The scroll bar on the right side of the table enables you to scroll through all the data points (51 in this case).

Creating a Fields Report

The following is the general procedure for creating any type of Fields Report. It includes drawing a polyline to use as the path along which field results will be plotted or tabulated. This step is necessary for the predefined displacement magnitude results. However, you can plot or tabulate *Design Volumes* and *Named Expressions* you create using the *Fields Calculator* (see note below) without creating any polyline geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The *Fields Calculator* is a powerful and flexible tool for post processing your analysis results. Use it to perform simple or complex mathematical operations on the raw results data. With an appropriate named expression defined, you can generate a report that does not require a polyline as its basis. For example, you could define a face list and create a named expression that integrates the heat flow rate for all faces in the list. Then, in a parametric analysis, you could base the plot's X axis on a design parameter. Finally, base the plot's Y axis on your named expression, thus creating a plot of total heat flow rate of the faces versus the swept parameter. There are many other possibilities as well.

You can access the *Fields Calculator* by right-clicking **Field Overlays** in the Project Manager and selecting **Calculator** from the shortcut menu. For more information, search the Ansys Electronics Desktop help for "using the fields calculator" (include the quotation marks to limit search results to instances of the full phrase). Also, refer to the *Related Topics* links at the bottom of the *Using the Fields Calculator* page.

1. Create a polyline path to use as the basis of the plot's X axis using *one* of the following two methods:
   - **Draw a polyline manually:**
     a. On the **Draw** ribbon tab, click **Draw line**.
     b. If prompted to create a non-model object (because a solution already exists), click **Yes**.
     c. Draw a line or series of lines along which you wish to determine the thermal results. The line segments should pass through or along the surface of the solid model. Use one of the following three drawing methods:
        - Specify the line start and end points graphically (by clicking in the Modeler window). You can snap to model vertices (endpoints), midpoints or edges, grid points, and more.
        - Specify the points by typing the coordinates into the **X, Y,** and **Z** text boxes in the status bar.
        - Draw an arbitrary line and edit it's properties afterward (in the docked *Properties* window or the *Properties* dialog box).

          You can switch between straight and curved segments by right-clicking in the Modeler window and choosing the desired **Set Edge Type** option.

     d. After clicking the last endpoint, right-click in the Modeler window and choose **Done** to terminate the *Draw Line* command.

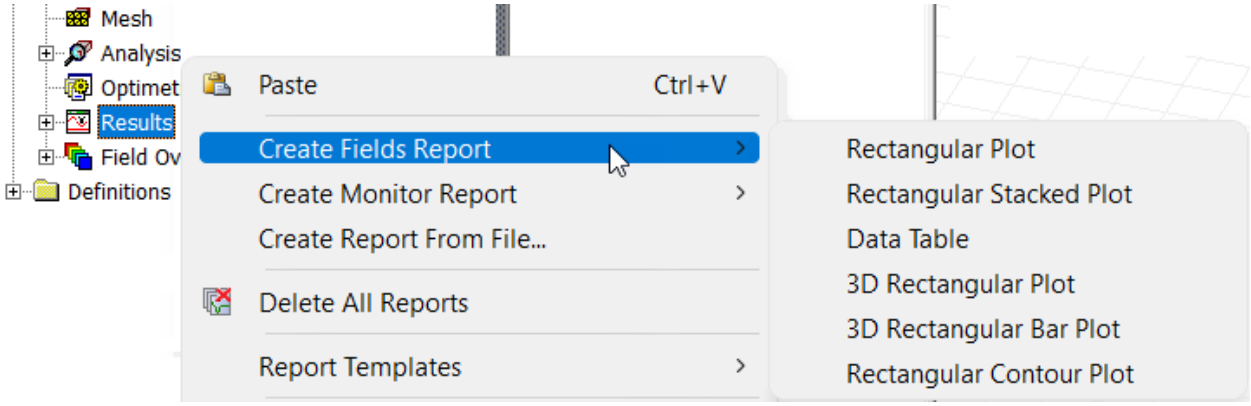
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notice the name of the line object that has been added to the History Tree. The default name for the first line object is *Polyline1*, and you can change the name in the docked *Properties* window if desired.

- **Convert model edges to a polyline:**
  a. In *Edge* selection mode, select a model edge or a contiguous series of model edges to select them.
  b. On the **Draw** ribbon tab, click **Edge** > **Create Object From Edge**.
  c. While the objects are still selected, click **Unite** on the **Draw** ribbon tab.

  Notice the name of the united object that has been added to the History Tree. You can change the name in the docked *Properties* window if desired.

2. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Fields Report** and choose the ***desired display type*** (such as, **2D**, **Data Table**, or **Stacked**).
   - Using the menu bar, click **Mechanical** > **Results** > **Create Fields Report** > ***desired display type***.
   - Right-click **Results** in the Project Manager and choose **Create Fields Report** > ***desired display type***:

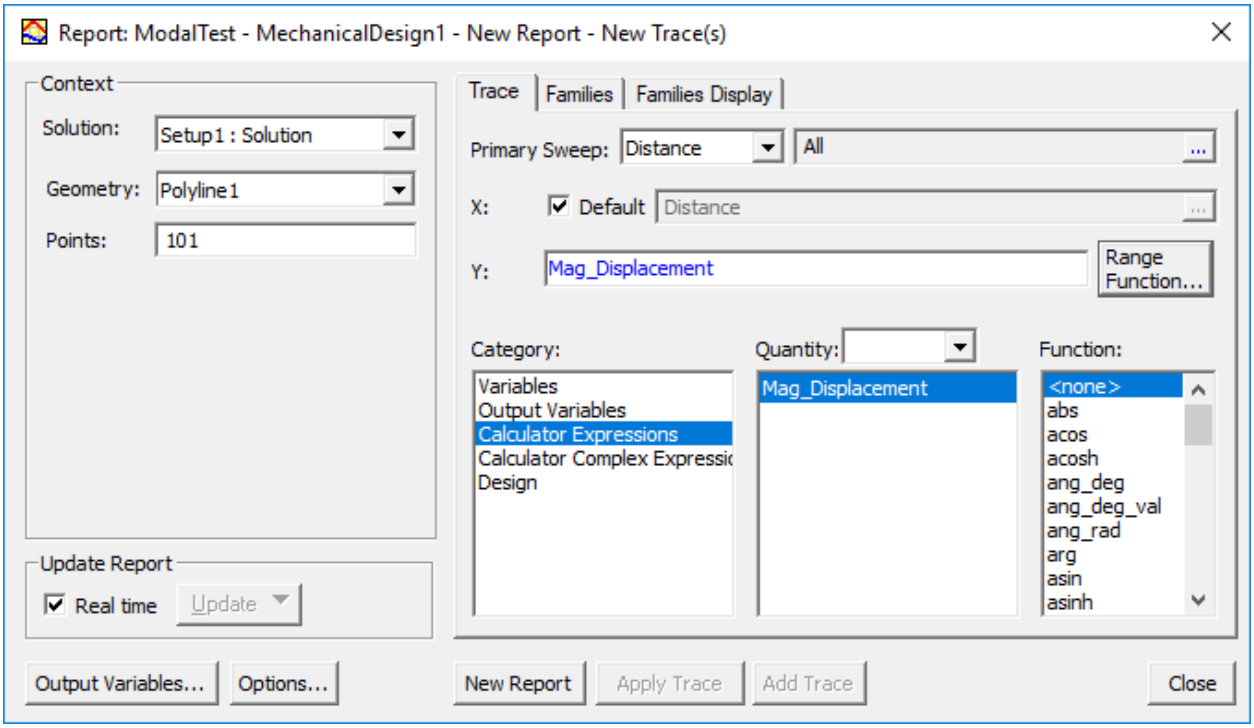


3. Specify the following settings in the *Trace* tab of the *Report* dialog box:
   a. Choose the desired solution setup from the **Solution** drop-down menu (if you solved more than one setup).
   b. Choose the polyline you created previously from the **Geometry** drop-down menu.
   c. Optionally, specify a different number of **Points** to calculate along the specified polyline. (The default number of points is 101, which results in one hundred trace segments between the first and last data points.)
   d. In the **Category** list, select **Calculator Expressions**, if it is not already selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Select the desired **Quantity** or **Quantities** to plot. (You do not need to specify a *Function* for modal field results.)

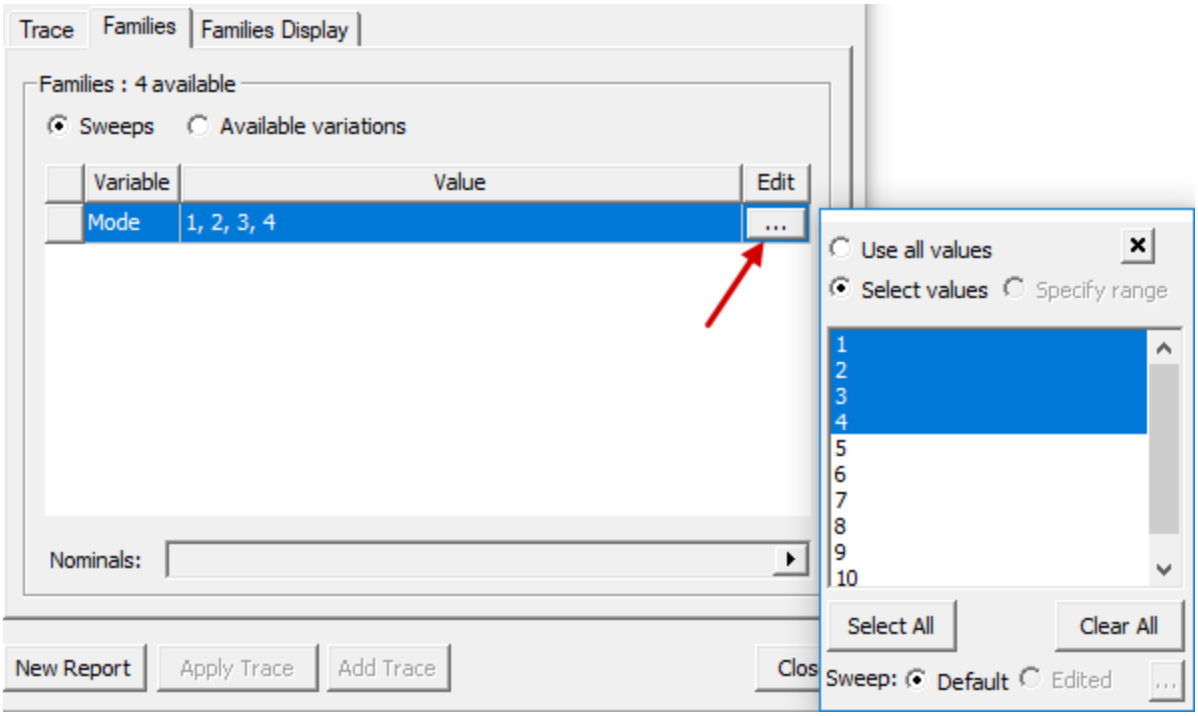
The following example shows the settings for plotting or tabulating *Mag_ Displacement* results along *Polyline1*:



4. In the *Families* tab, click the **ellipsis** button (**...**) at the right end of the **Mode** row. Then,
   a. Select the mode or modes you want to tabulate or plot.

   The following example shows Modes 1 through 4 selected:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Optionally, select **Use all values** to include every calculated mode in the report.
c. Click the **X** in the upper right corner to close the variation selection dialog box.

5. Click **New Report**.

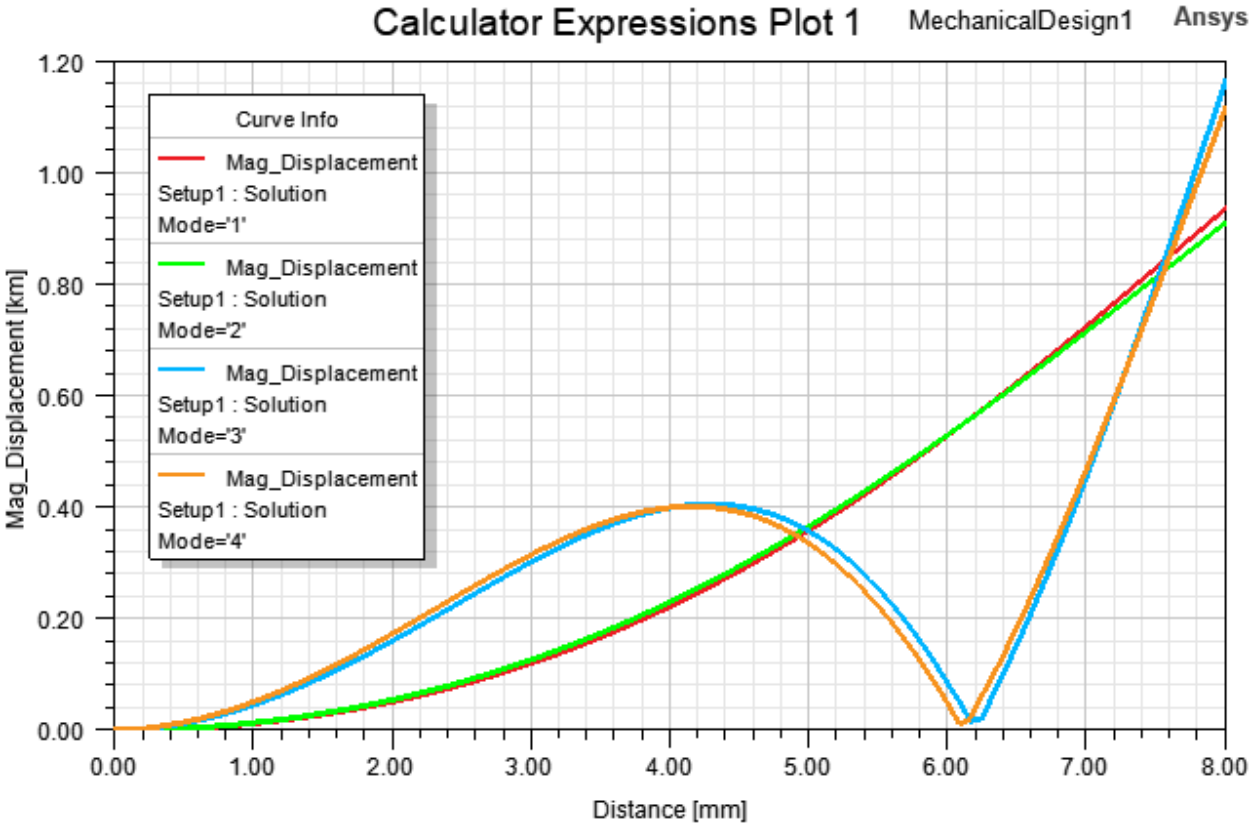
   The plot or data table appears in a new window.

6. Optionally, if you've created a plot, you can double-click inside the legend box to access the *Properties* dialog box and customize many plot parameters, such as:
   - Trace colors, style, and width
   - Enable data point symbols and change their color, fill, style, and frequency
   - Grid options
   - Headers
   - X and Y axis parameters and scaling
7. Optionally, if you're created a data table, you can customize it in the following ways:
   - Drag the column borders to resize the columns.
   - Click on a column heading and then, in the docked *Properties* window, adjust the **Units, Field Width,** or **Field Precision** values.
8. Optionally, select parameters for one or more additional results to add to the current plot or table and click **Add Trace**.
9. Click **Close** to exit the *Report* dialog box.

Examples:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following example is a 2D plot showing the displacement magnitude along Polyline1 for the first four vibration modes:



The following example presents the same results as the preceding plot, except in tabular form. The *Field Precision, Field Width,* and column widths were customized for the five columns with headings:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Expressions Table 1 MechanicalDesign1 Ansys

	Distance [mm]	Mag_Displacement [meter] Setup1 : Solution Mode='1'	Mag_Displacement [meter] Setup1 : Solution Mode='2'	Mag_Displacement [km] Setup1 : Solution Mode='3'	Mag_Displacement [km] Setup1 : Solution Mode='4'
1	0.00	0.000	0.000	0.000	0.000
2	0.40	1.940	2.095	0.008	0.009
3	0.80	7.758	8.396	0.029	0.033
4	1.20	17.706	19.072	0.063	0.070
5	1.60	32.026	34.316	0.107	0.117
6	2.00	50.950	54.302	0.159	0.172
7	2.40	74.701	79.186	0.216	0.230
8	2.80	103.488	109.099	0.273	0.288
9	3.20	137.505	144.142	0.327	0.339
10	3.60	176.913	184.373	0.370	0.378
11	4.00	221.835	229.800	0.398	0.399
12	4.40	272.339	280.369	0.404	0.396
13	4.80	328.424	335.949	0.381	0.364
14	5.20	389.991	396.319	0.323	0.297
15	5.60	456.835	461.159	0.225	0.191
16	6.00	528.597	530.021	0.083	0.047
17	6.40	604.754	602.343	0.104	0.137
18	6.80	684.571	677.413	0.331	0.353
19	7.20	767.128	754.428	0.593	0.596
20	7.60	851.319	832.528	0.876	0.855
21	8.00	936.091	910.954	1.168	1.119

Field Overlay and Animation

In modal analyses, a field overlay provides a visual indication of the mode shapes by overlaying a color contour or vector plot of the geometry's *displacement* on a deformed view of the model. The absolute displacement magnitudes have no meaning in modal analyses because they are not scaled to any particular force, acceleration, or power spectral density excitation. Only the relative displacements are significant because they indicate the shape of the natural vibration modes. Since the displacement magnitudes are arbitrary, and may well be disproportionate to the model size, the deformation is scaled to produce reasonable deformed shapes.

To Produce a *Displacement Magnitude* or *Displacement Vector* Overlay and Animate it:

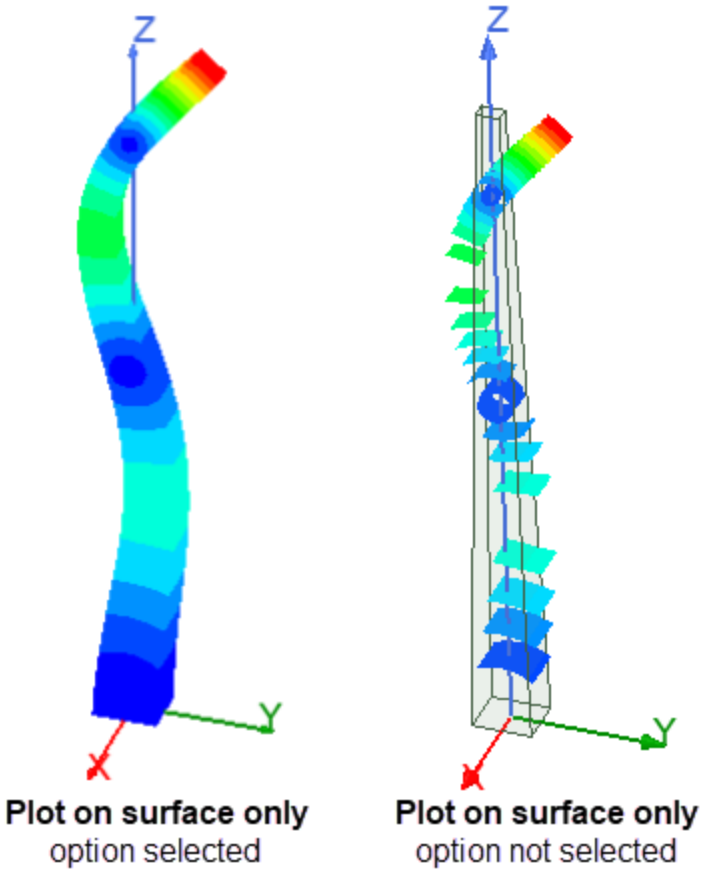
1. Select the *Faces* or *Objects* on which you want to overlay displacement results.
2. Access the *Create Field Plot* dialog box using *one* of the following three methods:
   - Using the menu bar, click **Mechanical** > **Fields** > **Plot Fields** > **Mag_ Displacement** or **Displacement_Vector**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click Field Overlays in the Project Manager and choose **Plot Fields** > **Mag_ Displacement** or **Displacement_Vector** from the shortcut menu.
- Right-click in the Modeler window and choose **Plot Fields** > **Mag_Displacement** or **Displacement_Vector**.

3. Specify the **Solution** and the **Mode** to plot from the provided drop-down menus.
4. For the best rendering of displacement results, ensure that the **Plot on surface only** option is enabled (this is the default setting).

   When this option is selected, the displacements are rendered on the model surfaces only. If you deselect this option, the displacements are rendered for the full model volume, but they are rendered as IsoValSurfaces. Rather than the full volume of the model being rendered in varying colors, surfaces connect points of the same displacement value and are rendered for each scale division specified in the plot settings (15 values by default). This plot style enables you to see the results on the interior of solids without creating clipping planes. The following displacement magnitude images compare the two plot styles:

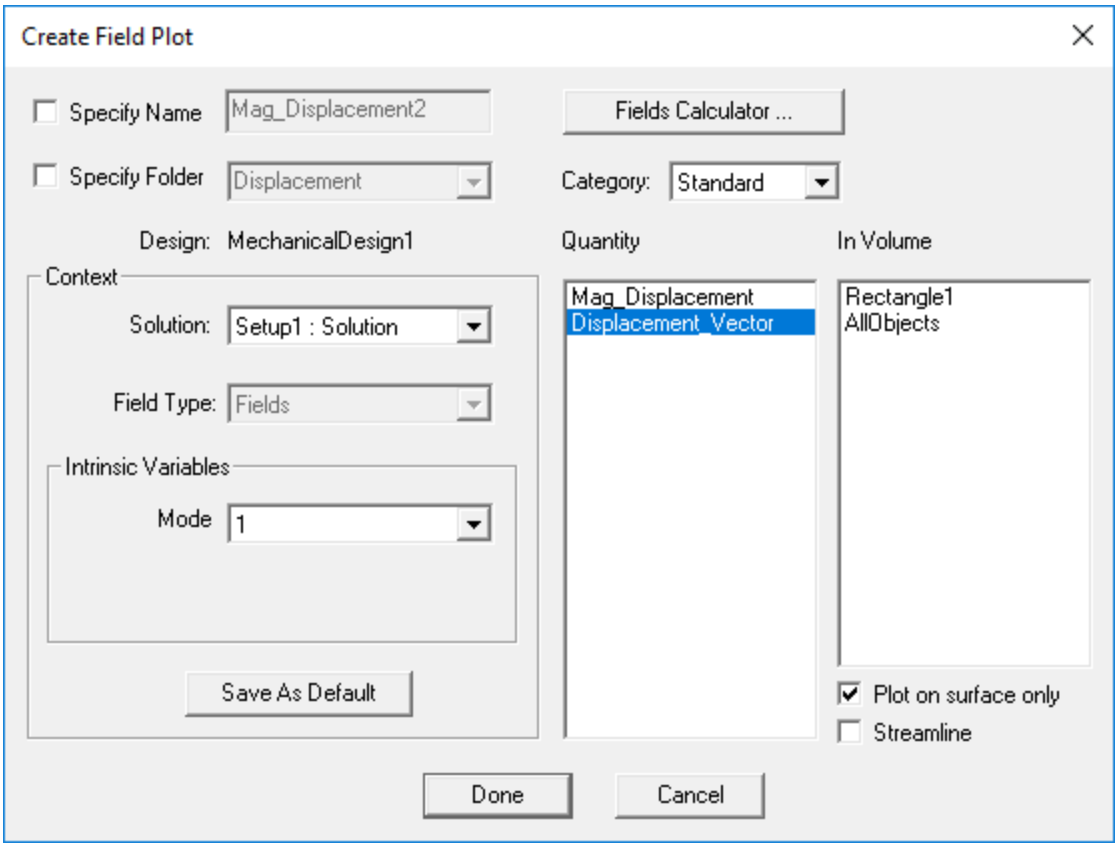


Plot on surface only option selected

Plot on surface only option not selected

   The following field plot setup example shows the displacement magnitude for Mode 1 and plots only on the surface of the selected object or objects. These settings would produce an overlay similar to the left side of the preceding image.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

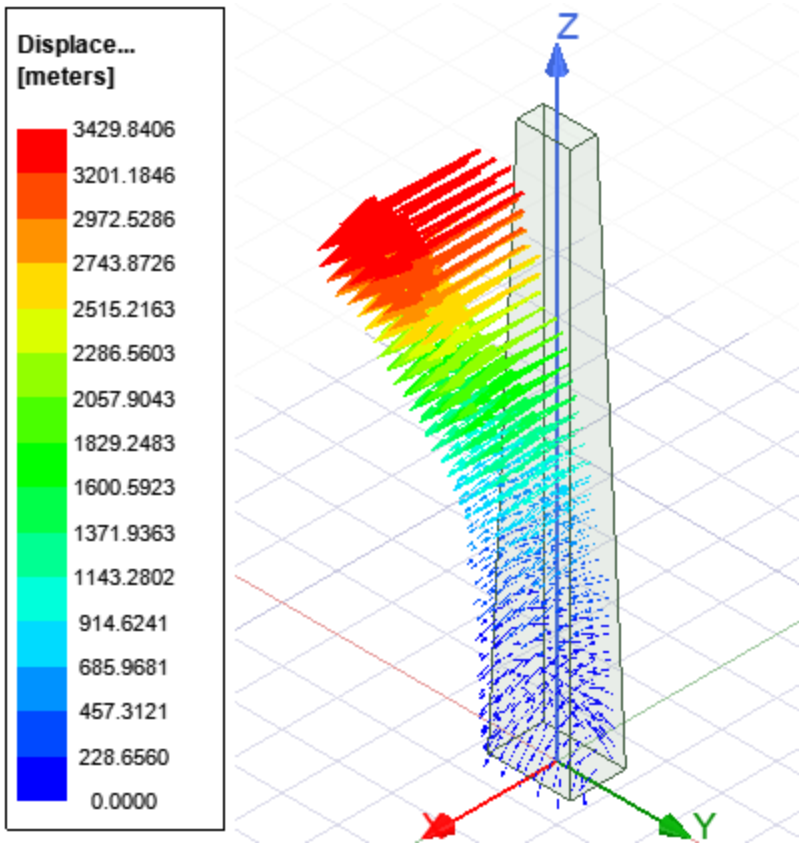


5. Click **Done**.

   The overlay and displacement legend appear on the model in the *Modeler* window.

   The following image is an example of a *Displacement_Vector* overlay:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optionally, under **Field Overlays** > **Displacement** in the Project Manager, right-click **Mag_Displacement*x*** or **Displacement_Vector*x*** and choose **Animate** from the shortcut menu.

   The *Create Animation Setup* dialog box appears.

7. Optionally, adjust the number of animation **Steps**. A higher number of steps produces a smoother animation.

   > **Note:**
   >
   > The default Sweep type setting (Harmonic) is best suited for modal displacement animations as it most faithfully demonstrates the complete 360° vibration cycle. For example, the maximum deflection in one direction progressively decreases, crosses through zero, and then continues until the same maximum magnitude is achieved in the opposite direction. Then, the animation continues to show the reverse progression back to the original calculated displacement.

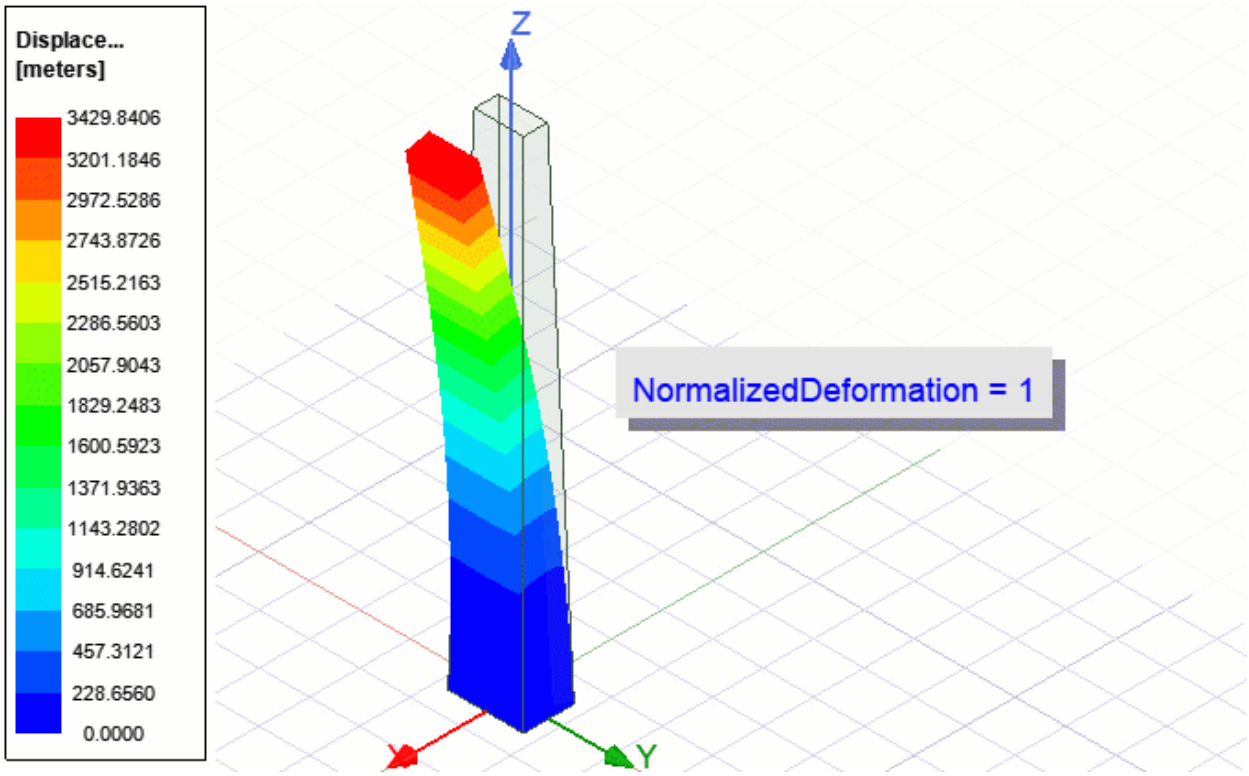
8. Click **OK**.

   The *Animation* dialog box appears and the displacement results are animated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Use the animation controls to stop, restart, reverse, increase/decrease the speed, or to go to a particular animation frame.

   The following figure is the animated *Mag_Displacement* overlay:



10. Click **Close** when you're done viewing the animation.
11. Optionally, to display the deformed shape for a different mode, select **Mag_ Displacement*x*** or **Displacement_Vector*x*** under **Field Overlays** > **Displacement** in the Project Manager.

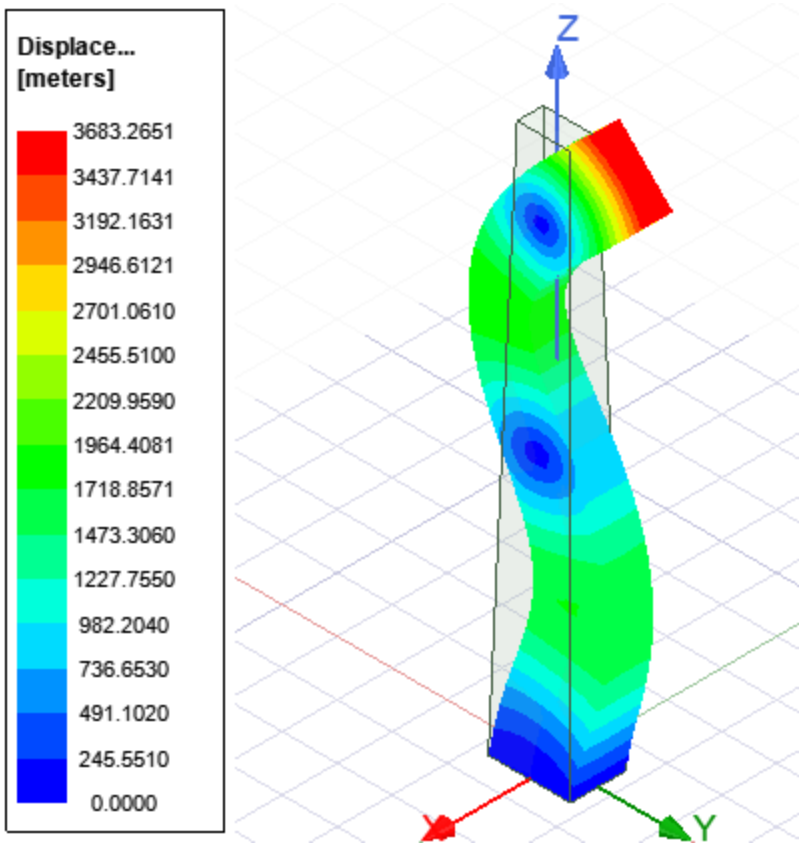
    The overlay properties appear in the docked *Properties* window.

12. Change the **Mode** number and press **Enter**.

    The displacement overlay is updated to display the selected mode.

    The following image shows the displacement magnitude for the eighth mode of the sample model:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7 - Thermal Solutions

This section contains information regarding the material assignment, setup options, and results that are available for a *Mechanical* design, *Thermal* solution. There are two thermal solution types:

- **Steady-State Thermal:** Determines the temperature and heat flux distributions when a design with static boundaries and excitations reaches a state of equilibrium. In this state, the sum of heat in and out of all objects equals zero, and the temperature and heat flux results remain static. This solution type does not determine how long it takes for the design to reach steady-state conditions.
- **Transient Thermal:** This solution type determines how heat flux and temperature results vary over a defined period of time. As such, it solves the design at multiple time steps, as specified in the solution setup. Certain boundaries and excitations can vary over time for this solution type.

Organization of the Thermal Help Topics:

- Help for features unique to the **transient** solution type is located in the Transient Thermal Solutions subbranch.
- All remaining topics within the top-level *Thermal Solutions* folder are applicable to both steady-state and transient solution types, unless otherwise noted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The following requirements apply for both steady-state and transient thermal solutions, unless otherwise indicated:

- The *Thermal Conductivity* must be defined for all assigned materials. Not all library materials include thermal properties.

  For *Transient* thermal analyses, the *Mass Density* and *Specific Heat* must also be defined for all assigned materials.

- There must be a source of heat input (either a *Heat Generation, Heat Flux, or EM Loss* excitation; or a *Temperature* or *Convection* boundary).
- There must be a place for heat to escape (either a *Temperature* or *Convection* boundary). Additionally, for rotating machine models, a *Rotating Fluid* boundary is needed to provide heat flow across the gap between the rotating and stationary objects.
- Objects (whether solid, liquid, or gaseous) must be in contact with each other for heat to flow from one object to the next.
- By default, contact is automatically generated between adjacent objects and has zero resistance to the flow of heat. Assign Thermal Contact manually to impose thermal resistance between contacting parts.

Transient Thermal Solutions

A transient thermal solution determines the distribution of temperatures and heat fluxes at multiple time steps (as the model heats or cools due to assigned thermal sources and loads). The boundaries, excitations, and ambient temperature, which represent the thermal sources and loads, can either be static or can vary with time. Results can be reported, and field overlays can be animated, as a function of time.

All boundaries, excitations, contact, and results available for steady-state thermal analyses are also available for transient thermal analyses, plus the following additional capabilities:

- For **Convection** boundaries:
  - Time-dependent **Film Coefficient**
  - Time-dependent **Ambient Temperature**
- Time-dependent **Temperature** boundaries
- Time-dependent internal **Heat Generation** excitations
- Time-dependent **Heat Flux** excitations
- **Initial Temperature** (object-based assignment)
- **Thermal Monitor** to report temperature vs. time at specified points

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information about how to define time-dependent boundaries and excitations, see Using Datasets to Define Transient Parameters.

For a workflow video of the transient thermal analysis process using a trace mapping example, see Video: Transient Thermal Analysis with Trace Mapping.

For information about performing transient thermal restart analyses (continuing from the results of a previous thermal solution), see Transient Restart Analyses.

> **Note:**
>
> You can import **EM Loss** excitations from a transient source solution. However, the loss **is *not* time-varying** in the target design solution. An average EM power loss is calculated, applied as a step load at the start of the target transient thermal solution, and maintained at a constant value throughout the analysis duration.

Video: Transient Thermal Analysis with Trace Mapping

This video demonstrates the workflow for setting up a mechanical – transient thermal solution using a trace mapping example. A printed circuit board is included as a layout component. A combination of constant and time-varying thermal excitations are assigned to the components. Thermal monitor points are assigned to report temperature versus time results at points of interest, namely the centroids of the electronic component bodies.

Fields overlays are shown, including the temperatures of all objects at the final timestep and the thermal conductivities in the X and Y directions that were mapped to the signal layer of the circuit board. The duration of the video is 10 minutes : 52 seconds.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Video Narration Script (click to view)

0:16 In this video, we will look at the workflow for setting up, solving, and evaluating the results of a mechanical – transient thermal analysis in the Ansys Electronics Desktop application. The model is a trace mapping example consisting of a small printed circuit board (or PCB) and five components.

0:33 The PCB will be included as a Layout Component, based on an HFSS 3D Layout design. More on that process later.

0:42 To save time, the discrete components are already present, and the boundaries, excitations, and thermal monitors have been assigned.

0:49 Each component, except the pin header, is a source of heat. For two of them, U1 and R1, the heat generation is constant. For Q1 and R2, the heat is generated intermittently at an 80% duty cycle.

1:03 Convection is assigned to all exposed faces of the electronic components, excluding the bottom portion of the pin header, which will be inserted into a socket in operation and therefore not exposed to ambient air.

1:15 All component faces except the ones in contact with the PCB were selected. The PCB contact faces are excluded to prevent overlapping convection boundary assignments, since we will assign convection to the top and bottom PCB faces separately, as part of the Layout Component definition.

1:32 The solver will automatically exclude body-to-body contact areas from participating in convective heat transfer. Specifically, the areas of the PCB that are in contact with the pin header and component terminals will be excluded along with the faces in component assemblies where the bodies and terminals meet.

1:50 In the boundary definition, the global ambient temperature variable is specified, equal to 20° Celsius by default, and the film coefficient is 20 watts per meter squared Kelvin.

2:00 Now, let's look at the constant Heat Generation excitations. The total power dissipation for U1 is 40 mW, and it is 25 mW for R1.

2:12 Prior to assigning heat generation to Q1 and R2, datasets were defined to represent the 80% duty cycle. Both datasets start at zero power but go immediately to full power at 0.01 seconds, sustain full power to 16 seconds, drop back to zero at 16.01 seconds, and stay at zero to 20 seconds.

2:33 16 divided by 20 equals 0.8, or 80% on and 20% off. The Q1_ and R2_Dissipation datasets are identical except for the magnitude of the power when on, specifically 50 and 25 mW, respectively.

2:49 Both datasets are referenced using a piecewise linear, periodic function, seen here

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as pwl_periodic, expressed as a function of the dataset name and Time. Because we're using a periodic function the on/off cycle will be repeated every 20 seconds throughout the event duration.

3:08 A thermal monitor point has been assigned to each of the four electronic component bodies. Resultant temperature versus time will be reported at the centroid of each of these bodies.

3:20 Let's look at the HFSS 3D Layout design that will be the source of the layout component representing the PCB in our mechanical-thermal solution. It is a single-sided PCB consisting of two layers. The top is the signal layer, and the bottom is the dielectric layer. Additionally, there is a rectangular polygon drawn in the **Outline** layer, indicating the extents of the PCB. All other layers are empty and unused for this design.

3:47 For convenient portability, this design was exported as a layout component.

3:55 To add the PCB to the mechanical-thermal design, right-click **3D Components** in the Project Manager, access the **Browse Layout Component** flyout menu, and choose **Import .aedbcomp File**.

4:07 Alternatively, you could choose an **aedb folder** (which is the database folder for HFSS 3D layout designs) or an existing layout component, if any has already been added to the project.

4:28 We will place the component according to the Global coordinate system.

4:32 In the **Boundary** tab, choose **Convection** for both the **Top** and **Bottom** faces, specify a **Film Coefficient** of **20 watts per meter squared Kelvin**, and keep **Temperature** equal to the global **AmbientTemp** variable.

4:45 We will **not** consider **Radiation** effects for this solution but, if enabled, you would also specify the **Emissivity**, **View Factor**, and **Temperature** of the surrounding environment.

4:56 In the **Board** tab, specify the resolution of the grid into which the component will be divided for trace mapping purposes. The grid cells will be used to store metal fraction data. For good results, the size of the grid cells should be no greater than half of the narrowest trace width.

5:12 The slider specifies the resolution for the smaller of the board length or width dimensions. For this design, one-one hundredth of the narrower board dimension is 3.03 mils, which is less than half of the narrowest trace width of 7.5 mils.

5:29 The resolution of the longer dimension is proportional to the narrow resolution based on the length/width ratio of the layout. The resolution along the longer dimension of this board will be 163.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5:41 Next, choose **Polygon** for the **Extents** option, since the source design includes an outline rectangle.

5:47 Under **Variable Mapping**, there are no variables in the source layout design that are applicable to a mechanical–thermal solution. If there were, we could choose to map them to variable instance parameters in the mechanical design.

6:00 Since there are no variables to map, there can be no entries under the Parameters tab. If there were source variables and you mapped them to instance parameters, you could also choose to create local design or project variables here.

6:15 Click **OK** to insert the layout component.

6:18 Let’s customize the appearance of the PCB. First, we’ll change the color of the top and bottom layers from the default red color to dark green. The 80% transparency is fine.

6:29 Next, we’ll clear the **Wireframe** option for only the dielectric layer so that it is rendered with shaded geometry.

6:37 Note that, while the conductors are visualized as solid geometry, they are not directly included in the solution. Rather, the entire signal layer is initially represented as only the dielectric fill material.

6:50 A metal fraction map is derived from the layout of the traces and vias (though this single-sided PCB has no vias). A metal fraction is determined for each cell in the trace mapping grid. Then, the cell metal fractions are mapped to the elements comprising the mesh by changing the material properties of the individual elements. As such, the trace mapping results are mesh sensitive.

7:13 We've already specified a sufficiently fine grid as part of the layout component definition. We will now assign a length-based mesh operation inside the layers of the board to ensure that the element lengths are no more than 4 times the grid cell width. Specifically, we’ll set the maximum length to 10 mils, which is 3.3 times the grid cell height or width.

7:35 Because the header pins intersect the PCB layers, we must enable material overrides in the design settings to resolve the overlapping materials automatically.

7:49 Before solving, let’s review the transient solution setup. The **Start** time is **0 seconds**, the **Stop** time is **156 seconds**, and the **Time Step** is **4 second**s, resulting in 39 time steps. This duration is sufficient for quasi-steady-state results to be observed for this design.

8:08 Click **Time variation preview** to see a graph with two curves, one for the Q1 heat generation and one for the R2 heat generation. The periodic extrapolation of the datasets is clearly shown, as we can see the cycles repeated throughout the event.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8:24 156 seconds consists of 7 full power cycles plus an additional 16 seconds or 7.8 cycles total. The simulation ends just before a 4-second OFF period of Q1 and R2 would have begun, so peak temperature fields will be captured at the end of the transient event.

8:45 In the **Save Fields** tab, **None** is selected by default, which means that field results will be saved only for the first and last time step. Optionally, you can specify the save-interval numerically, such as **3** to save fields at every third time step. You can also use a dataset to specify different save intervals for different times.

9:06 We will keep the default option, which reduces storage requirements and solution time. Rather than plotting temperature versus time from global fields results, we will use the Thermal Monitor to track the temperature variations only at specific points of interest.

9:22 We're ready to analyze the design, which takes a little less than 45 minutes to complete on a current, high-end computer workstation. You can observe the progress of the thermal monitors plot while solving.

9:38 Here, it is shown in time-lapse captures.

9:41 Often, as observed here, the global maximum temperature trace corresponds to one of the other specified monitor points.

9:48 Q1 is the hottest object, R2 the second hottest, R1 the third hottest, and U1 the coolest.

9:56 Even though power dissipations at U1 and R1 are constant, these component temperatures are affected by the change in power dissipation at Q1 and R2. During the cooling intervals, R1 momentarily becomes the hottest object, and Q1 the coolest.

10:13 The minimum and maximum temperature points are consistent for the last three cycles, indicating that we have achieved quasi-steady state results. In other words, the average heat generation and the average heat dissipated via convection have reached equilibrium.

10:30 We can look at an overlay of the temperature results for the last time step.

10:47 Finally, you can display the thermal conductivity values that have been mapped to the elements comprising the layout component mesh to represent the effects of the conductors.

10:57 Here we see the conductivity in the x direction.

11:07 And here we see the conductivity in the Y direction.

11:11 We hope you've found this video helpful in understanding the workflow for transient

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

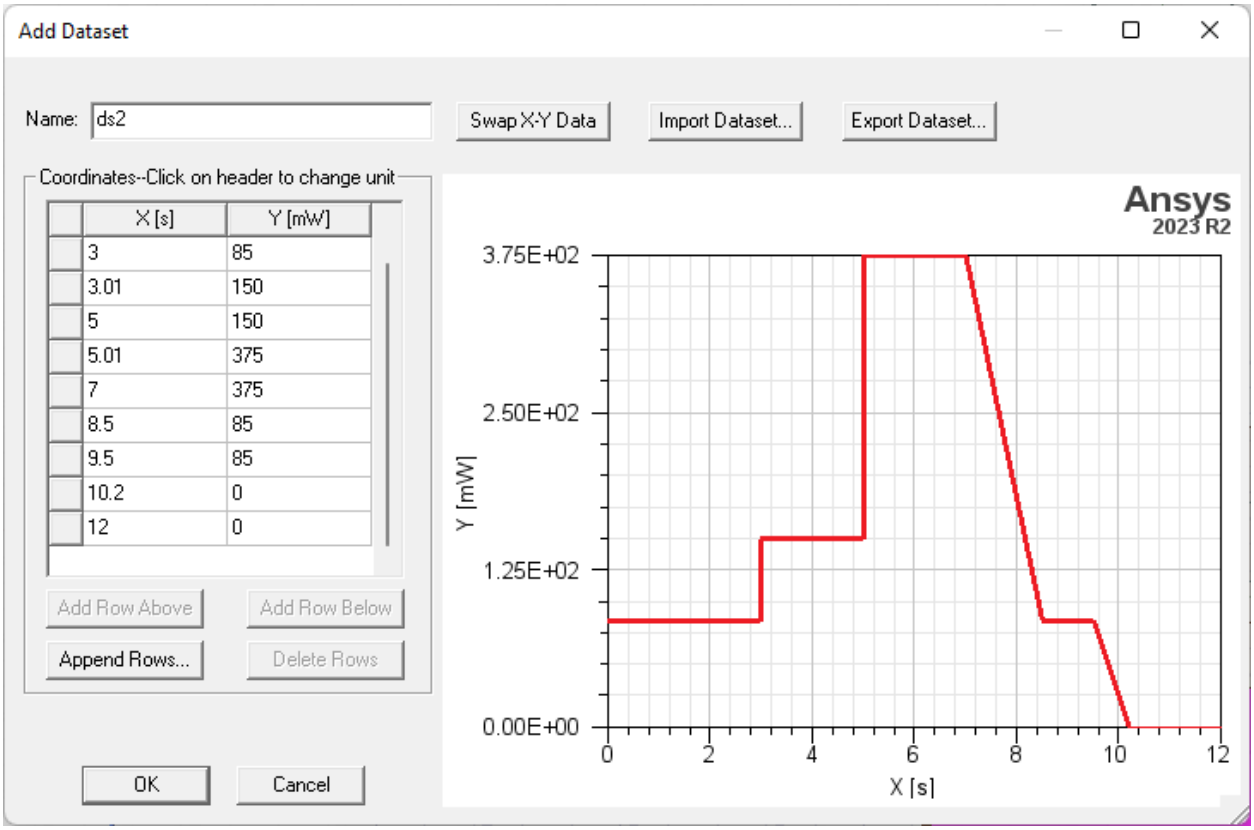
thermal solutions and for trace mapping using layout components. Thank you for watching.

Using Datasets to Define Transient Parameters

Time-varying parameters are defined using piecewise linear (pwl or pwl_periodic) or piecewise constant (pwc) datasets, with Time as the X-coordinate and the appropriate parameter and units as the Y-value. You must define a **Design Dataset**; *Project Datasets* are not currently supported for this purpose.

You can predefine the design dataset before referencing it in a boundary, excitation, or solution setup dialog box. Alternatively, you can enter the type, name, and X-dimension of a new dataset, such as **pwl(ds1,Time)** or **pwc(ds1,Time)**, into a supported input field (text box) of a boundary, excitation, or solution setup dialog box. If you choose the latter method, the *Add Dataset* dialog box appears automatically when you press **Enter** or click **OK** to complete the assignment. In either case, the process of defining the time-varying values is the same.

Example – Dataset for Time-Varying Heat Generation:



This example shows a pwl curve of power in milliwatts (mW) versus time is seconds (s). There is a heat generation of 85mW beginning at time=0, a quasi-step increase to 150mW at 3.01s, and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a quasi-step increase to 375mW at 5.01s. The heat generation decreases linearly from 375mW to 85mW over a 1s period beginning at 7s and then remains constant for a short time. At 9.5s, the power decreases linearly to 0mW over a 0.7s period and remains zero until time = 12s.

Note:

- For solution timesteps lying between points defined in the design dataset, the Y values are interpolated.
- For solution timesteps that are at a time beyond the last time value in the dataset, the behavior depends on the function used:
  - **pwl**: The last dataset value is used for all remaining timesteps. Downstream Y values are *not* linearly extrapolated from the curve trend.
  - **pwl_periodic**: Periodic extrapolation occurs. That is, the pattern of Y values is repeated in the same sequence for solution times beyond the last time value in the dataset. Use this function when you wish to define a repeating pattern, such as a set of intermittent power levels.

    For example, assume there is a set of values, Y_1 through Y_{10} defined for X = 0 through 10 seconds, respectively. For Time = 10 through 20 seconds, the corresponding Y values will be a repetition of Y_1 through Y_{10}. The same is true for Time = 20 through 30 seconds and so forth.

    > **Important:**
    >
    > For a dataset to be properly recognized as periodic, and the extrapolation performed accordingly, the first and last Y values must be identical. The equality of these two values ensures that there are no discontinuities in the resulting data.

- If the first value in the dataset is specified for a time later than the first solution timestep: The first Y value in the dataset is used for all timesteps preceding the first X (time) value in the dataset. Upstream extrapolation is *not* performed based on the curve trend.
- **Best Practice:** To avoid confusion and unexpected thermal source or load magnitudes, define the datasets to fully cover the range of time used for the solution setup. This practice is not a requirement, but exercise caution when forcing the solver to use assumed constant values beyond the range of a dataset. The exception is when you are using the pwl_periodic function, and the expected periodic extrapolated data is clearly known.
- See Transient Thermal Solutions for a list of boundary and excitation parameters that support time-varying values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Create and Use a Design Dataset

This procedure shows the creation of a **piecewise linear** (pwl or pwl_periodic) dataset, suitable for time-varying parameters such as temperatures, convection film coefficients, or power (as in heat flux or heat generation excitations).

Additionally, see Piecewise **Constant** (pwc) Datasets, which you can use in transient thermal solution setups to time-vary the *Time Step* parameter.

1. Access the *Add Dataset* dialog box using one of the following two methods ("a" or "b"):
   a. Before beginning to assign a boundary or excitation:
      i. From the menu bar, click **Mechanical** > **Design Datasets**.

         The *Datasets* dialog box appears.

      ii. Click **Add**.

         The *Add Dataset* dialog box appears

      iii. Continue to step 2.
   b. While assigning a boundary or excitation:
      i. Type a dataset reference into a supported parameter text box using the following syntax:

         pwl(<*dataset_name*>,Time) or **pwl_periodic(<*dataset_name*>,Time)**

         where **pwl** indicates the piecewise linear dataset type, **pwl_periodic** indicates piecewise linear with periodic extrapolation, and **Time** indicates the X-dimension type.

         The following table show examples of acceptable dataset names and how they would be referenced within a boundary or excitation dialog box:

Design Dataset Name Examples:

Dataset Name	How to Reference it from a Boundary or Excitation Dialog Box
ds1	pwl(ds1,Time)
ConvectionFilmCoef	pwl(ConvectionFilmCoef, Time)
HeatGen2	pwl_periodic(HeatGen2, Time)

      ii. Press **Enter** or click **OK**.

         The *Add Dataset* dialog box appears with the **Name** parameter and column

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data types already specified along with the default units.

iii. Continue to step 3.

2. Set up the datasheet parameters as follows:
   a. Specify the desired dataset **Name**.

      Refer to the preceding table for typical name examples.

   b. Click the **X-column header** and choose **Time** from the available choices that appear.
   c. Optionally, click the **X-column header** again, to change the units of time, and select the desired units.
   d. Click the **Y-column header** and choose the desired data type from the available choices that appear.

      Suitable data types for transient thermal analyses are as follows:

      - Power – for Heat Generation or Heat Flux (with Total Power option selected)
      - Irradiance – for Heat Flux (with Surface Flux option selected)
      - Temperature – for Temperature boundaries or for time-varying Ambient Temperature in a Convection boundary
      - Thermal Convection - for the Film Coefficient of a Convection boundary

> **Note:**
>
> If you accidentally select the wrong data type for either the X or Y column, click the column header again and choose **No unit**. This step will clear the units list and restore the list of data types. Then, select the desired data type.

3. Optionally, click the **X-column header** to change the units of time, and select the desired units.
4. Optionally, click the **Y-column header** to change the units of the Y-value, and select the desired units.

   The available units depend on the specified data type.

5. Fill in the individual X and Y values to define how the Y-quantity varies with time.

   Verify that the appearance of the piecewise linear plot is as expected.

6. Click **OK** to close the *Add Dataset* dialog box.
7. If, in step 1, you chose to create the dataset before starting to assign a boundary or excitation, you will be returned to the *Datasets* dialog box at this point. If so, click **Done** to close the *Datasets* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Piecewise Constant (pwc) Datasets

For Transient Thermal solutions, you can use **piecewise constant** (pwc) datasets to vary the **Time Step** parameter or the interval for saving fields data in the *Thermal Solve Setup* dialog box. You can also use pwc datasets to specify time-dependent parameters in certain boundaries and excitations. Where supported, the capability is mentioned in the individual boundary or excitation help topics.

> **Tip:**
>
> While pwc data may technically be supported for non-transient parameters (such as temperature-dependent material properties or temperature-dependent convection film coefficients), its usage in these contexts is *not* recommended. Step changes in a parameter can lead to solution instability and difficulty in convergence.
>
> For example: If a temperature result is very close to where a step change occurs, the solution can oscillate between the values on each side of the step and fail to converge.
>
> This phenomenon is not a problem for transient parameters because the time always progresses in the forward directon. Once a step change occurs, the time-dependent parameter remains stable at the post-step value.

The process of defining a piecewise constant dataset is the same as defining a piecewise linear (pwl) dataset, which is covered in the topic Using Datasets to Define Transient Parameters.

The characteristics of pwc data are:

- For each datapoint interval (that is, between each pair of XY values), the slope of the curve is zero.
- All changes in the Y value are step changes.

> **Important:**
>
> The *Add Dataset, Edit Dataset,* and *Design Settings* dialog boxes are not aware of whether a dataset will be used to define piecewise linear (pwl) or piecewise constant (pwc) parameters. These dialog boxes will always interpret and display the data points as a piecewise **linear** curve. In fact, the same datset could be referenced in both ways for different parameters: *pwl(ds1,Time)* for one parameter; *pwc(ds1,Time) for another*.

Consider the following example to understand the distinction between the raw dataset and the interpreted pwc curve.

Example – Comparison of Dataset Definition and PWC Interpretation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

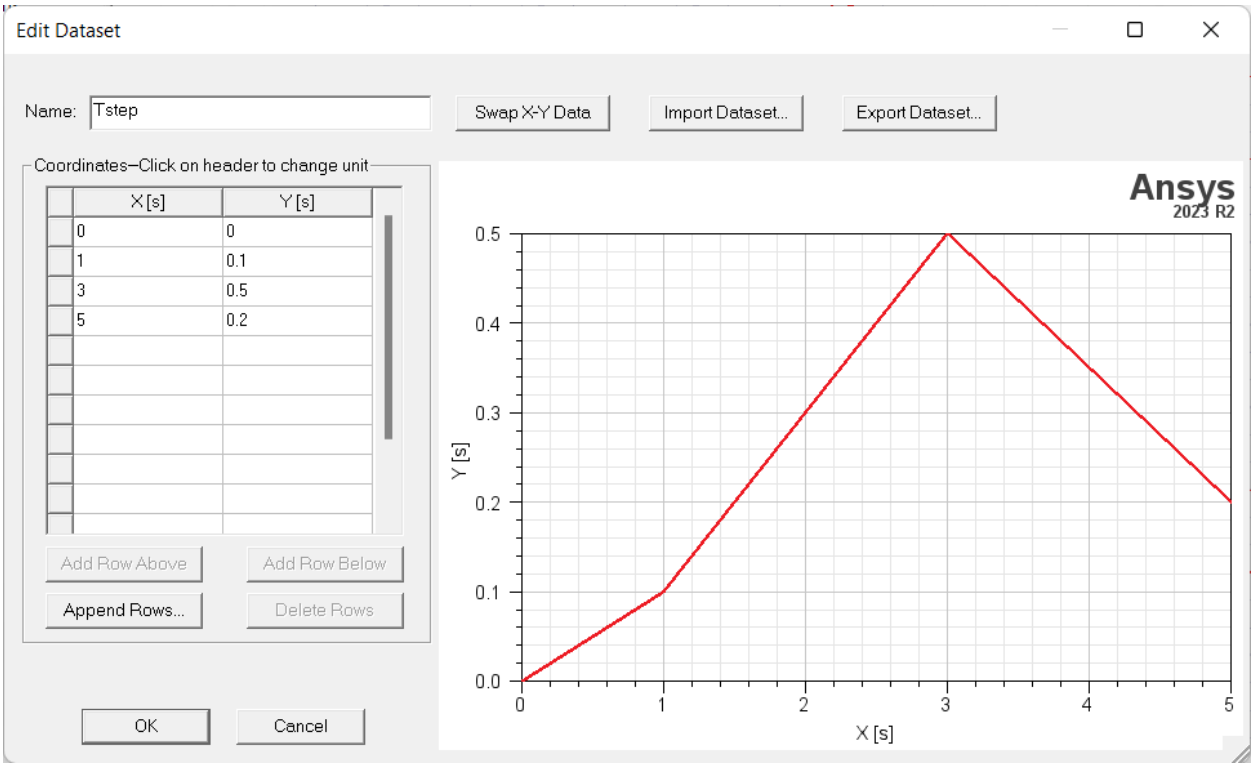


Figure 1: Dataset (Tstep) Definition

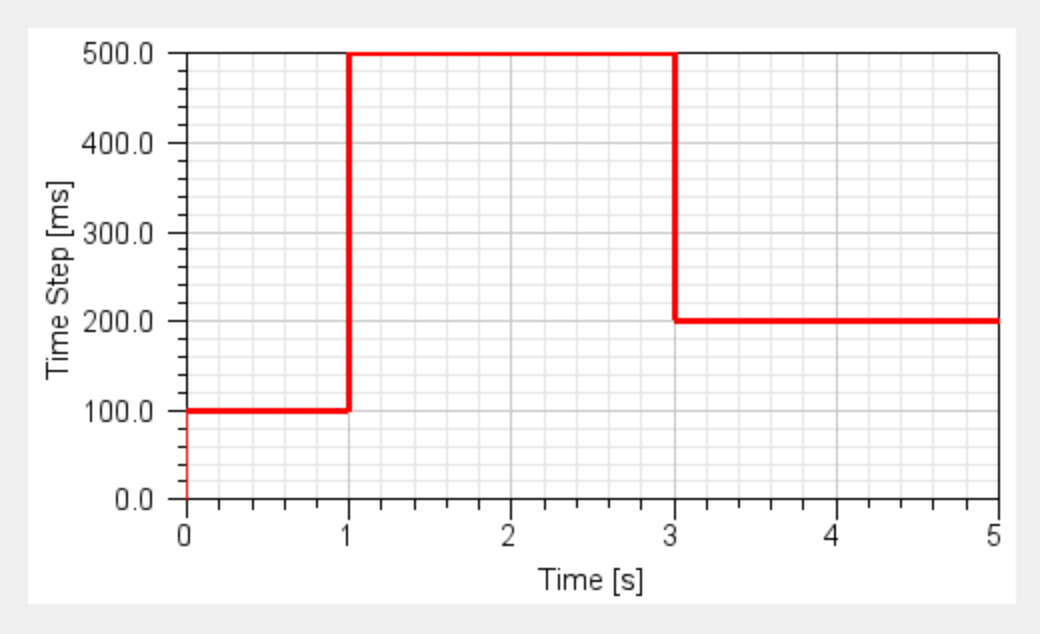


Figure 2: Dataset Interpretation as a PWC *Time Step* control – *pwc(Tstep, Time)*

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

For each interval (or each consecutive pair of X and Y values), the constant Y value is dictated by the **second** point of the pair, *not* the first one:

Interval Number	Datapoint Numbers	Specified Y Values (s)	Interpreted Constant Time Step for Interval (s)
1	1 2	0 0.1	**0.1**
2	2 3	0.1 0.5	**0.5**
3	3 4	0.5 0.2	**0.2**

Thermal Monitor

Prior to solving a *Transient Thermal* analysis, you can assign points at which the temperature result will be monitored. After solving, you can plot the temperature versus time results for all user-assigned points. Additionally, the global maximum temperature versus time is plotted for all transient thermal solutions.

Points are defined by selecting vertices, a drawn point, linear edges, polyline objects, planar faces, planar objects, or solid objects. For each entity type, the monitor point is placed as follows:

- **Vertices:** The monitor points are located at each selected vertex.
- **Drawn Points:** Select drawn points in the History Tree rather than graphically. Monitor points are located at the coordinates of each selected point.
- **Linear Edges:** Monitor points are located at the midpoint of each selected linear (straight) edge. Curved edges are not supported.
- **Multisegment Open Polyline Objects:** Select multisegment polylines as objects or select them from the History Tree. Monitor points are located at the centroid of the combined set of line segments comprising each selected polyline object (see Note below).
- **Planar Objects** or **Planar Faces**: Monitor points are located at the centroid of each selected planar object or face (see Note below). Curved faces are not supported.
- **Solid Objects:** Monitor points are located at the centroid of each selected 3D object (see Note below).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> For curved faces, faces with holes or cutouts, and hollow solids: The centroid may lay outside of the selected face or object. The point may also be outside of any model geometry, where no thermal result exists. Additionally, for multisegment polylines, there is a high probability that the centroid will not lie on one of the line segments.
>
> Carefully verify the monitor point location in such cases. When you select a point under *Monitor* in the Project Manager, its coordinates appear in the docked *Properties* window. However, you cannot edit the coordinates. To correct a point, you either have to reassign it to a new selection or delete and recreate it.

Once monitor points have been assigned and the analysis solved, the plotted results appear in the *Thermal Monitor* tab of the *Solutions* dialog box. Access it from the **Simulation** ribbon tab (**Solution Data** command) or from the solution setup's shortcut menu (**Thermal Monitor** command).

To Assign Thermal Monitor Points

1. From the **Select** drop-down menu on the **Draw** or **Model** ribbon tab, choose the desired selection mode (**Object, Face, Edge, Vertex,** or **Multi**).

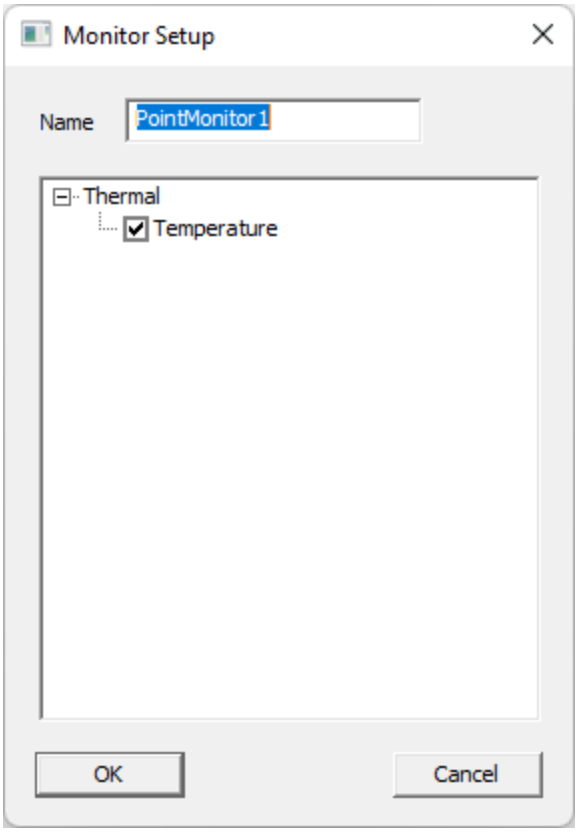
   Alternatively, click in the *Modeler* window to make it active and press **O, F, E, V,** or **M** to choose the selection mode.

2. Click the desired entity whose centroid is where you want to monitor the thermal results.

   > **Note:**
   >
   > You can select multiple entities and assign thermal monitor points to each of them in a single operation, providing you do not wish to customize the names of each monitor point. If you wish to specify custom names, assign the monitor points one at a time.

3. Access the *Monitor Setup* dialog box using *one* of the following three methods:
   - From the menu bar, click **Mechanical** > **Monitor** > **Assign** > **Point**.
   - Right-click in the *Modeler* window and choose **Assign** > **Point** from the shortcut menu.
   - In the Project Manager, right-click **Monitor** and choose **Assign** > **Point**.

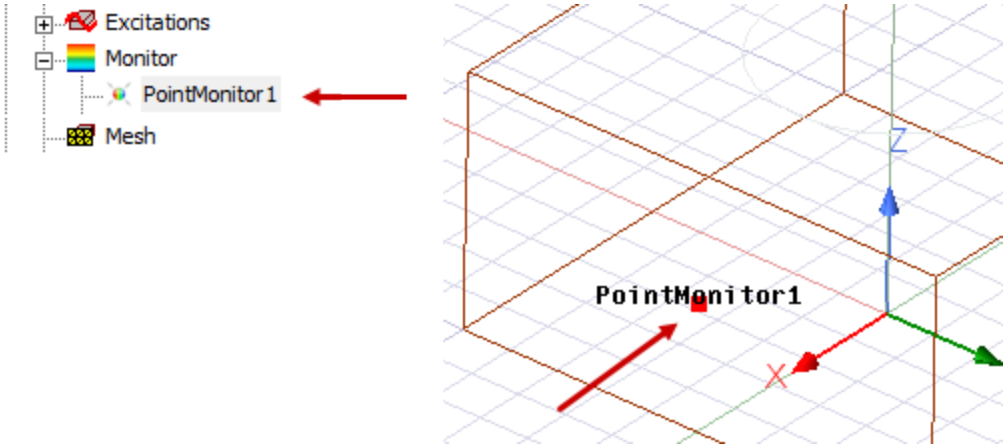
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, specify a custom **Name** for the monitor point.

   The default name is PointMonitor*x*, where "*x*" is an automatically incremented serial number.

5. Click **OK** to assign the point and close the dialog box.
6. Optionally, to visualize the monitor point, select it under *Monitor* in the Project Manager:



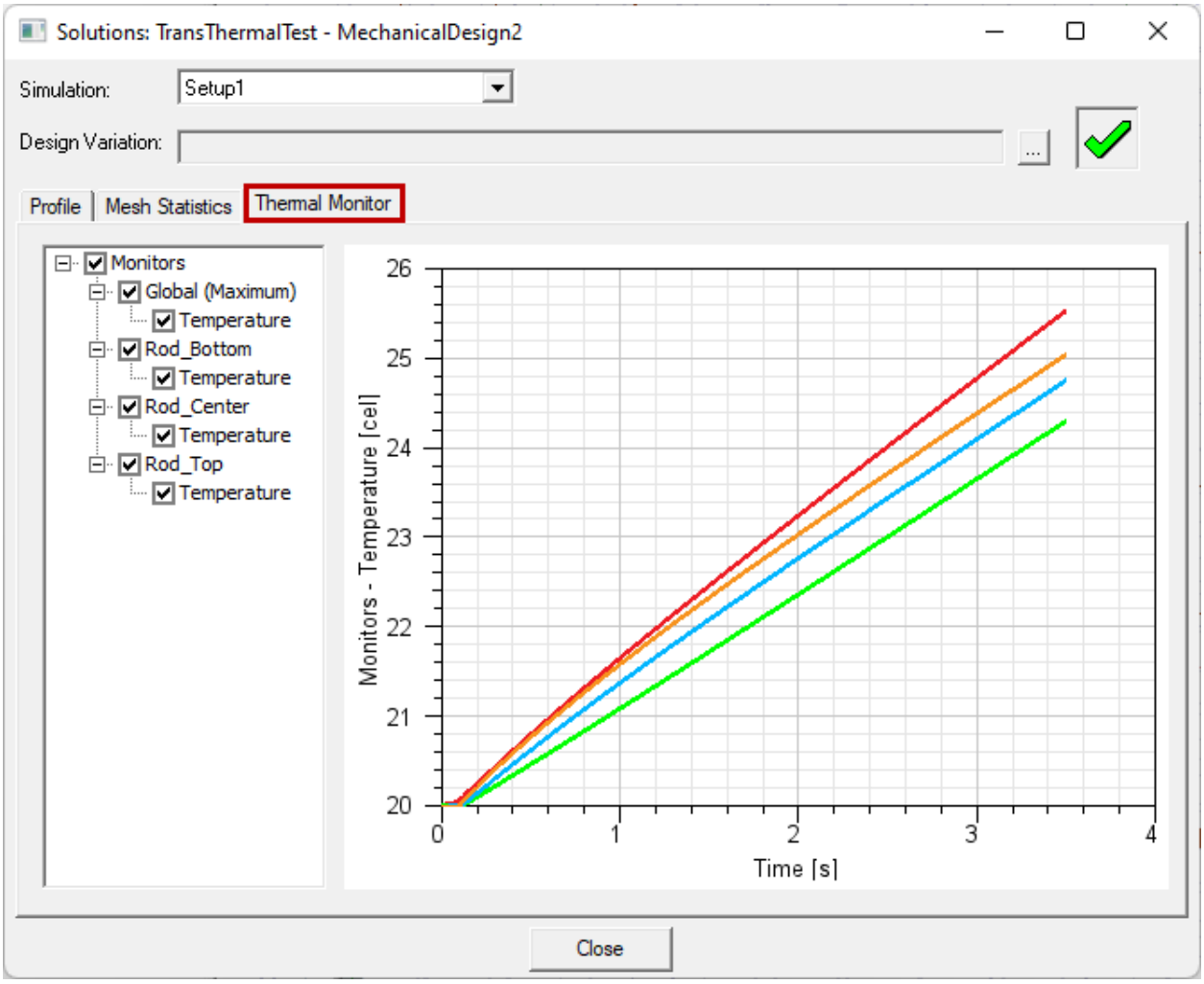
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Display Thermal Monitor Results

After the monitor points have been assigned and the analysis has been started, access the plot of monitor results as follows. The monitor graph will be updated as the transient solution progresses. You need not wait until the solution is complete to observe the temperature:

1. Access the *Thermal Monitor* tab of the *Solutions* dialog box using *one* of the following three methods:
   - Under *Analysis* in the Project Manager, right click the **<*solution_setup_name*>** and choose **Thermal Monitor** from the shortcut menu.
   - On the **Results** ribbon tab, click **Solution Data**. Then, select the **Thermal Monitor** tab.
   - From the menu bar, click **Mechanical** > **Results** > **Solution Data**. Then, click the **Thermal Monitor** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Optionally, click any checkbox to toggle the visibility of individual plot curves. The visibility options are useful for presenting single results or when two curves overlap, making the color ambiguous.
3. Click **Close** when done.

Note:

You can choose to export thermal monitor results to CSV files at the completion of the analysis. See the Export Settings Tab section of the *Design Settings for Mechanical* topic.

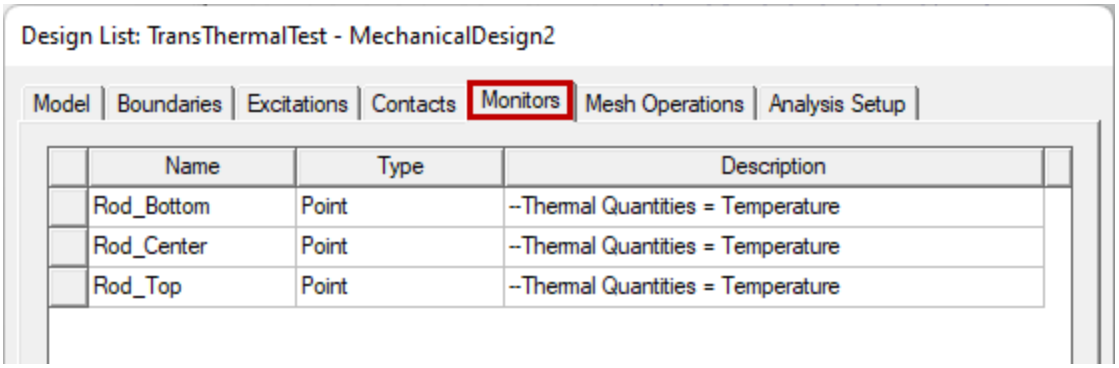
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Available Thermal Monitor Commands

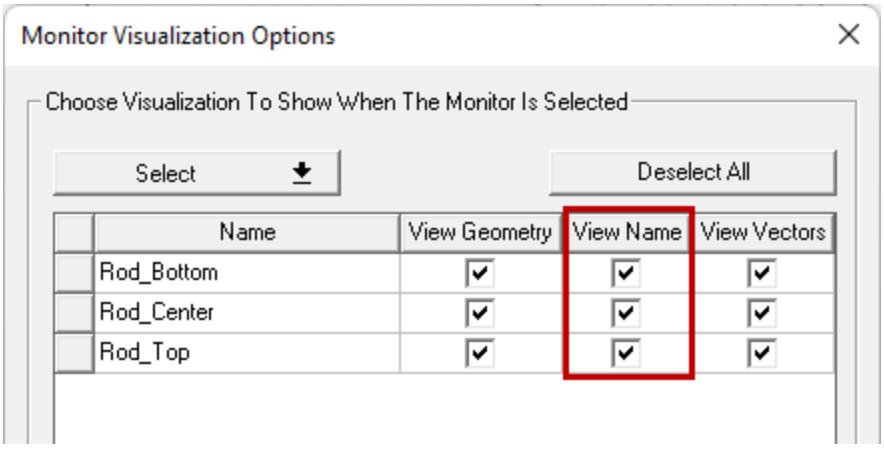
The Monitor shortcut menu in the Project Manage, and the Monitor submenu within the Mechanical menu, provide the following additional commands:

- **List:** Opens the *Design Lists* dialog box with the *Monitors* tab selected:

  This tab lists all user-defined thermal monitor points.



- **Reassign:** Execute this command after selecting an entity of the model to reassign an existing monitor point to a new point. The *Reassign Monitor* dialog box appears, in which you select the previously defined point to reassign. This command is also available when you right-click a point name under *Monitor* in the Project Manager, immediately reassigning the point that was right-clicked. In this case, the *Reassign Monitor* dialog box does not appear.
- **Delete All:** Delete all user-defined thermal monitor points.
- **Visualization:** Opens the *Visualization Options* dialog box:

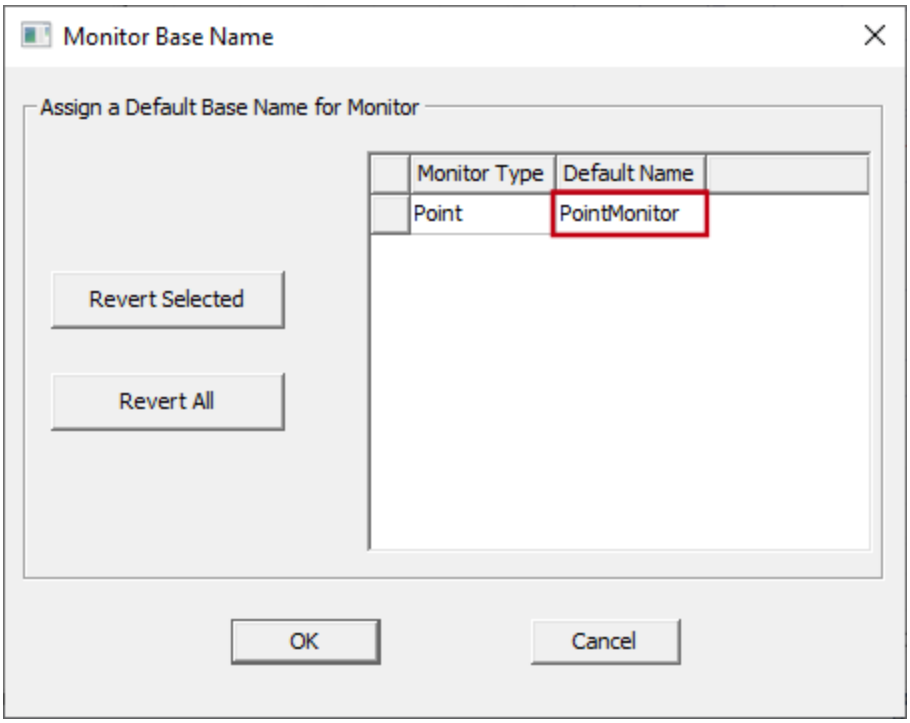


  This dialog box is a common feature shared by *Boundaries, Excitations, Contacts, Initial Temperature,* and the thermal *Monitor*. In the context of thermal monitor points, only the **View Name** options are relevant. *Vectors* and *Geometry* (visualization overlays) are not applicable. Clear a **View Name** checkbox to prevent the name of a thermal monitor point

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from being displayed in the Modeler window when the point is selected in the Project Manager.

- **Set Default Base Name:** Opens the *Monitor Base Name* dialog box:



You can edit the default base name assigned to thermal monitor points in this dialog box.

Assigning Initial Temperatures

For Transient Thermal solutions, you can assign initial temperatures to selected objects. This capability is in addition to the global *Initial Temperature* property, which is defined in the *Initial Conditions* tab of the *Thermal Solve Setup* dialog box. Unlike the object-based assignment discussed in this topic, the global setting is applicable to both steady-state and transient thermal solutions. Object-based *Initial Temperature* assignments override the global setting.

You can assign an **Initial Temperature** to the following items:

- **Solid Objects**
- **Shell Objects** (sheet objects with the **Shell Element** option selected)

Applicable *Solution Type*: **Transient Thermal** only

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

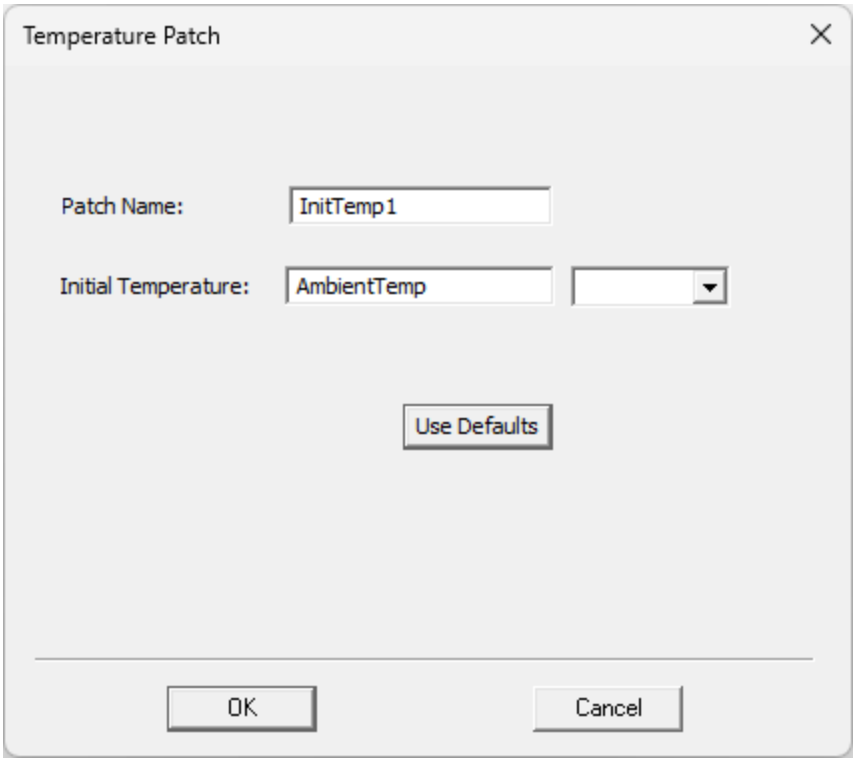
Tip:

Examples of when you might assign *Initial Temperatures*:

- To determine how long it takes for an assembly to cool down from a heated condition, such as post-soldering cool down of a circuit board.
- To reduce simulation times and storage requirements, by starting a transient solution at or near the expected operating temperatures of the components.

To Assign an Initial Temperature:

1. Select one or more solid or shell **objects**.
2. Access the *Initial Temperature* dialog box using *one* of the following three methods:
   - Right-click in the Modeler window and choose **Assign Initial Temperature** from the shortcut menu.
   - In the Project Manager, right-click **Initial Temperature** and choose **Assign**.
   - From the menu bar, click **Mechanical** > **Initial Temperature** > **Assign**.



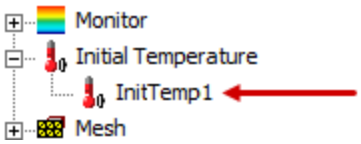
3. Optionally, change the **Patch Name** from the default name (InitTemp*x*).
4. Type the desired **Initial Temperature** value, which can be numeric, a variable, or an expression (consisting of constants, variables, and mathematic operators). Also, choose the temperature units from the adjacent drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default value is the *AmbientTemp* variable specified in the *Design Properties*, which is also the default global initial temperature defined in the *Thermal Solve Setup* dialog box. Therefore, unless you've specified a user-defined global initial temperature, there's no point assigning an object-based initial temperature using the *AmbientTemp* variable. For the most efficient setup, choose a global initial temperatue that suits the majority of objects, minimizing the number of object-based initial temperature assignments that are needed.

5. Click **OK**.

   The assignment appears under *Initial Temperature* in the Project Manager:



Additional *Initial Temperature* Commands:

Aside from **Assign**, the following additional commands are available in the shortcut menu that appears when you right-click **Initial Temperature** in the Project Manager:

- **List:** Open the *Design List* dialog box with the *Initial Temperature* tab selected. This tab lists all initial temperature assignments in the active design, and you can access and edit their properties or delete them from this dialog box.
- **Delete All:** Delete all existing initial temperature assignments in the active design.
- **Visualization:** Access the *Visualization Options* dialog box, which lists all initial temperature assignments and allows you to toggle the **View Geometry** and **View Name** options. These options affect the visualization overlays that appear on the model when an initial temperature assignment is selected in the Project Manager. The options do not affect the model geometry visibility.

  The contents of this dialog box are shared by *Contacts, Boundaries, Excitations, Monitor,* and *Initial Temperature*. For initial temperature assignments, the *View Vectors* option is not applicable and, therefore, toggling this option has no effect on visualization.

- **Set Default Base Name:** Access the *Initial Temperature Base Name* dialog box, where you can redefine the default name for this type assignment. A serial number (starting at 1) is appended to the end of the specified name and is incremented for each subsequent assignment.

Transient Thermal Restart Analyses

It is often desirable to start a transient analysis from the results of a previous steady-state or transient solution. This capability exists for Transient Thermal analyses in the Ansys Electronics Desktop application. There are several workflow scenarios, as summarized in the following list:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Starting from the results of a Steady-State Thermal solution:** For a steady-state-to-transient thermal restart, set up a link to the steady-state solution as part of the *Advanced* analysis setup of the transient solution. The source steady-state temperature distribution becomes the starting condition for the target transient solution.

  See the *Restart Requirements and Limitations* section below and the Restarting from a Steady-State Thermal Solution topic for more details.

- **Transient Thermal-to-Transient Thermal restart (separate designs):** For this scenario, set up a link to a source transient design, either in the same project or a different one. The restarted transient solution begins with the results of the source solution's last completed step.

  See the *Restart Requirements and Limitations* section below and the Restarting from a Separate Transient Thermal Solution topic for more details.

- **Extending the duration of the current Transient solution:** You can increase the value of the **Stop Time** for the current Transient Thermal solution. Use this scenario to continue the existing solution just as if a longer duration had been specified originally.

  See the *Restart Requirements and Limitations* section below and the Extending the Current Transient Thermal Solution topic for more details.

Restart Requirements and Limitations:

- **For *all* Transient Thermal restart scenarios:**
  - The source and target geometry must be identical, including face and object IDs.
  - All boundaries and excitations must remain the same between the source and target designs and have identical assignment sets. Their names, as well as the values of boundary or excitation parameters, can be changed in the target solution, as detailed in the next bullet.

  The best way to ensure that both of these requirements are satisfied is to use the Project Manager to copy the source design and paste it into the target project. Copying the source geometry and using the command, *Modeler > Import from Clipboard*, in the target design duplicates all geometry IDs but does not preserve boundary and excitation IDs. Therefore, this command is *not* sufficient for transient restart analyses.

- **When restarting a steady-state or transient source design in a separate target transient design:**

  You may change the magnitude of boundary or excitation parameters in the target design. For example, you could specify a different power for Heat Flux or Heat Generation excitations. Similarly, you could modify the ambient temperature or film coefficient for convection boundaries.

  The original values continue to apply to the original (source) steady-state or transient solution. The altered boundary or excitation parameter values apply only to the time interval calculated in the target transient solution. When restarting from a steady-state

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source, the altered parameters apply from Start Time = 0 seconds through the specified Stop Time of the transient solution. For a transient-to-transient restart, the Start Time of the target solution is equal to the Stop Time of the previously run source solution. In this case, the altered boundary or excitation parameters only affect the target results solved beyond the Stop Time of the source design. Saved fields copied from the source solution are unaffected.

Also, see Cached Link Data and Solution Invalidation (Transient Thermal Restart).

- **When extending the duration of the current transient solution:**

  > **Important:**
  >
  > For this type of restart, you cannot change the values of boundary or excitation parameters, since editing any boundary or excitation invalidates the current design's results. If you wish to extend the solution and also change the value of a boundary parameter (such as the convection film coefficient) or the value of an excitation parameter (such as the power of a Heat Generation), you must use the *Transient Thermal-to-Transient Thermal restart (separate designs)* method discussed in the previous bullet.

Restarting from a Steady-State Thermal Solution

A steady-state operating condition can take a long time, and many transient solution steps, to achieve. Suppose you want to see the thermal behavior of a design with power or cooling parameters cycling between certain values. There is an initial warm up period before temperature fluctuation patterns become stable or repeatable. This initial warming period will often exceed the time it takes to respond to thermal load changes that occur during operation. Starting a transient simulation with a steady-state temperature distribution that is based on nominal conditions is an efficient way to solve such problems. In this way, you can focus on the area of interest and not expend time and storage determining and saving the results during the initial warm-up period.

When you start a transient thermal analysis from the results of a previously solved steady-state thermal source solution in a separate design, you are free to modify the parameter values of assigned boundaries or excitations in the target design. Since you are not modifying the boundaries or excitations in the source design, the previously solved results are not invalidated. However, you cannot modify the geometry or add, remove, or reassign any target boundary or excitation for a restart analysis to work. One exception is that you can draw points to select for defining Thermal Monitor points in the target design that didn't exist in the source design.

Steady-State-to-Transient Thermal Restart Procedure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following procedure assumes that you have already set up and solved a steady-state thermal analysis of your model using the same boundary and excitation assignments that are needed for the transient solution.

1. Optionally, if you wish to create the transient thermal design in a different project from the one containing the steady-state solution, insert a new project or open the desired existing one.

   Add or open the second project, if desired, in the same Ansys Electronics Desktop session, not in a separate session.

   Otherwise, do nothing and continue to step 2.

2. In the Project Manager, right-click the *source* ***<steady-state_thermal_design_name>*** and choose **Copy** from the shortcut menu.
3. In the Project Manager, right-click the ***<project_name>*** heading of the project to which you want to add the *target* transient thermal design and choose **Paste**.

   In most cases, you will likely want to paste the target design into the same project that contains the source design.

4. Right-click the **just-pasted *target* design** heading and choose **Solution Type**.
   a. In the *Solution Type* dialog box, select **Transient Thermal**.
   b. Click **OK**.
5. Optionally, edit any boundary or excitation in the target transient design for which you want to change the value of a parameter from what was used for the source steady-state solution.

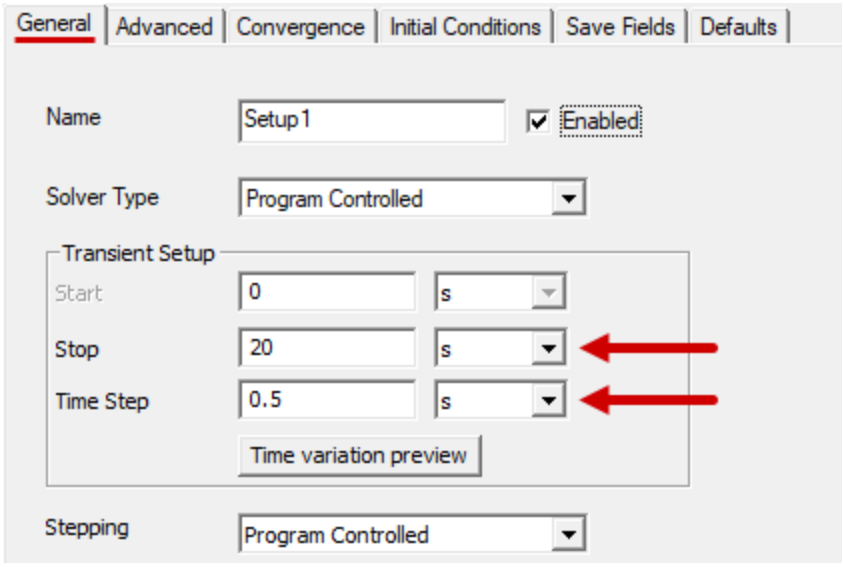
   For example, you might want to change a static *Convection Film Coefficient* value to a pwl (<*dataset*>, Time) function to vary the convective heat transfer over time (representing the effects of a variable speed fan). Similarly, you might want to vary Heat Flux or heat Generation excitations over time.

   > **Important:**
   >
   > You cannot add, remove, or change the assignment set of any boundary or excitation. Doing so will cause the transient analysis to start at the default initial conditions, typically ambient temperature. The results of the source steady-state solution will not be imported if the geometry or boundary/excitation assignments do not remain identical between the two designs. Only changes of boundary/excitation parameter values are permitted.

6. Optionally, define any Thermal Monitor points at which you wish to track the temperature results versus time.
7. Under *{target_design_name}* > *Analysis*, in the Project Manager, double-click the **{analysis_setup_name}** to access the *Thermal Solve Setup* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In the *General* tab, specify the **Stop** and **Time Step** values and their units in the *Transient Setup* section.



b. In the *Advanced* tab, select **Start/Continue from a previously solved setup**.

The *Setup Link* dialog box appears.

c. Follow the instructions on the Setup Link (Transient Thermal Restart) page to complete the required link definition.

Continue from here after completing the link setup:

d. Additional settings in the *Advanced* tab are not applicable to this restart scenario.

> **Note:**
>
> - The *"Import mesh"* option and its associated *"Setup Link"* button are unavailable when you choose the *"Start/Continue from a previously solved setup"* option because this option implicitly imports the mesh as well as the thermal solution results from the source design.
> - The *"Copy fields from source"* option has no effect when restarting from the results of a steady-state thermal solution. The final result of the source design is always copied to the target design, and there is only one result for steady-state analyses.

e. Optionally, customize the options in the *Convergence* or *Save Fields* tabs.

f. Click **OK** to accept the solution setup.

8. Right-click the **transient solution setup** and choose **Analyze**.

The field results for Time = 0 seconds will match the field results of the source steady-state solution.

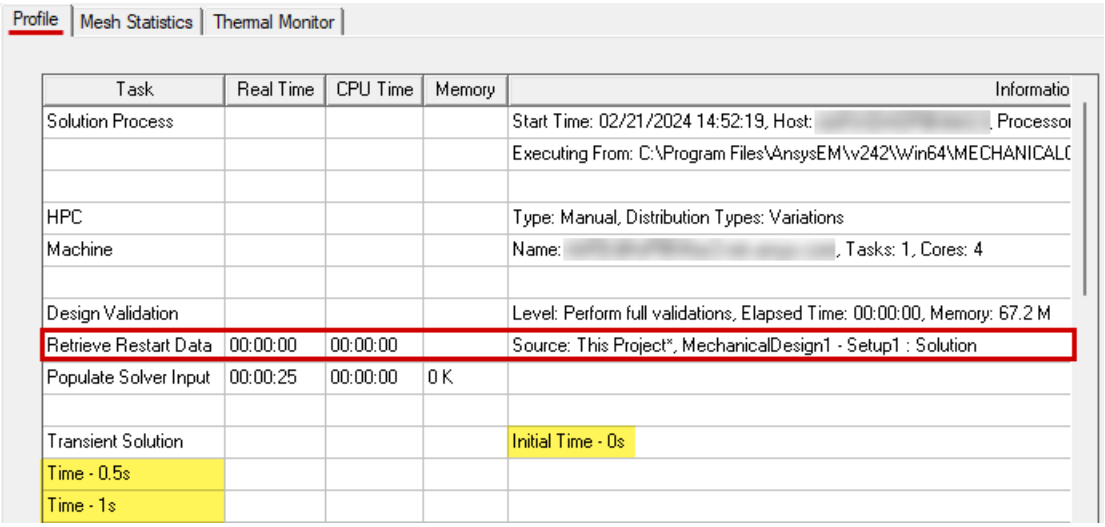
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verifying Restart Data Linking

There are two ways to verify that proper transient restart linking has been performed after the target setups solution has begun:

- Under *Analysis* in the Project Manager, right-click the target setup and choose **Profile** from the shortcut menu.

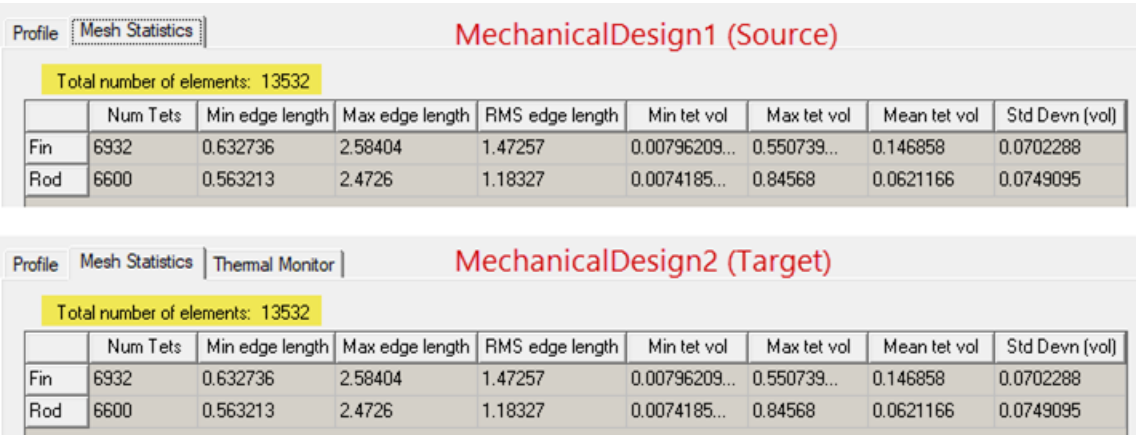
  The retrieval of restart data is listed in the profile. Additionally, when restarting from a steady-state thermal solution, the *Initial Time* of 0s is listed followed by the transient time steps solved:



Profile | Mesh Statistics | Thermal Monitor

Task	Real Time	CPU Time	Memory	Informatio
Solution Process				Start Time: 02/21/2024 14:52:19, Host: [redacted], Processo
				Executing From: C:\Program Files\AnsysEM\v242\Win64\MECHANICAL(
HPC				Type: Manual, Distribution Types: Variations
Machine				Name: [redacted], Tasks: 1, Cores: 4
Design Validation				Level: Perform full validations, Elapsed Time: 00:00:00, Memory: 67.2 M
Retrieve Restart Data	00:00:00	00:00:00		Source: This Project*, MechanicalDesign1 - Setup1 : Solution
Populate Solver Input	00:00:25	00:00:00	0 K	
Transient Solution				Initial Time - 0s
Time - 0.5s				
Time - 1s				

- Under *Analysis* in the Project Manager, right-click each of the setups in turn (source and target) and choose **Mesh Statistics**. The mesh statistics should be identical between the two designs, since the mesh is also imported when you perform a transient thermal restart analysis:



Profile | Mesh Statistics — MechanicalDesign1 (Source)

Total number of elements: 13532

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Fin	6932	0.632736	2.58404	1.47257	0.00796209...	0.550739...	0.146858	0.0702288
Rod	6600	0.563213	2.4726	1.18327	0.0074185...	0.84568	0.0621166	0.0749095

Profile | Mesh Statistics | Thermal Monitor — MechanicalDesign2 (Target)

Total number of elements: 13532

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Fin	6932	0.632736	2.58404	1.47257	0.00796209...	0.550739...	0.146858	0.0702288
Rod	6600	0.563213	2.4726	1.18327	0.0074185...	0.84568	0.0621166	0.0749095

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Restarting from a Separate Transient Thermal Solution

When you start or continue a transient thermal analysis from the last step of a previously solved transient thermal solution in a *separate* source design, you are free to modify the parameter values of assigned boundaries or excitations in the target design. Since you are not modifying the boundaries or excitations in the source design, the previously solved results are not invalidated. However, you cannot modify the geometry or add, remove, or reassign any target boundary or excitation for a restart analysis to work. One exception is that you can draw points to select for defining Thermal Monitor points in the target design that didn't exist in the source design.

If you only wish to extend the duration of a transient simulation without changing anything in the design except the *Stop Time*, there is no need to create a second design. Instead, follow the procedure for Extending the Current Transient Thermal Solution.

Transient-to-Transient Thermal Restart Procedure:

The following procedure assumes that you have already set up and solved a transient thermal analysis of your model using the same boundary and excitation assignments that are needed for the subsequent transient restart analysis.

1. Optionally, if you wish to create the target transient thermal design in a different project from the one containing the source transient solution, insert a new project or open the desired existing one.

   Add or open the second project, if desired, in the same Ansys Electronics Desktop session, not in a separate session.

   Otherwise, do nothing and continue to step 3.

2. In the Project Manager, right-click the *source* ***<transient_thermal_design_name>*** and choose **Copy** from the shortcut menu.
3. In the Project Manager, right-click the ***<project_name>*** heading of the project to which you want to add the *target* transient thermal design and choose **Paste**.

   In most cases, you will likely want to paste the target design into the same project that contains the source design.

4. Optionally, edit any boundary or excitation in the target transient design for which you want to change the value of a parameter from what was used for the source transient solution.

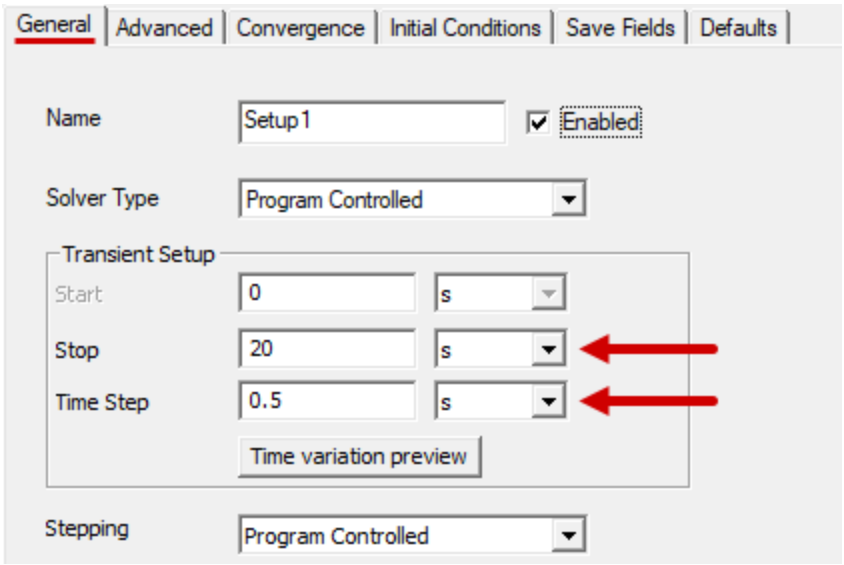
   For example, you might want to change a static *Convection Film Coefficient* value to a pwl (<*dataset*>, Time) function to vary the convective heat transfer over time (representing the effects of a variable speed fan). Similarly, you might want to vary Heat Flux or heat Generation excitations over time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> You cannot add, remove, or change the assignment set of any boundary or excitation. Doing so will cause the transient analysis to start at the default initial conditions, typically ambient temperature, at time = 0. The results of the source transient solution will not be imported if the geometry or boundary/excitation assignments do not remain identical between the two designs. Only changes of boundary/excitation parameter values are permitted.

5. Optionally, define any Thermal Monitor points at which you wish to track the temperature results versus time.
6. Under *Analysis*, in the **target design** branch of the Project Manager, double-click the ***<analysis_setup_name>*** to access the *Thermal Solve Setup* dialog box.
   a. In the *General* tab, specify a **Stop** time that is greater than the Stop time of the source transient solution.
   b. Optionally, you can adjust the **Time Step** value for a different calculation increment in the restart analysis.

      Do not set the *Start Time*. That value will automatically become the last time solved in the source solution.



   c. In the *Advanced* tab, select **Start/Continue from a previously solved setup**.

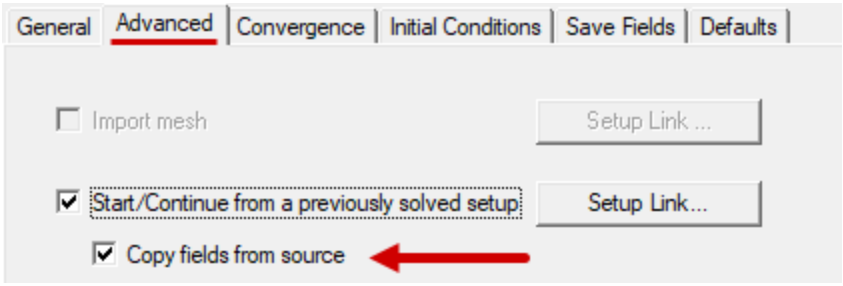
      The *Setup Link* dialog box appears.

   d. Follow the instructions on the Setup Link (Transient Thermal Restart) page to complete the required link definition.

      Continue from here after completing the link setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Select **Copy fields from source** if you want the target design to include all saved fields from the source design.



Note:

- The *"Import mesh"* option and its associated *"Setup Link"* button are unavailable when you choose the *"Start/Continue from a previously solved setup"* option because this option implicitly imports the mesh as well as the thermal solution results from the source design.
- If the *"Copy fields from source"* option is cleared, then only the field result of last time step solved is copied to target design.

f. Optionally, customize the options in the *Convergence* or *Save Fields* tabs.

g. Click **OK** to accept the solution setup.

7. Right-click the transient solution setup and choose **Analyze**.

   Depending on whether you selected the *Copy fields from source* option in step 7d, the first field result will either be the same as the first solved time step of the source solution (option selected) or the last solved time step of the source design (option cleared).

Verifying Restart Data Linking

There are two ways to verify that proper transient restart linking has been performed after the target setups solution has begun:

- Under *Analysis* in the Project Manager, right-click the target setup and choose **Profile** from the shortcut menu.

  The retrieval of restart data is listed in the profile. Additionally, when restarting from a transient thermal solution, the *Restart Time* is listed (20s in the following example, which was the last time step solved in the source design). This entry is followed by the individual time steps solved in the target design:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Profile | Mesh Statistics | Thermal Monitor

Task	Real Time	CPU Time	Memory	Informatic
Solution Process				Start Time: 02/23/2024 09:06:43, Host: [redacted], Processc
				Executing From: C:\Program Files\AnsysEM\v242\Win64\MECHANICAL
HPC				Type: Manual, Distribution Types: Variations
Machine				Name: [redacted], Tasks: 1, Cores: 4
Design Validation				Level: Perform full validations, Elapsed Time: 00:00:00, Memory: 68.7 M
Retrieve Restart Data	00:00:01	00:00:00		Source: This Project*, MechanicalDesign2 - Setup1 : Solution
Populate Solver Input	00:00:01	00:00:00	0 K	
Transient Solution				Restart Time - 20s
Time - 21s				
Time - 22s				

- Under *Analysis* in the Project Manager, right-click each of the setups in turn (source and target) and choose **Mesh Statistics**. The mesh statistics should be identical between the two designs, since the mesh is also imported when you perform a transient thermal restart analysis:

Profile | Mesh Statistics — MechanicalDesign1 (Source)

Total number of elements: 13532

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Fin	6932	0.632736	2.58404	1.47257	0.00796209...	0.550739...	0.146858	0.0702288
Rod	6600	0.563213	2.4726	1.18327	0.0074185...	0.84568	0.0621166	0.0749095

Profile | Mesh Statistics | Thermal Monitor — MechanicalDesign2 (Target)

Total number of elements: 13532

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Fin	6932	0.632736	2.58404	1.47257	0.00796209...	0.550739...	0.146858	0.0702288
Rod	6600	0.563213	2.4726	1.18327	0.0074185...	0.84568	0.0621166	0.0749095

Extending the Current Transient Thermal Solution

Suppose you have solved a transient thermal analysis with the goal of seeing the temperature results achieve steady-state conditions or to have reached a particular temperature, but the duration of the solution was not sufficient. That is, the temperature results were still trending upward at the end of the simulation and had not reached a stable value. If you make no changes to the geometry, boundaries, or excitations, which would invalidate the current results, you can extend the *Stop Time* of the transient setup and restart the solution.

In this case, you do **nothing** under the *Advanced* tab of the *Thermal Solve Setup* dialog box. No link setup is required since you are continuing the current design's solution. Additionally,

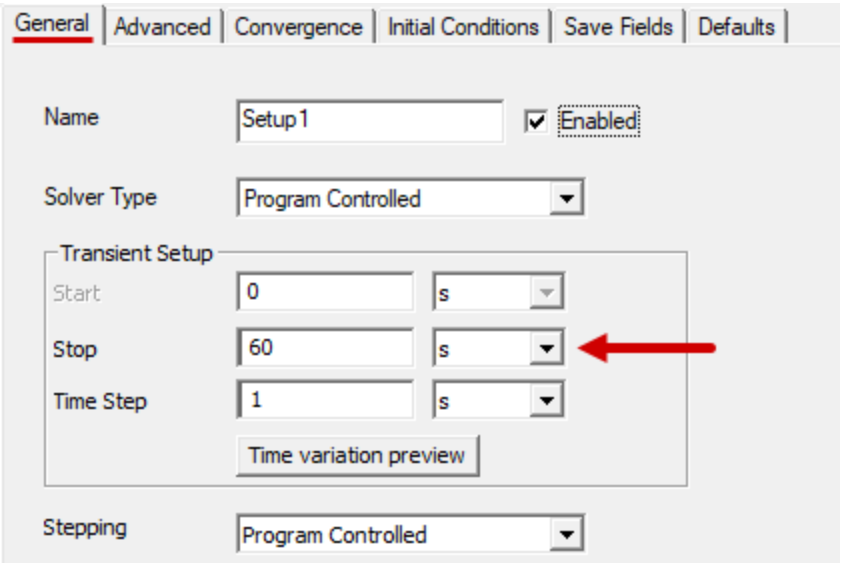
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

previously saved fields are retained without having to select the **Copy fields from source** option.

Procedure to Extend the Current Transient Solution:

The following procedure assumes that you have already successfully solved a transient thermal design:

1. Under *Analysis* in the Project Manager, double-click the ***<solution_setup_name>*** to edit its properties in the *Thermal Solve Setup* dialog box.
2. In the *General* tab, specify the increased **Stop** time in the *Transient Setup* section.



Do not alter the *Start Time*; it will continue to reflect the original start time (typically 0 s).

3. Optionally, you can change the **Time Step** going forward or customize the options in the *Convergence* or *Save Fields* tabs.
4. Click **OK** to accept the solution setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> Make no changes to the model geometry, the assigned boundaries or excitations, or the parameter values of any boundaries or excitations. Any such changes will invalidate the previously solved time steps. If you need to alter parameter values for the extended portion of the solution (such as changing a convection film coefficient), follow the procedure under Restarting from a Separate Transient Solution instead.
>
> If you accidentally invalidate the results, simply **Undo** the changes that caused invalidation to restore them.

5. Right-click the **transient solution setup** and choose **Analyze**.

   The field results for all previously solved time steps will be available in addition to the specified steps from the original stop time through the new stop time.

Setup Link (Transient Thermal Restart)

When starting or continuing a transient thermal analysis from a previously solved Mechanical Steady-State Thermal or Transient Thermal setup, use the *Setup Link* dialog box to specify the source project, source design, and other linking options.

The link between source and target designs is a one-way coupling operation. Changes you make in the target Mechanical–Transient Thermal design have no effect on the source design's results.

How to Set Up Transient Restart Linking

The following procedure assumes that you have already set up and solved a source thermal design (either steady-state and transient) and that you have already created a suitable target transient thermal design (as outlined in the Restart Requirements and Limitations section of the *Transient Restart Analyses* topic). Additionally, it is assumed that you have selected the **Start/Continue from a previously solved setup** option under the *Advanced* tab of the *Thermal Solve Setup* dialog box, and the *General* tab of the *Setup Link* dialog box is now displayed.

1. If the data is to be imported from a Mechanical–Thermal design in the same project (whether steady-state or transient), complete the following substeps. Otherwise skip to **step 2**:

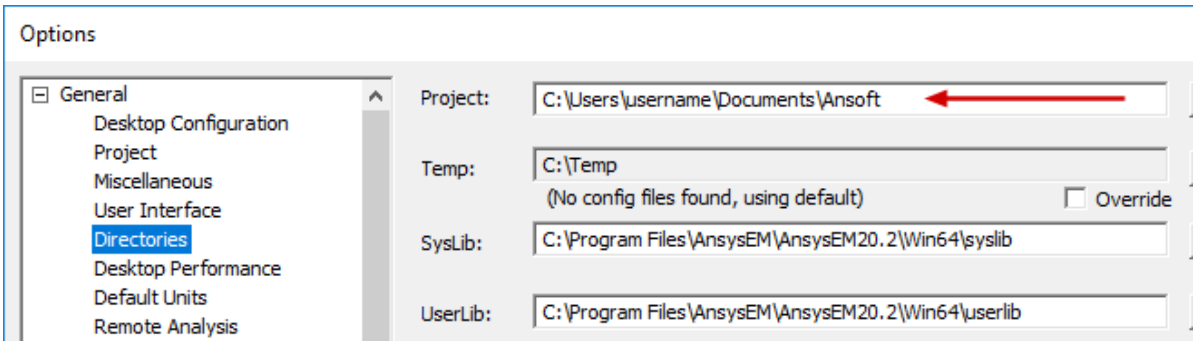
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Select **Use This Project**.

> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

b. Skip to **step 3**.

2. Alternatively, if the data is to be imported from a Mechanical–Thermal design in a different project, clear the **Use This Project** option and complete the following substeps:
   a. The target design saves the relative path to the source project in one of two ways. Choose one of the following two options under *Save source path relative to*:
      - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):



      - **This project:** Save source project path relative to the target project location.

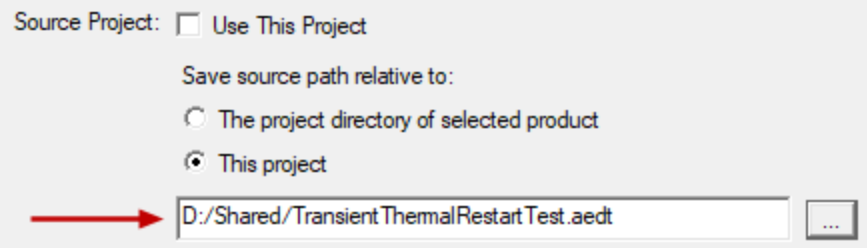
   b. Click the ellipsis button (**...**) to the right of the *Source Project* text box.

      The *Open* dialog box appears.

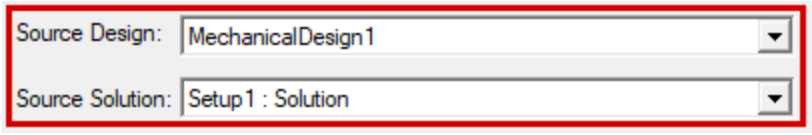
   c. Locate the source project file and select it.
   d. Click **Open**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected **Source Project** is listed in the *Setup Link* dialog box:



3. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:



> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

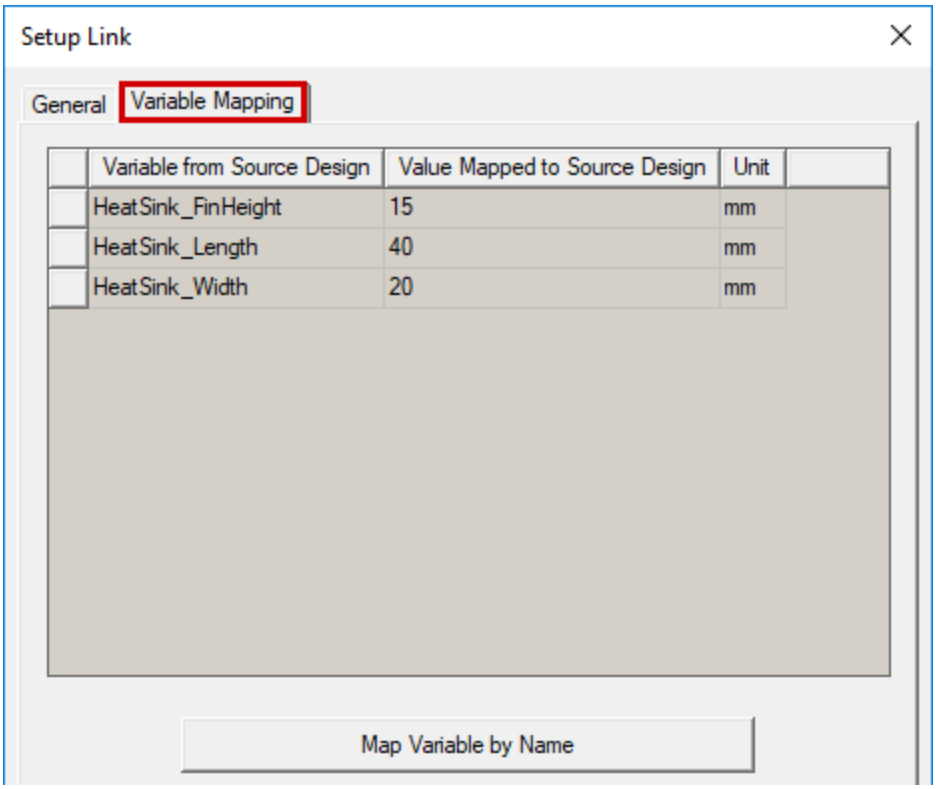
4. Optionally, select one or both of the following two options:
   - **Simulate source design as needed:** Select this option if you want an updated solution to be linked to the *target* design when the *source* design has been modified.

     > **Note:**
     >
     > Certain changes to the source design may not invalidate its solution. In these cases, a new source solution will only be run if the previous one did not converge. To achieve convergence, you may have to increase the specified maximum number of iterations or relax the convergence criterion in the source solution setup.

   - **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option has no effect if the source design is in the same project as the *target* design.
5. Click the **Variable Mapping** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:

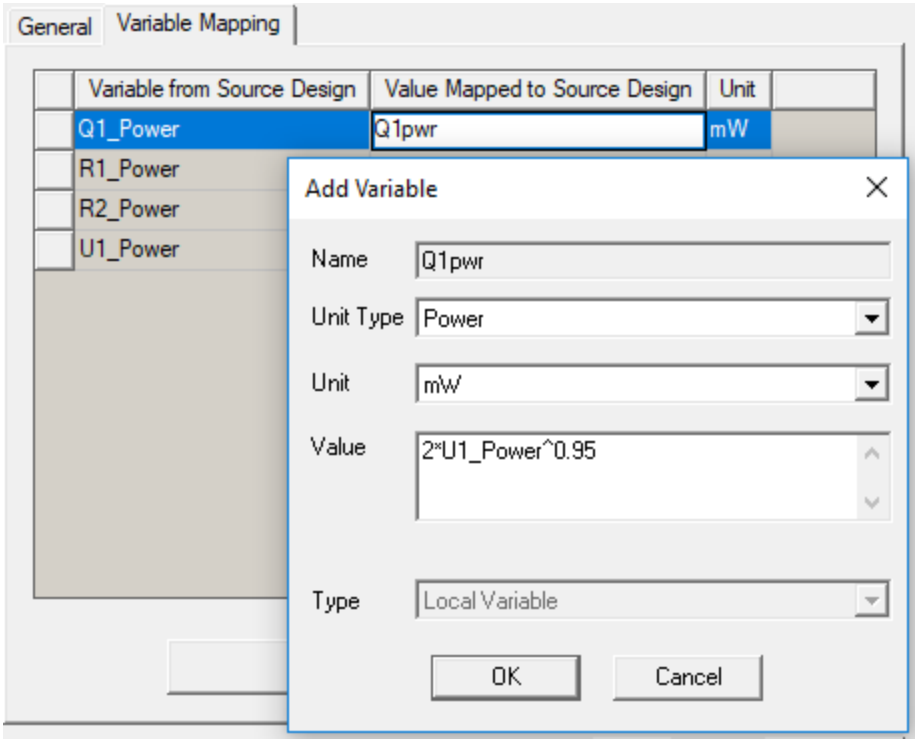


The values passed back to the source design from the mechanical design (second column) can either be constants (numeric values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

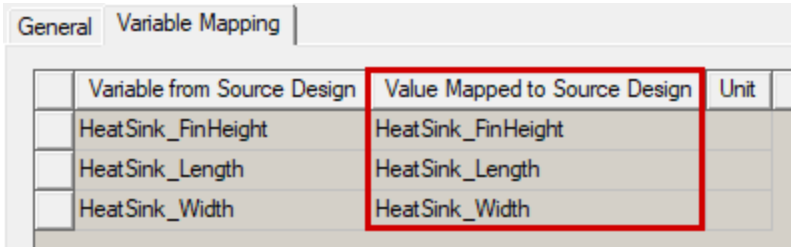
Note:

Automatic variable creation is supported in the *Variable Mapping* tab . You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the linked analyses.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- Source variables can only be mapped by name if they already exist in the target Mechanical–Thermal design. When you copy and paste the entire source design to create the target design, there is no concern. All geometry and non-geometry-related variables (such as a wattage specified in an excitation) will be present in both designs.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

Mapping variables to the source design by name is useful for Optimetrics analyses (for cases where changing the value of a variable affects both the target and source designs). You can create an Optimetrics setup in the target design that sweeps both the target and source designs. However, a variable might be used only in the target transient design (for example, a dataset variable used to control an excitation or boundary over time). Whereas the source design might have used static boundaries and excitations. In such cases, variable mapping is not applicable.

7. Click **OK** to accept the link settings and close the *Setup Link* dialog box.
8. Depending on which type of transient thermal restart scenario you are setting up, choose the appropriate link below to continue:
   - **Steady-State Thermal source design:** Continue at **step 7d** of the *Restarting from a Steady-State Thermal Solution* topic.
   - **Transient Thermal source design:** Continue at **step 6d** of the *Restarting from a Separate Transient Thermal Solution* topic.

Note:

If you later revisit the *Thermal Solve Setup* dialog box to verify or modify its properties, click the **Setup Link** button under the *Advanced* tab to return to the *Setup Link* dialog box. This dialog box will reopen in **View Only** mode. To modify the properties, first select the **Edit Link** radio button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cached Link Data and Solution Invalidation (Transient Thermal Restart)

The mesh and results from the source design are cached in the target design after its simulation. You can remove this cached data (instructions below).

If you've updated the source design and want the imported data in the target design to reflect the changes in the source project, you need to clear the linked data. Doing so invalidates the previous solutions. Clear the linked data using one of the following three methods:

Method 1: Use the **Revert to Initial Condition** command, accessible from one of the following two locations:

- Under *Analysis* in the Project Manager, right-click the setup you want to revert and choose **Revert to Initial Condition** from the shortcut menu.
- Using the menu bar, click **Mechanical** > **Analysis** > **Revert to Initial Condition**. The behavior depends on whether a solution setup is currently selected under *Analysis* in the project manager when you click the command:
  - **If a solution setup is currently selected in the Project Manager:** Only the selected setup is reverted.
  - **If no setup is selected in the Project Manager:** *All* setups in the active design are reverted.

> **Note:**
>
> This command clears the saved fields, restart solution, and any thermal monitor data. However, the mesh data (for the current and initial mesh) is preserved.

Method 2: Use the **Clean Up Solutions** command, available from the menu bar, to clear linked data, meshes, and/or solutions for *all* solution setups in the active design:

1. Using the menu bar, click **Mechanical** > **Results** > **Clean Up Solutions**.

   The *Clean Up Solutions* dialog box appears.

2. In the *Solutions* section of the dialog box, choose one of the following sets of options:
   - Select the **Linked Data Only** option.
   - Select the **Fields and Meshes** and **Include Linked Data** options.
   - Select the **All Solutions** and **Include Linked Data** options.
3. Click **Do Deletions**.

Method 3: Use the **Clear Linked Data** command, accessible from the following two locations:

- Under *Analysis* in the Project Manager, right-click a solution setup and choose **Clear Linked Data** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This method clears the linked data and invalidates the thermal monitor and field results for the specific setup that was right-clicked.

- From the menu bar, click **Mechanical** > **Analysis** > **Clear Linked Data**. The behavior depends on whether a solution setup is currently selected under *Analysis* in the project manager when you click the command:
  - **If a solution setup is currently selected in the Project Manager:** Linked data is purged from the selected setup, and its thermal monitor and field results are invalidated.
  - **If no setup is selected in the Project Manager**: Linked data is purged from *all* setups in the active design. However, all thermal monitor and field results from the target solution are retained.

Note:

- Data is cached for each solution setup that is linked to a source thermal solution.
- The cached data and solutions are automatically cleared when the target Mechanical design is edited in a way that invalidates the previous solution (such as adding, removing, or modifying geometry, boundaries, or excitation; or reassigning boundaries/excitations).
- Cached data in the target design is *not* automatically removed when the *source* design is edited.

Assigning Thermal Materials

When choosing a material for a thermal analysis, you will see a different set of materials and material properties compared to a circuit, electromagnetics, or structural analysis. Instead of properties such as Electrical Conductivity and Relative Permittivity, the physical properties of interest for thermal analyses are the Thermal Conductivity, Mass Density, Specific Heat, Thermal Material Type (Solid or Fluid), and Dynamic Viscosity (fluids only). Materials with at least the first of these five properties defined will appear in the *Select Definition* dialog box, where you choose the default material or assign an object material.

Only the Thermal Conductivity is needed for **Steady-State Thermal** solutions (other thermal material properties can be zero). For **Transient Thermal** solutions, non-zero Mass Density and Specific Heat values are also required. When defining materials for a steady-state solution, it is best to include the mass density and specific heat values when available, even though they are not required. Doing so ensures that the same materials will also be suitable for a transient analysis if the solution type is later changed, avoiding a transient solution error.

Anisotropic Properties

Materials for thermal solutions can have simple isotropic properties (that is, behaving quantitatively the same way in every direction), or anisotropic properties (exhibiting different

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

behavior in three orthogonal directions). Specifically, the Thermal Conductivity can differ in each of three orthogonal directions. These directions are defined by the **Orientation** property of each object that uses the material (indicating a global or user-defined coordinate system).

All predefined library materials are of the simple (isotropic) type. You must define your own custom material to use anisotropic properties.

See the Defining Anisotropic Tensors topic for more information.

Temperature-Dependent Properties

Thermal materials can also have temperature-dependent properties. Specifically, the Thermal Conductivity, Mass Density, and Specific Heat can vary as a function of temperature.

Additionally, the Dynamic Viscosity of fluids can vary as a function of temperature. This property is used in the calculation of the equivalent thermal conductivity for rotating fluid boundaries.

> **Note:**
>
> If your design will experience temperatures below 0° C, you must define the thermal modifiers for all applicable properties using manually defined dataset tables. General expressions (linear, quadratic, conditional, and so on) are automatically converted to tables for the solver, and these automatically generated tables only contain temperature datapoints in the range of 0–1000° C.

All predefined library materials are of the non-temperature-dependent type. You must define your own custom materials to use temperature-dependent properties.

See the Specifying Thermal Modifiers topic for more information.

Default Material and Applicable Libraries

The default material for thermal analyses is *air*. Thermal materials can be solid, liquid, or gaseous, with the *Fluid* material type representing both liquids and gases. All 3D elements are treated as solids, but the dynamic viscosity and other properties of fluid materials are used to estimate the equivalent thermal conductivity within the gap of rotating machine simulations.

The following libraries contain materials suitable for Thermal solutions:

- *[sys] ArnoldMagnetics*
- *[sys] GRANTA Materials Data for Simulation*
- *[sys] GRANTA Producers Materials Data for Simulation*
- *[sys] Materials*
- *[sys] Vacuumschmelze*

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The Specific Heat of the materials is *not* required because this property is only applicable to transient thermal analyses (that is, those in the time domain). A steady-state solution provides the temperatures and heat flux results that exist once a steady-state condition (or thermal equilibrium) is achieved. However, the time it takes to achieve thermal equilibrium is not determined.

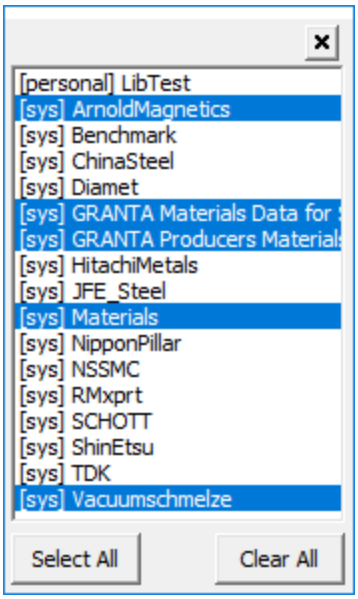
Material Selection

To set the default material for all objects you subsequently create:

1. At the far right end of the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.

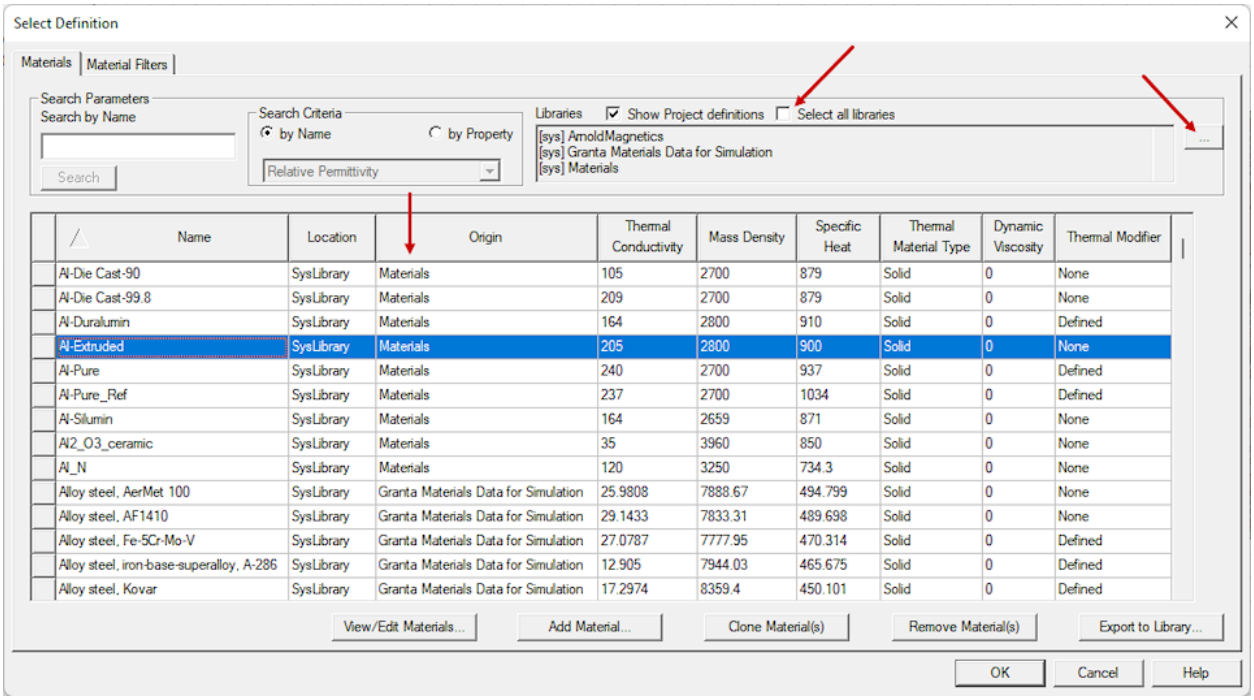
   The *Select Definition* dialog box appears.

2. In the *Libraries* list, ensure that one or more of the previously listed suitable libraries are selected, or a user library with thermal material properties. Click the ellipsis button (**...**) to the right of the *Libraries* list to access a pop-up dialog box in which you can select which libraries to include in the *Materials* list:



   Alternatively, select the **Select all libraries** option:

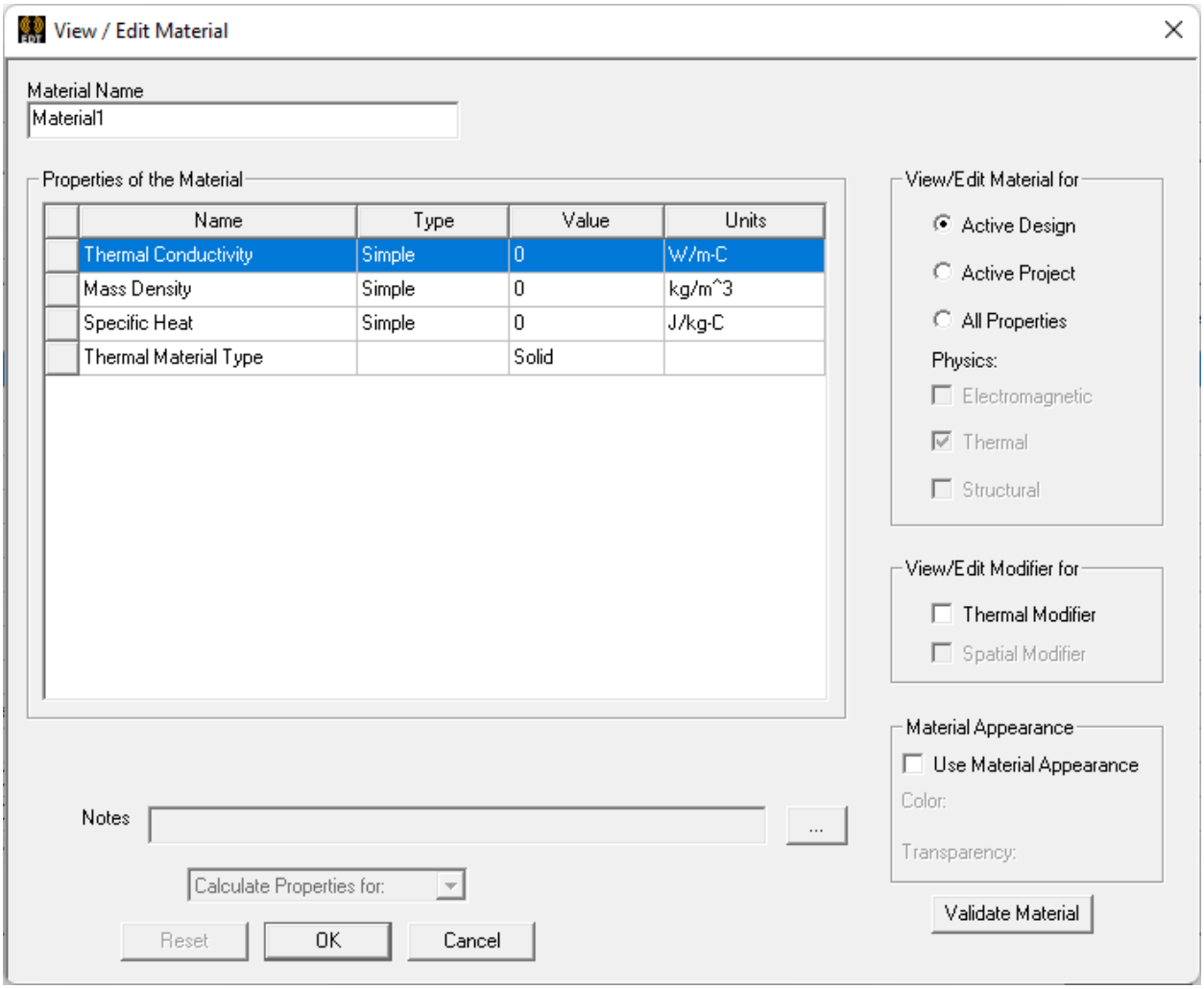
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Optionally, type a portion of the material name in the **Search by Name** text box. The list will automatically scroll as needed and the first material matching the search string will be selected. If you scroll the selected material off screen, click **Search** to quickly bring the searched material back into view.
4. Select the desired material, if present.
5. Optionally, to define your own material properties, click **Add Material**.

   The *View / Edit Material* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optionally, if you are adding a gas or liquid material, choose **Fluid** from the **Material Type** drop-down menu in the *Value* column.
7. Optionally, to specify anisotropic *Thermal Conductivity* values (when the conductivity varies in the three global directions), select **Anisotropic** from the **Type** drop-down menu. Otherwise, the *Simple* option is for isotropic properties (when the conductivity is the same in every direction).
8. Optionally, if you want to define temperature-dependent material properties, follow the instructions in the Specifying Thermal Modifiers topic.
9. If editing properties, or defining a new material, specify the desired **Material Name, Thermal Conductivity, Mass Density, Specific Heat,** and **Dynamic Viscosity** values.

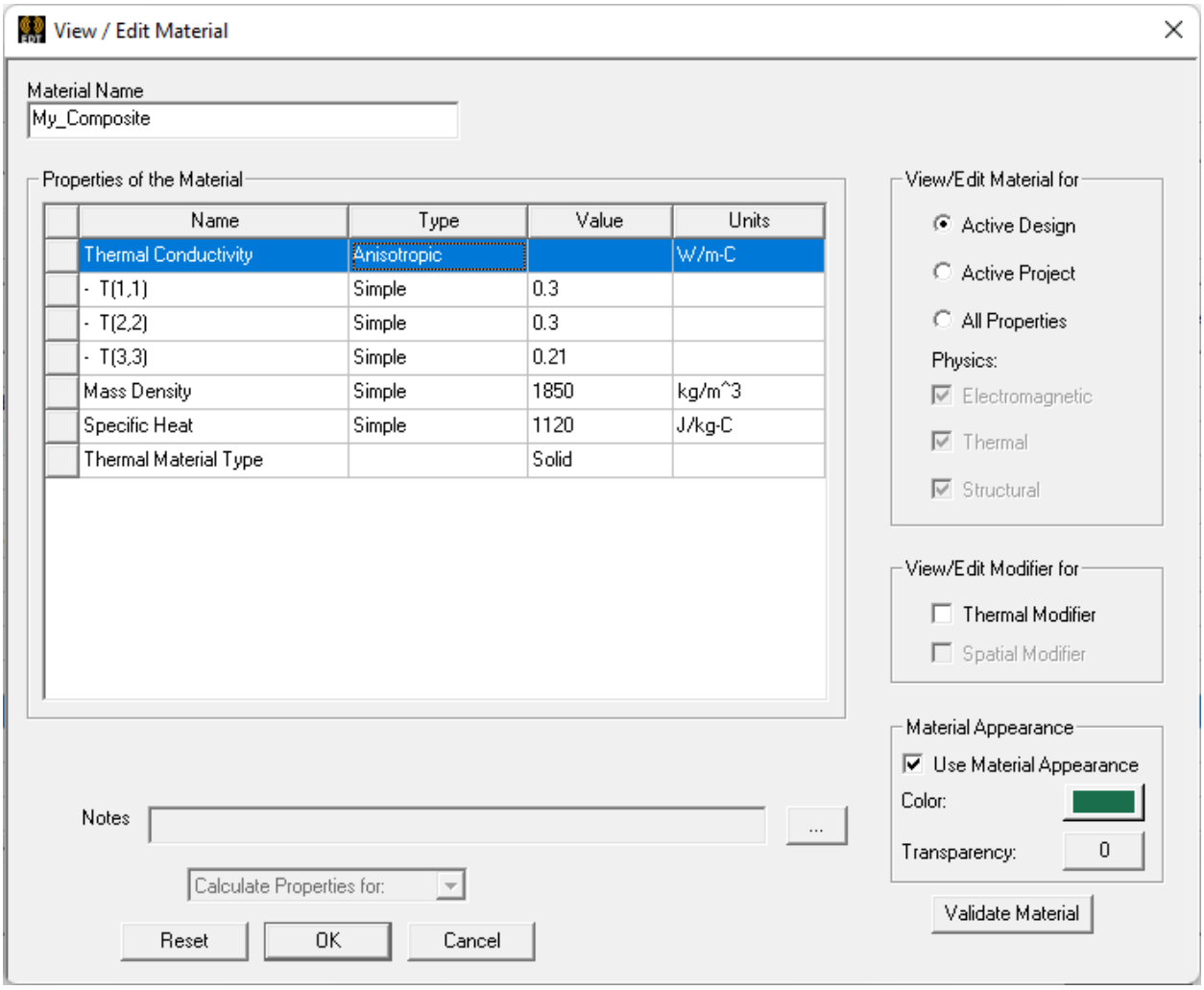
   The following material validation rules apply:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Simple (isotropic) materials:
  - Thermal Conductivity (λ) must be greater than zero: (λ > 0)
  - For Transient Thermal solutions, Mass Density and Specific Heat must both be greater than zero.
- Anisotropic materials:
  - Thermal Conductivity (λ) must be greater than zero in all three directions: [for λ, T(1,1) > 0, T(2,2) > 0, and T(3,3) > 0]

    For *Thermal Conductivity*, T(1,1), T(2,2) and T(3,3) respectively correspond to λ_x, λ_y, and λ_z (the conductivity in the three axis directions, whether the coordinate system is global or user-defined).

The following image shows an example of the properties for an *Anisotropic* material:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Optionally, select the **Use Material Appearance** option and specify the desired **Color** and **Transparency** attributes.
11. Optionally, click **Validate Material** to check for errors in the specified properties.

    If the properties pass the validation check, a green check mark appears to the right of the *Validate Material* button. Otherwise, a red X appears, and a pop-up message provides the error details.

12. Click **OK** to close the *View / Edit Material* dialog box.
13. Click **OK** to close the *Select Definition* dialog box.

To assign a different material to a selected solid object:

1. Select the object to which you want to assign a material.
2. Use one of the following five methods of accessing the *Select Definition* dialog box:
    - On the **Draw** ribbon tab, click **Material**.
    - Using the menu bar, click **Modeler** > **Assign Material**.
    - Right-click in the **Modeler** window and choose **Assign Material** from the shortcut menu.
    - In the **Attribute** tab of the docked *Properties* window, choose **Edit** from the **Material** *Value* drop-down menu.
    - If the *Properties* dialog box appears automatically when you create an object, select the **Attribute** tab. Then, choose **Edit** from the **Material** *Value* drop-down menu.
3. Complete steps 2 through 6 of the previous procedure.

To add, remove, or edit materials:

1. Use one of the following two methods of accessing the *Edit Libraries* dialog box:
    - Using the menu bar, click **Tools** > **Edit Libraries** > **Materials**.
    - Under *Definitions* in the Project Manager, right-click **Materials** and choose **Edit Library** from the shortcut menu.
2. Select the desired material library from the *Libraries* list, or select **Show all libraries**.
3. Select an existing material or click **Add Material** to define a new one.
4. Optionally, you can click the following buttons once you've selected one or more existing materials:
    - **View / Edit Materials:** To change the name or properties.

    > **Note:**
    >
    > When you change the properties of a *SysLibrary* material, a new *Project* library material is created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Clone Material(s):** Make copies of the selected materials as a starting point for defining new ones.
- **Remove Material(s):** Remove the selected materials from the library.

> **Note:**
>
> You cannot remove predefined *SysLibrary* materials. You can only delete materials that you've added to the project library or a user library (and only if the material is not being used in the current design).

5. After making the desired changes, click **OK** to close the *View / Edit Material* and/or *Edit Library* dialog boxes.

Shell Objects

In addition to solid objects, Mechanical – Thermal models can contain shell objects, which you can draw in the mechanical design or import from a different design or CAD model. Shell objects are sheet objects for which you have selected the **Shell Element** attribute. When you select this attribute, in the object *Properties* dialog box or the docked *Properties* window, the following two additional attributes become available:

- **Material:** Choose a material from the library, as you do for solid objects, or define your own material properties.
- **Effective Thickness:** The shell is always represented as a zero-thickness object within the model geometry, but the thickness is defined numerically, and the specified value is passed to the solver.

> **Note:**
>
> The default thickness for shell objects in Mechanical – Thermal solutions is 1E-7 mm. For many situations, this thickness can be considered to be infinitesimally small, preventing the shell object from providing a path for significant heat flow in parallel with the object on which it is overlaid.

Thermal results are calculated at the midplane of the shell (that is, at the middle of the shell's thickness). The face of the sheet, as represented in the model geometry, corresponds to the midplane of the shell object.

Shell objects provide the following capabilities in Mechanical – Thermal solutions:

- Excitations and boundaries in other design types are frequently applied to sheet objects. An example is a sheet object in an HFSS model with a Wave Port excitation applied to it. You can use a corresponding shell object in the Mechanical design as the target of an imported surface EM Loss excitation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Shell objects can represent thin conductors on the face of, or passing through, a solid object (such as the copper traces of a printed circuit board). In this case, you will need to specify the actual thickness of the trace instead of using the default shell thickness.

  > **Note:**
  >
  > Field overlays will appear on shell elements that are overlaid on or pass through a solid body. The solid body can be a solid material, fluid, or gas. Currently, field overlays will not appear on free-standing shell elements.

- You can overlay a shell object on the face of a solid object as a means of superimposing two or more boundary conditions or excitations that cannot be combined on a single face.
- You can use a shell object to encompass multiple contact areas between a large object (such as a printed circuit board) and several adjacent objects (such as surface-mounted electronic components). Then, assign a singe contact definition to the shell to take care of all ojbect-to-object contact areas within its perimeter. Keep the default infinitesimal thickness to prevent the introduction of a significant heat conduction path parallel to the contact plane. See Contact Guidelines and Examples.

  > **Warning:**
  >
  > Shell objects in Mechanical–Thermal designs do **not** support a change in temperature across the shell's thickness. There is only one temperature value at any particular point along the shell, and it is at the mid-plane. Temperature variations can occur only in the in-plane directions. Therefore, do not attempt to use a shell object's material properties and specified thickness to represent thermal contact resistance. Rather, assign *Contact* to the shell, or to one or more of the adjacent object faces, and specify the desired thermal resistance in the contact definition.
  >
  > See Thermal Contact and How to Assign Thermal Contact for more information.

- You can overlay a shell object on the face of a solid object as a means of limiting a boundary or excitation to only a portion of a larger face. The shell object effectively becomes a subface of the solid object's larger face.

  An example of such a model is a heat sink with a shell object representing the contact area of a semiconductor. You could apply a heat flux excitation to the shell object rather than including the semiconductor body in the model and assigning heat generation to it. As stated in the previous note, the extremely thin default shell element thickness will not conduct a significant amount of heat in most cases.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Alternatively, you can create a sheet object or a non-model solid object to use temporarily as a tool part for imprinting onto the face of a solid part, creating a subface. You can assign thermal boundaries and excitations directly to the resulting subface of the solid part. The advantage of this alternative method is that there are no shell elements to potentially act as a parallel path for heat flow.
- You must use the *Tau* mesher when you have performed imprinting and have **not** kept (that is, *Cloned*) the sheet objects used as the *Tool Parts*. The *Classic* mesher will simplify coplanar and fully tangential subfaces by merging them back into the original, larger face (unless there is an adjacent sheet or solid body at the imprinted subface). The model geometry will not be altered, but the Classic mesher will ignore the subfaces when generating the solid mesh.

The method of imprinting an object on a flat or curved face, for the purpose of limiting boundaries or excitations to a portion of a larger face, is the same for Mechanical – Modal and Mechanical – Thermal solutions. See the **Related Topics** below for more information.

Boundaries (Thermal)

The boundaries currently available for thermal analyses in the Ansys Electronics Desktop application are the following:

- **Convection:** Simulate the transfer of heat between the faces of objects and the surrounding environment. Heat transfer is calculated based on the object surface temperature, specified ambient temperature, and film coefficient (also referred to as the heat transfer coefficient). The film coefficient can be a user-defined, uniform value or non-uniform values imported from an Icepak design solution.
- **Temperature:** Dictate a fixed temperature at selected faces.
- **Rotating Fluid:** Estimate the heat transfer through the fluid (typically air) within the envelope of a rotating electric machine model (motor, generator, or alternator).

Use the preceding links for a fuller definition of each available thermal excitation and instructions on assigning them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For multi-body models, perfect thermal contact exists between adjacent bodies. That is, the thermal contact resistance is zero between contacting objects. To simulate thermal contact resistance, say between a transistor and a heat sink, introduce a separate thin part between them, and set the thermal conductivity for its material to an appropriate value (typically, significantly less than that of the other two objects).

Excitations (Thermal)

Two types of excitation are available for *Thermal* solutions in Ansys Electronics Desktop. Click the links below to see complete information about each excitation and how to apply them:

- **EM Loss:** Surface or volumetric heat rates (thermal power) based on the electromagnetic loss results of another Electronics Desktop design analysis.
- **Heat Flux:** User-specified thermal power entering or exiting a face or group of faces.
- **Heat Generation:** User-specified thermal power produced or absorbed by a 3D object or each object within a group of selected objects.

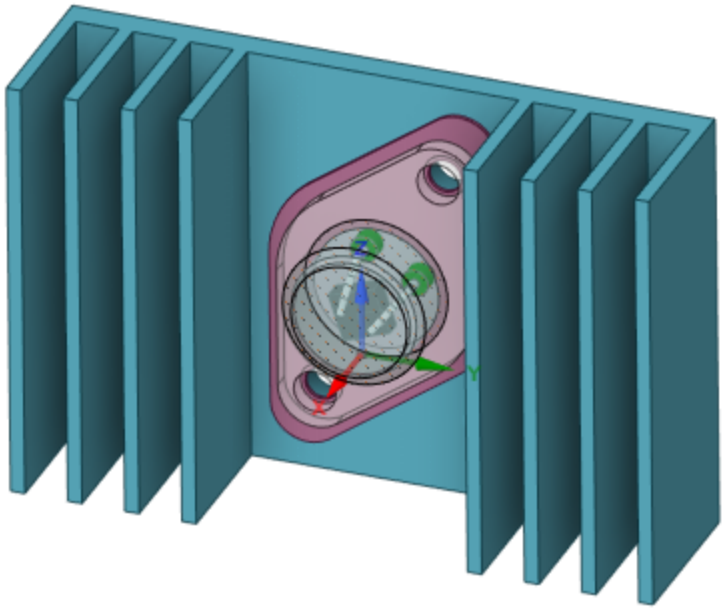
Use the preceding links for a fuller definition of each available thermal excitation and instructions on assigning them.

Thermal Contact

Contact with support for thermal resistance is a feature of Mechanical–Thermal solutions in the Ansys Electronics Desktop application. Adjacent parts in electronic assemblies typically have less than perfect thermal contact between them. The roughness of part surfaces reduces the area where the parts are truly touching, and this effect introduces a resistance to the flow of heat between the parts. In order to obtain accurate and realistic temperature results, you must take this thermal contact resistance into account when setting up an analysis model.

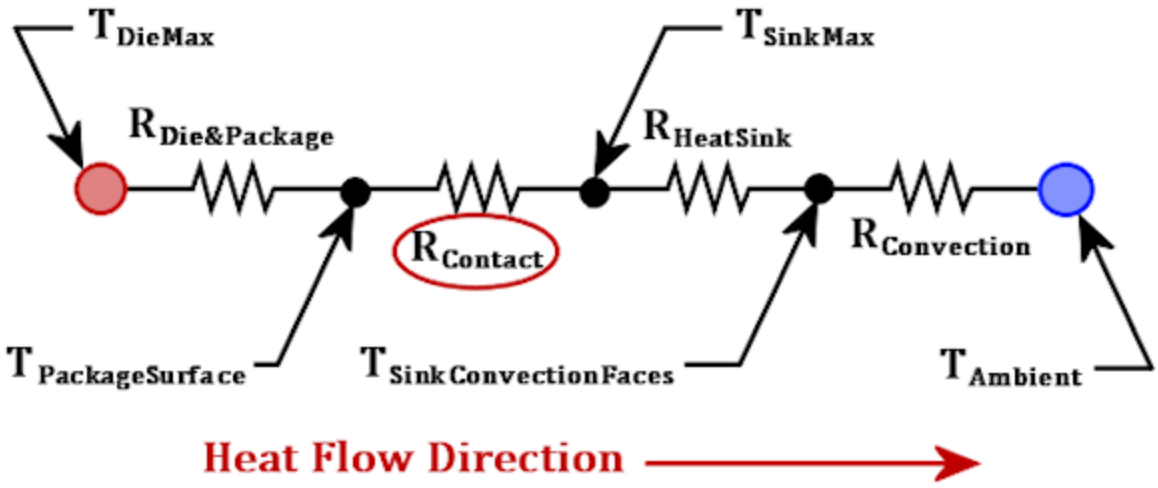
Consider a semiconductor device mounted to an aluminum heat sink:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Typically, the greatest temperature gradients occurs at the mounting surface where the semiconductor package and heat sink meet (especially when an electrical insulator is used) and between the heat sink fins and the ambient air. The reason for these greater temperature gradients is that the thermal conductivities within the semiconductor and heat sink are relatively high compared to the conductivity across the part-to-part interface and via convection to the ambient air.

The heat flow through materials, contact faces, and convection boundaries can be represented by a series of resistors (analogous to electrical current flowing from a higher voltage to a lower voltage through a series of resistors). In this case, temperature drives heat flow (in units of power) rather than voltage driving current in amperes. The higher the relative thermal resistance, the greater the temperature change across any of the resistors (in the same way that voltage drop is proportional to resistance in an electrical circuit):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When an insulator is used between the semiconductor and heat sink, $R_{Contact}$ can include the combined effective of the insulator's thermal conductivity and thickness as well as the contact resistance on both sides of the insulator. You do **not** have to include the insulator in the model. Empirical data is available for various standard component packages, insulators, heat transfer enhancement materials (such as silicone grease and thermal pads), and for different clamping forces. Alternatively, you can choose to include the insulator as a solid object in the model and define contact for its front and back faces. In this alternative case, $R_{Contact}$ in the preceding diagram would be replaced by three series-resistors representing the semiconductor contact resistance, insulator material conductivity, and heat sink contact resistance, respectively.

The following table shows typical temperatures for a convection-cooled semiconductor and heat sink assembly, as represented by the preceding diagram:

Location	Temperature (°C)	Corresponding Resistor	ΔT (°C)
T_{DieMax}	100		
$T_{PackageSurface}$	94	$R_{Die\&Package}$	-6
$T_{SinkMax}$ (mounting surface)	72	$R_{Contact}$	-12
$T_{SinkConvectionFaces}$	68	$R_{HeatSink}$	-4
$T_{Ambient}$ (surrounding air)	25	$R_{Convection}$	-43

You can consider an increase in thermal contact resistance as a decrease in thermal contact conductance. In fact, resistance is the exact reciprocal of conductance. When defining contact, you can choose to specify either conductance or resistance. Additionally, you can define the conductance or resistance as a total value (for the combined area of the selected faces) or as a distributed value (that is, an area-based value). Additionally, you can specify thermal resistance that is equivalent to a material's thermal conductivity and thickness.

Thermal Contact Video

This video demonstrates how to account for thermal contact resistance in your designs and the importance of doing so. The model is a thyristor bridge output module from an industrial motor control.The presentation includes a comparison of temperature results for with and without contact resistance. It also compares the results with and without electrical insulators installed between the semiconductors and heat sink.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Video Narration Script (click to view)

0:05 This is one of three output modules belonging to a full-wave, half-controlled, thyristor bridge – specifically, one designed to drive a 20 horsepower, 1770 RPM, 460 VAC, 3-phase electric motor. We will be performing a Mechanical-Thermal analysis to determine the steady-state operating temperatures when driving the motor at 15% overload.

0:28 In this sectioned view, you can see the internal objects comprising the components. The semiconductor dies are the heat source.

0:36 To maximize thermal conduction to the heat sink, the components are mounted without insulators, so the heat sink is electrically hot and must be insulated from the cabinet.

0:46 For time and convenience, several operations have already been performed, as follows:

0:51 1. Appropriate materials have been assigned to all objects.

0:56 2. The objects in contact with the heat sink have been imprinted on it for the purpose of isolating contact areas from the exposed portion of the front and back mounting faces.

1:07 Later, we will assign convection boundaries to the exposed faces of the heat sink. Currently, user-assigned contact areas are not automatically excluded from convection boundaries, as they are for default bonded contact areas. So, the mounting faces had to be split manually. Our convection assignment will exclude these contact faces.

1:29 To keep the objects used as imprinting tool, the option, “Clone tool objects before operation” was selected before completing the Imprint command.

1:39 3. A total of four face lists have been created to accommodate the selection of assignment faces for contact and for convection boundaries.

1:48 We’ll define different thermal resistances for the contact faces on the front and back sides of the heat sink. Thermal grease is applied between the component bodies and the front contact faces to reduce thermal resistance. These are the primary heat flow faces since heat generation occurs on the front side of the module. No heat sink compound is applied to the stud and nut threads, between the washer or terminal and the nut, or between the washer or terminal and the heat sink. Accordingly, we’ll define a higher thermal contact resistance for the back contacts.

2:22 We will also be assigning two different convection boundaries to account for the fins having grooved faces on their left and right sides.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2:29 V-grooves increase the surface area for heat dissipation by about 41% relative to smooth faces. Rather than complicating the geometry with many additional and very thin faces, we can account for the increased surface area by raising the convection film coefficient while leaving the model's faces smooth.

2:48 4. A length-based mesh operation has been assigned inside all objects, specifying a maximum length of 4 mm.

3:02 5. The ambient temperature variable has been increased from its default value to 30 Celsius.

3:09 6. An analysis setup has been added using default settings.

3:15 To obtain realistic simulation results, you must consider the thermal resistance that exists between adjacent objects. Roughness along faces of manufactured parts, exaggerated in this image for clarity, reduces the area of true metal-to-metal contact to an amount much less than the apparent contact area. Thermal grease applied to the parts fills in the surface irregularities and improves heat transfer. Nonetheless, even with thermal grease applied, contact resistance is significant. In electronic assemblies, temperature step-changes occur across component-to-heat sink interfaces.

3:50 If you don't assign contacts, adjacent objects are bonded by default and there is no thermal resistance between them. That is, the thermal conductivity from one part to the next is perfect, and the temperatures are identical at both sides of the contact face. Therefore, you could significantly underestimate the temperature of the electronic components, especially if an insulator is used. In that case, the heat must flow through two contact interfaces and through the thickness of the insulator.

4:19 Let's select the front contact faces and assign contact. Note that, where objects meet, you only select the contact face of one of the two objects. The target face on the adjacent object is found automatically by the solver. For front contact, we are choosing the two imprinted circular faces on the front of the heat sink.

4:39 Specify Contact_Front as the Name.

4:43 For the Resistance Type, choose Thermal Impedance. This option is a distributed value for which the total resistance varies depending on the contact area. It is the inverse of the thermal conductivity per unit area. Specify 25 Celsius mm squared per watt, which is a reasonable assumption for smooth surfaces with thermal grease applied and with a relatively high clamping force.

5:08 Repeat the procedure for the back contact faces. For this assignment, we are choosing the front and back faces of the washer and terminal (for heat sink and nut contact, respectively) and the surface of the hole in each nut (for nut-to-stud contact).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5:23 This time, specify Contact_Back as the Name and 150 Celsius mm squared per watt as the Thermal Impedance. We are assuming that the contact resistance of a smooth, bare metal-to-metal interface is approximately six times greater than that of a similar interface with thermal grease applied.

5:44 The remaining object interfaces will all be treated as bonded contact without thermal resistance, which includes internal contact between all objects comprising the diode and SCR.

5:55 Next, select the smooth convection faces and assign a convection boundary.

6:00 Specify a uniform Film Coefficient of 10 watts per meter squared Celsius.

6:09 Repeat this procedure for the grooved convection faces but this time, specify 14.1 watts per meter squared Celsius as the Film Coefficient. As mentioned previously, this value is 41% greater than the other convection boundary to account for the increased surface area of grooved faces.

6:28 Next, select the SCR Die and assign a Heat Generation with a Total Power of 35 watts. The wattage is based on a current of 28 amps and a forward voltage drop of 1.25V through the SCR.

6:51 Repeat this procedure for the Diode Die, but this time, specify a Total Power of 21 watts, because the forward drop of the diode is about 0.75 volts, and the current is the same.

7:03 We're ready to run the analysis, which takes about 3 minutes to complete on a current high-end workstation.

7:15 Select all objects and overlay the temperature results.

7:26 It takes a few moments to read in the values. Press F6 to display the geometry as a wireframe, suppressing the model shading for a cleaner contour plot.

7:31 Press F6 to display the geometry as a wireframe, suppressing the model shading for a cleaner contour plot.

7:39 The minimum temperature is approximately 63 Celsius, putting the coldest part of the heat sink 33 degrees above ambient. The maximum temperature occurs at the SCR Die and is about 113.4 Celsius, which is a typical junction operating temperature for power semiconductors.

7:58 Hide the temperature overlay, select all objects again, and create a mesh plot.

8:12 As you can see, the mesh uniformity and quality appear to be good.

8:18 Next, let's create a Fields Summary listing the minimum, mean, and maximum temperatures for each of the objects in the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8:37 You can see, for example, that the temperatures within the heat sink range from about 63 to 75.8 Celsius and the temperatures in the SCR Die range from about 104 to 113.4 Celsius.

8:53 Finally, to better quantify the importance of considering thermal contact resistance, two variants of the original model were created. This design shows the results if thermal contact is not assigned, and all objects are in perfect bonded contact. The maximum temperature of 106.8 C is about 6.6 degrees cooler than the version with thermal contact resistance considered.

9:17 And this design shows the results if 3 mil thick mica insulators are added to electrically isolate the SCR and Diode from the heat sink. The Thermal Impedance was increased to include the effects of two contact interfaces (at the front and back of each insulator) plus the impedance of the mica itself, which has a much lower thermal conductivity than metals. The resultant temperature of 148 C is 34.6 degrees hotter than the thermal contact case without insulators and 41.2 degrees hotter than the case with perfect contact throughout.

9:52 Thermal contact resistance is easy to account for in Mechanical–Thermal simulations, and we hope you appreciate the importance of doing so. Use this tool to ensure the reliability of your circuit and system designs. Thank you for watching this video.

Thermal Resistance Types

When assigning contact, the required properties you need to define depend on the **Thermal Resistance Type** that you choose in the *Contact* dialog box. There are five available options, as follows:

- **Thermal Conductance:** Total conductance for the combined area of all selected faces (in units of power per Δtemperature). Do **not** use this option when the actual contact area differs from the area of the assignment faces (see Note below).
- **Thermal Conductance per Area:** The distributed conductance per unit area (in units of power per area per Δtemperature). This option is suitable for all contact cases.
- **Thermal Resistance:** Total resistance for the combined area of all selected faces (in units of Δtemperature per power). Do **not** use this option when the actual contact area differs from the area of the assignment faces (see Note below).
- **Thermal Impedance:** An area-based resistance value (in units of Δtemperature times area per power) for which the resultant total resistance is *inversely proportional* to the contact area. That is, the greater the contact area, the lesser the total resistance. The term "*Resistance per Area*" is **not** used because that term would imply a proportional relationship between resistance and area, rather than an inversely proportional relationship. This option is suitable for all contact cases.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Thickness:** This option simulates the resistive effect of an intermediate object (not included in the model geometry) sandwiched between the two contacting objects. For this resistance type, you define the thickness and the material. The effective conductivity per unit area is determined by the solver from the thermal conductivity of the material and the specified thickness.

  An example of when you might use this option is to simulate the effect of a thin epoxy layer bonding two parts together. You could also interpose an electrical insulator between a semiconductor and a heat sink when the insulator is not included in the geometry. However, this option would only account for the thermal resistance of the insulator and not for the contact resistance between the insulator and the object on each side of it. To account for both the insulator material and the two contact resistances, choose one of the other four options and define the appropriate value to account for all three effects ($R_{Contact}$ + $R_{Material}$ + $R_{Contact}$).

> **Note:**
>
> When the total area of the selected model faces differs from the actual contact area, you should **not** use the *Thermal Conductance* or *Thermal Resistance* option to specify contact. For either of the following two cases, the selected face area and the actual contact area differ, and you should only choose the **Conductance per Area**, **Impedance**, or **Thickness** resistance type:
>
> - When a large face is selected to cover contact with multiple smaller adjacent objects, and less than 100% of the face is in contact with other objects (see the first case under Contact Examples in the *Contact Guidelines and Examples* topic.)
> - When any model face in the selection set is only partially in contact with an adjacent object (that is, when there is only partial face overlap between the selected face and target parts)

Contact Guidelines and Examples

Before assigning contact to your Mechanical–Thermal design, be aware of the following important guidelines:

- You can assign contact only to the following entities:
  - **Faces** of solid objects
  - **Faces** of shell objects (that is, sheets objects with the **Shell Element** option selected)
  - Shell **Objects** (equivalent to assigning contact to all faces of a multifaced shell)

    Thermal contact can exist between a shell object and one or more adjacent solid objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal contact between two shell objects is not currently supported:

In order for heat to conduct from one shell object to another, they must intersect at a common edge. The default contact in this case is bonded (without any resistance to the flow of heat). If two shell objects share only some overlapping face area but do not have at least one common edge, no heat flow will occur between the two objects. That is, the default bonding (with zero thermal resistance) that occurs between two solid objects, or between a solid and shell object, does not occur between two shell objects.

- If the same conductance or resistance value is applicable to multiple contact faces or multiple shell contact objects, you can assign contact for all of the entities in a single operation. It is not necessary to assign each contact area separately, except when the conductance or resistance values differ.

  Important:

  - Thermal contact is applicable to the following types of object interfaces:
    - A flat face of a solid body and a coplanar face of an adjacent solid body
    - A curved face of a solid body and a concentric curved face of another solid body (for example, a solid rod inserted into a hole in another object).
    - A face of a solid body (flat or curved) and a coplanar or concentric shell (face or object)
    - A shell object sandwiched between two solid objects, where all three faces are coplanar or concentric
  - Though contact involves two or three faces, you only select **one** face or shell object for each contact area. The assignment face or shell object of the contact pair is the "**Contact** face," and the coplanar or concentric face of each adjacent object it touches is the "**Target** face."

    For thermal solutions, you must always select **only** the ***Contact* faces,** and **not** the ***Target* faces**. The solver automatically determines the target faces when you run the analysis.

- One *Contact* face can have multiple *Target* faces.

  For example, a single semiconductor substrate might have many separate dies in contact with it. Selecting the *Contact* face of the substrate is sufficient to define contact for all of the dies touching it (assuming that the same thermal resistance properties are suitable for every contact face). This method differs from many finite element analysis (FEA) applications that require selection of both entities in a contact pair, but it can be a great convenience when one object's face is in contact with multiple adjacent objects. For a fuller understanding, review the examples in the subtopics referenced under *Contact Examples* below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you select multiple faces for a contact assignment and specify a total *Conductance* or *Resistance* value, that total value is applied to the combined area of all selected faces; it is **not** treated as a per-face value.
- Contact can occur along an entire face or only a portion of a face. Also, a contact face may correspond to target faces on multiple objects, with each target involving either a full face or a portion of a face. Review the examples in the subtopics referenced under *Contact Examples* below.
- For a contact definition to be valid, at least a portion of the selected face or faces must be in physical contact with one or more faces (or a portion of one or more faces) belonging to a different body.
- If two solid bodies or a solid body and shell object touch, zero resistance (or perfect conductance) is assumed between the two objects unless overridden by a thermal contact assignment. You do not need to assign contact to support heat flow between objects unless you want to introduce thermal contact resistance.

Contact Examples

Examples and guidelines are divided into the following two subtopics:

- Solid-to-Solid Contact Examples
- Shell-to-Solid Contact Examples

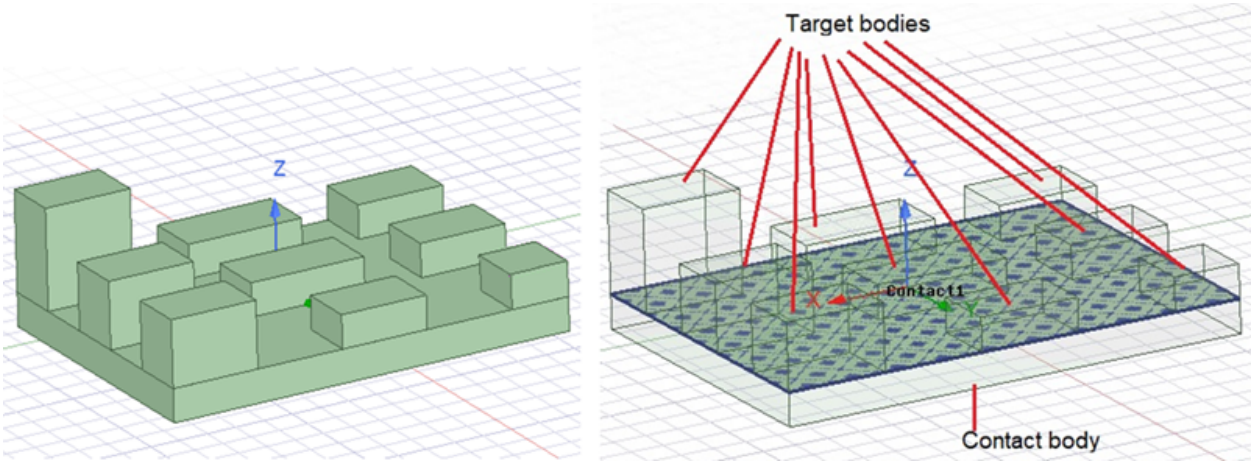
Solid-to-Solid Contact Examples

The following examples involve contact between faces of one solid body and coplanar or concentric faces of an adjacent solid body.

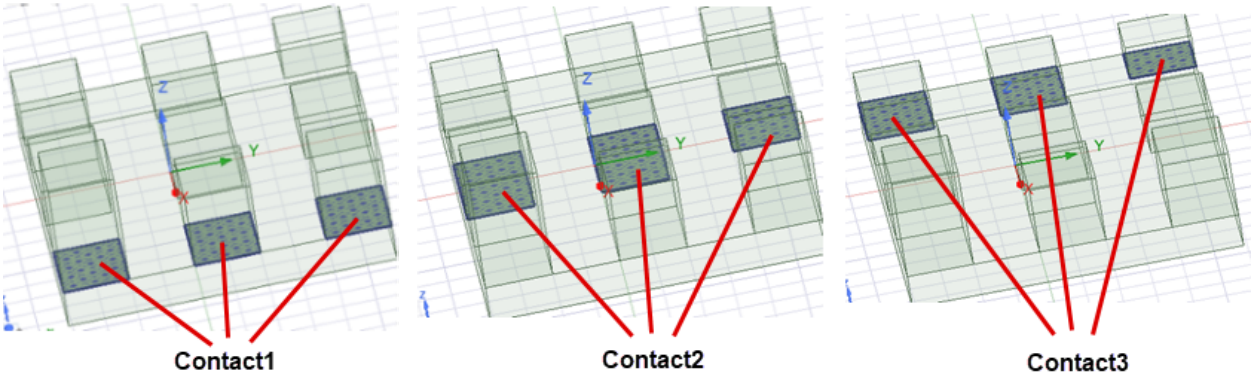
- **Example 1:** Multiple small bodies in contact with one larger body (for example multiple semiconductor dies on a common substrate):

  If the thermal resistance properties are the same for all contact areas, assign contact to the face of the large object using either the **Conductance per Area** or **Impedance** option for a uniform contact conductance. Do **not** use the *Thermal Conductance* or *Thermal Resistance* option for this case, since the total contact area differs from the area of the selected face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



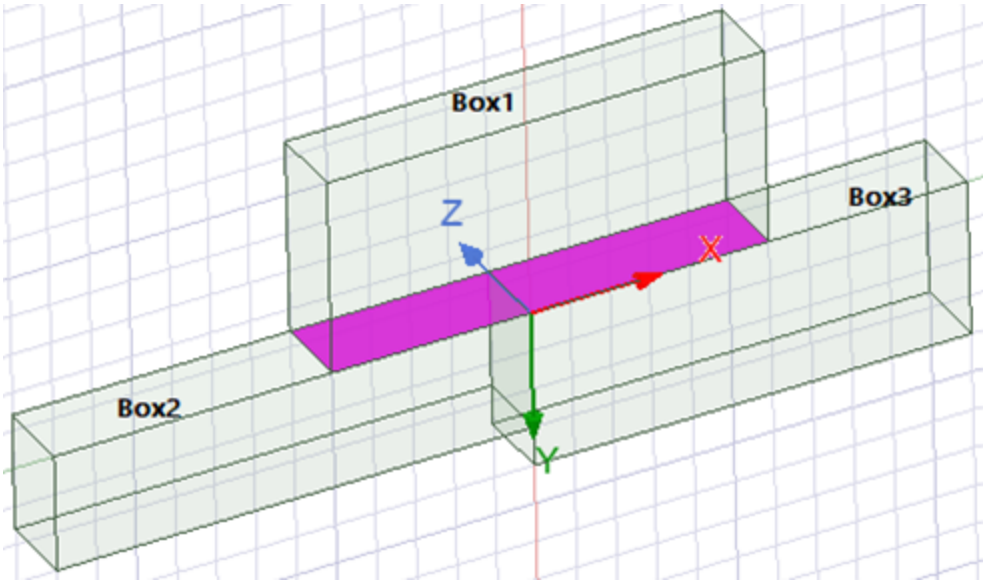
Alternatively, if thermal resistance properties differ among the smaller objects, select faces with the same thermal resistance properties and define contact separately for each set of such faces. If the total contact area matches the total area of the selected model faces and the total conductance or resistance is known for the total area, any resistance type can be used to define the contact. Otherwise **Conductance per Area** or **Impedance** options are recommended:



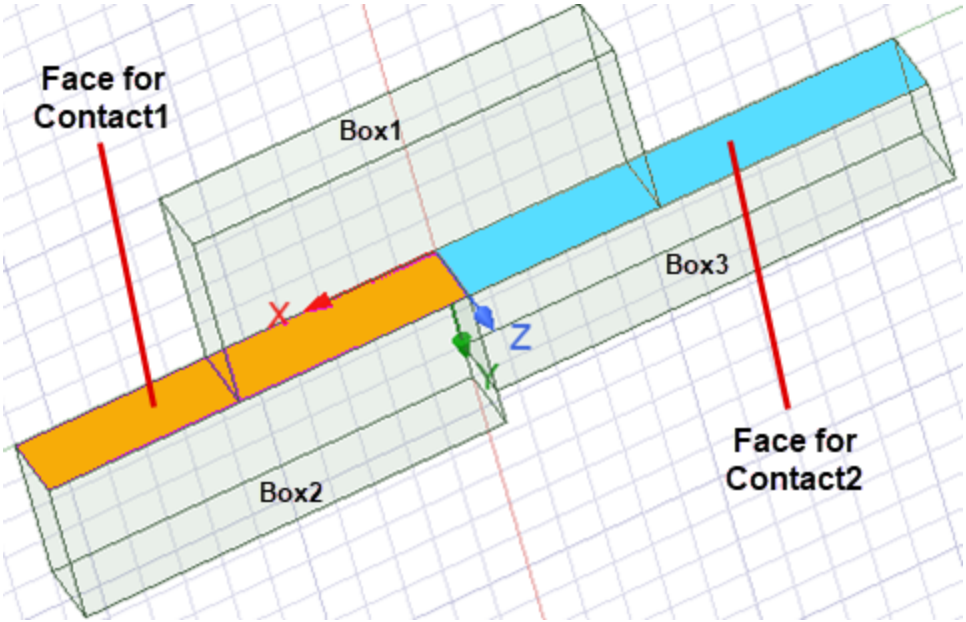
- **Example 2:** A contact body partially overlaps multiple target body faces:

  When the thermal resistance properties are the same for all overlapping face areas, assign contact to the common face (bottom of Box1 in this example). Since the total contact area matches the area of the selected face for this example, you can choose any resistance type for defining the contact:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the thermal resistance properties differ, assign contact separately to each of the non-common faces (top of Box2 and top of Box3 in this example) using either the **Conductance per Area** or **Impedance** option. Do **not** use the *Thermal Conductance* or *Thermal Resistance* option for this case, since the total contact area differs from the area of the selected face:



Shell-to-Solid Contact Examples

The following guidelines involve contact between a shell object, or selected faces of a multifaced shell object, and coplanar or concentric faces of one or more adjacent solid objects. Three main variations are considered:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A single shell in contact with a single solid object
- A single shell, which may represent an insulator or intermediate material, sandwiched between two solid objects
- A single shell object sandwiched between one large solid object and two or more smaller solid objects (used for the convenience of assigning contact at multiple interfaces with a single-object selection)

When a shell represents a thin object placed between two solids, such as the plastic or mica insulators often used to electrically isolate semiconductors from a heat sink, remember that there are actually three resistances to the flow of heat between the two solids:

- Contact resistance between the solid object on one side of the shell and the face of the shell
- The limited conductivity of the shell material through its thickness
- Contact resistance between the solid object on the opposite side of the shell and the face of the shell

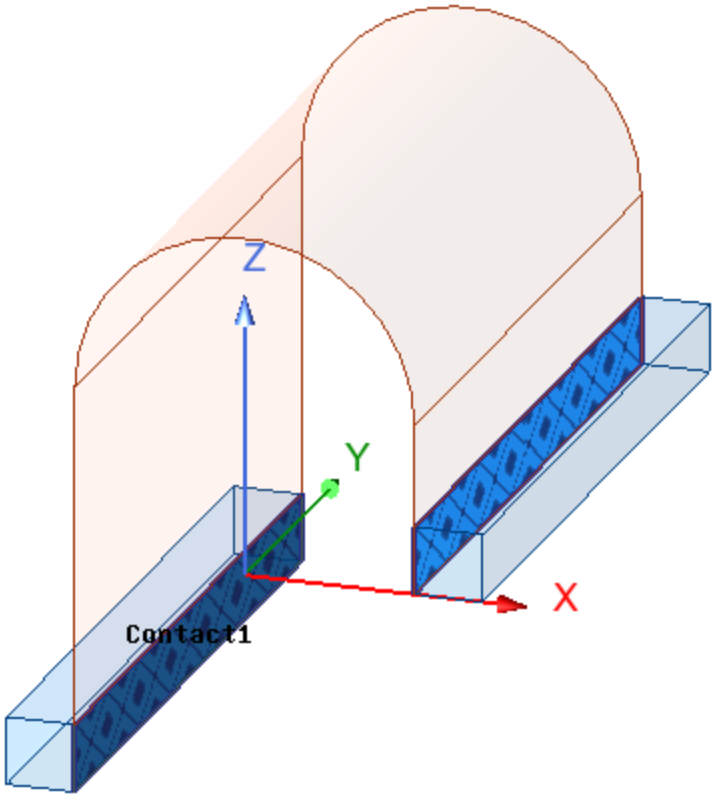
Accounting for all three resistances requires that you correctly calculate the effective conductance or resistance value to specify in the assignment. See the examples that follow for more detail.

Additionally, a shell may be used only as a means of conveniently assigning contact between multiple small solid objects and a single larger solid object (such as many surface-mount components on a printed circuit board. In this case, you do not want to consider three resistances as described in the preceding insulator case. Since the software will effectively double the resistance you specify by assuming that it applies to the shell-to-solid interfaces on both sides of the shell, you must calculate the effective resistance to assign differently than for the insulator case. Again, see the examples that follow for more detail.

Examples and guidelines for various scenarios are as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

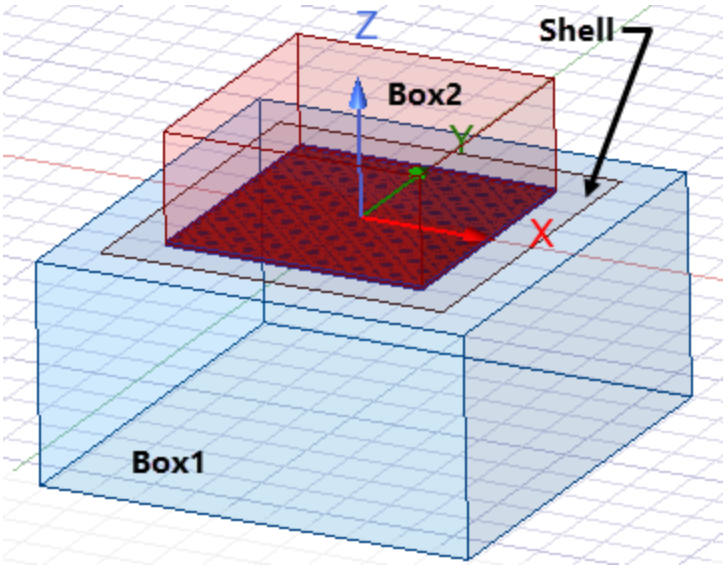
- **Example 1:** Contact between a single solid object and a single shell:



In this example, a single contact assignment is used for the face of each box that is in contact with the shell object. Since the actual contact area matches the total assignment face area, you can choose any *Resistance Type*. If you choose *Thermal Conductance* or *Thermal Resistance*, the specified value represents the total combined contact area of both selected faces (not the conductance or resistance per contact face).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Example 2:** A shell is sandwiched between two solids, and you wish to treat it as an electrical insulator:



You can assign contact to one of the solid object faces or to the shell object or face, which becomes the primary *Contact* face. The solver will find the other two adjacent solid or shell faces, and these will be the *Target* faces.

If you plan on specifying a total resistance or conductivity value, it is best to assign contact to the smallest of the three faces so that the contact area matches the selected face area. Otherwise, if you will used area-based options (*Thermal Impedance* or *Thermal Conductance per Area*), it doesn't matter which face you choose for the contact assignment.

The resistance you specify is doubled, or the conductivity is cut in half, because the solver will apply the resistance to contact interfaces on both sides of the shell. This behavior is consistent for all five *Resistance Type* options, including *Thickness*. That is, the thickness you specify is effectively doubled because the equivalent resistance calculated from the material conductivity and thickness is applied to contact interfaces on both sides of the shell.

When calculating the effective thermal contact resistance to specify in the contact definition, include the following two constituent values:

- The resistance or conductivity at a single solid-to-shell interface.
- The equivalent resistance or conductivity of the shell material for **half of its thickness**.

The solver will double the specified resistance, effectively accounting for **two** contact interfaces and the **full** thickness of the shell object.

Avoid using the *Thickness* option for the *Contact Resistance Type* unless you only want to consider the shell material's limited thermal conductivity and are assuming there is no

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

contact resistance. An example of this usage is when the shell represents an epoxy film used to bond a component to a PCB or substrate, where the epoxy fills in surface roughness and provides a full contact area, mitigating thermal contact resistance. In this case, specify **half** of the actual epoxy film thickness, since the solver will double the equivalent resistance. Note that, if the shell is omitted, you can assign contact directly between the two solids, in which case you would use the full film thickness. Since there's no shell, there's a single contact interface, and the resistance is not doubled.

The following equations show how to calculate an effective *Impedance* value to assign to a solid-shell-solid sandwich so that the material's resistance and thermal contact resistance at both sides of the shell are included correctly:

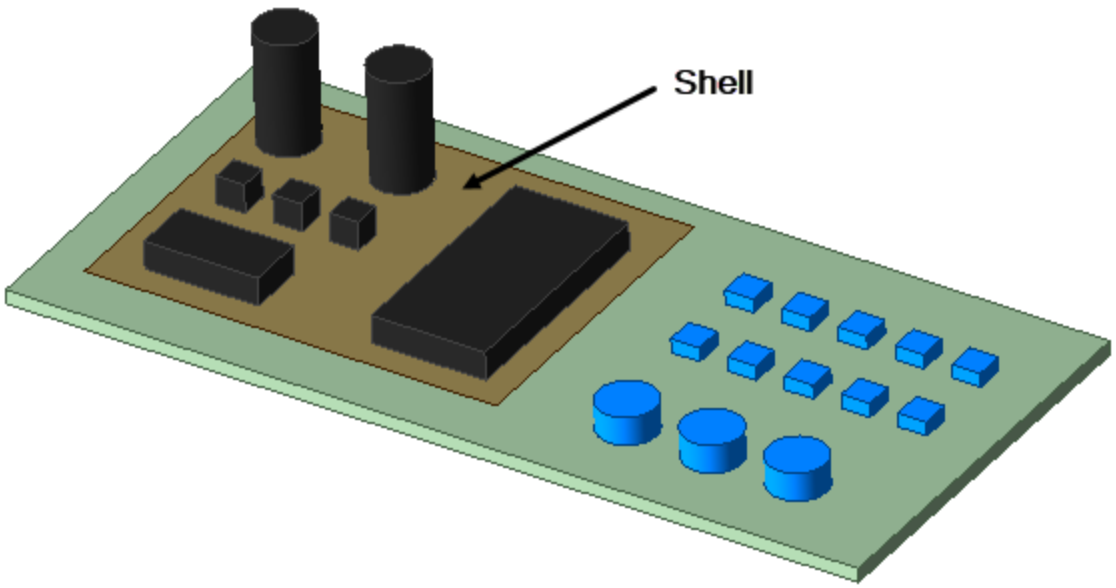
Assume:

- Contact impedance per contact interface, $\mathbf{Impedance_{contact}}$ = 97 °C·mm^2/W
- Material conductivity, **k** = 0.0006 W/(mm·°C)
- Insulator thickness, **Thk** = 0.076 mm

Therefore:

- Effective Impedance of shell material:
  $\mathbf{Impedance_{half\text{-}material}}$ = Thk/2 * 1/k = Thk / 2k
  So, $Impedance_{half\text{-}material}$ = 0.076 mm / [2 * 0.0006 W/(mm·°C)] = 63.3 °C·mm^2/W
- Finally, the *Impedance* to assign = $Impedance_{contact}$ + $Impedance_{half\text{-}material}$
  = 97 + 63.3 = **160.3 °C·mm^2/W**

- **Example 3:** A shell object is sandwiched between one large solid object and multiple small solid objects. (In this case, the shell is used for the convenience of assigning solid-to-solid contact to multiple objects based on a single-shell selection):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For this example, assume that the blue components on the right half of the model are bonded to the substrate without thermal resistance. The black components on the left half of the model are mounted to the substrate with a common thermal contact resistance assigned. Since approximately half of the substrate face has bonded contact, and the other half has areas of thermal contact resistance, contact cannot be assigned to this face. You could select all of the individual component bottom faces on the left half of the board for assigning contact. However, it might be more convenient to draw a shell object that encompasses all of the component-to-substrate interfaces to which you wish to assign contact. In this example, we use the latter approach, so the contact assignment includes only the shell object or its face.

For this application, we don't want to include the resistance of the shell material across its thickness. We also don't want the resistance to be doubled (representing contact interfaces on both sides of the shell). The solution is to specify either half of the contact resistance or double the contact conductivity in anticipation of the solver imposing the resistance twice.

Since the area of the shell face is much greater than the actual contact area, do **not** specify total *Thermal Resistance* or total *Thermal Conductance*. Instead, use an area-based contact resistance type (*Thermal Conductance per Area* or *Thermal Impedance*).

The following equations show how to calculate an effective *Conductance per Area* value to assign to a solid-shell-solid sandwich so that it correctly represents the thermal conductance that would occur between the components and substrate through a single contact interface (ignoring the shell):

Assume:

- Contact conductance per area, **Unit Conductance** = 0.103 W/(°C·mm^2)

Therefore:

- Effective contact conductance per area:
  Unit Conductance$_{Eff}$ = Unit Conductance * 2 = **0.206 W/(°C·mm^2)**

> **Note:**
>
> To prevent the shell from providing a redundant thermal conduction path along the surface of the substrate, specify an infinitesimal thickness in the shell object attributes and specify a material with negligible thermal conductivity.
>
> - The default shell object thickness is 1e-7 mm, which is less than a micron and is acceptable.
> - Define a custom material with a thermal conductivity of 1e-6 W/(m·°C), which is orders of magnitude lower than the conductivity of air and therefore insignificant.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Assign Thermal Contact

It is important that you read and understand the Contact Guidelines and Examples topic before attempting to assign contact to your model. Doing so will help to ensure that the appropriate object faces are selected and that thermal resistance is correctly defined. The procedure for assigning thermal contact is as follows:

1. Select the contact assignment geometry according to one of the following two methods:
    - Select one or more **faces** of solid objects that are touching one or more adjacent solid objects or one or more adjacent shell objects.
    - Select one or more shell **objects** or shell **faces** that are touching one or more adjacent solid objects. Shell-to-shell thermal contact assignments are not currently supported.

      Selecting a shell object is equivalent to selecting all of its faces. For a single-face shell, the face and object selection methods produce equivalent contact definitions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

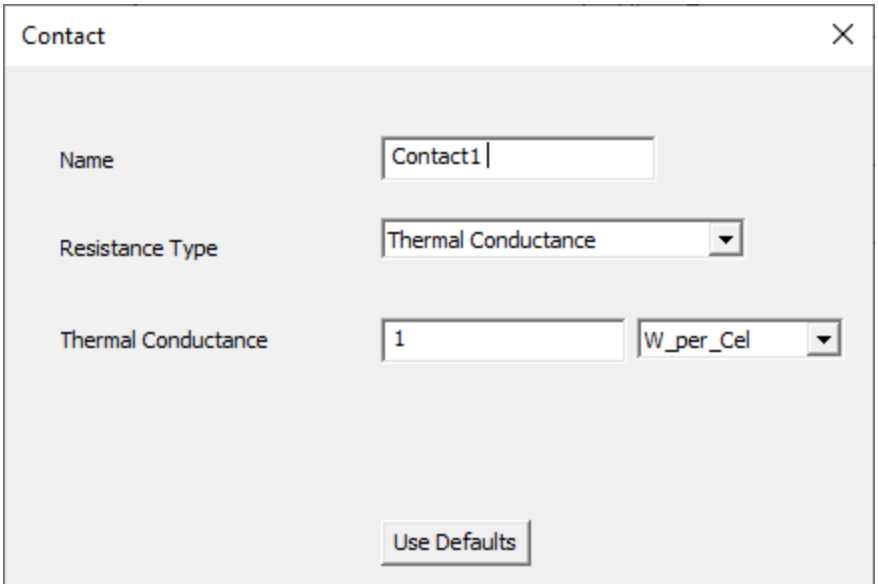
Important:

- Select **only the primary *Contact* faces** (or shell objects) and **not the *Target* faces**. For a fuller explanation, see the guideline concerning contact assignment in the *Contact Guidelines and Examples* topic.
- You cannot assign contact to the same face to which a convection boundary has been assigned nor to a face that fully or partially overlaps a convection boundary face. An error message will be produced if you attempt to do so. If convection is assigned to an object, that boundary is equivalent to the convection being assigned to all faces of the object. Therefore, that object's faces and any overlapping adjacent object faces are restricted from receiving a contact assignment. The reasons for this restriction are the following:
  - Portions of faces that are in contact with other objects are not exposed to the ambient environment and therefore cannot lose heat by convection. Allowing convection at part-to-part interfaces would exaggerate the convective heat flow and skew the temperature and heat flux results.
  - A fields summary would report the wrong convective heat flow results at a convection boundary if a portion of that boundary were to be conducting heat directly to another part. Similarly, heat flux and heat flow results would be skewed at areas of overlapping contact and convection.

  You can use the imprint tool to split the contact portion of a face from the portion to receive a convection boundary, thereby creating two or more separate faces from one original model face.

2. Access the *Contact* dialog box using *one* of the following three methods:
   - In the Project Manager, right-click **Contacts** and choose **Assign** > **Contact** from the shortcut menu.
   - Right-click in the Modeler window and choose **Assign Contact** > **Contact**.
   - From the menu bar, click **Mechanical** > **Contacts** > **Assign** > **Contact**.

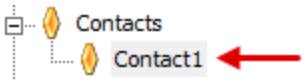
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Optionally, type a new **Name** to substitute for the default one.
4. Choose the desired **Resistance Type** from the drop-down menu. The choices are as follows:
   - **Thermal Conductance**
   - **Thermal Conductance per Area**
   - **Thermal Resistance**
   - **Thermal Impedance**
   - **Thickness**

   For more information on each type, see the Thermal Resistance Types topic.
5. Depending on the resistance type selected in step 4, do one of the following:
   - Type in the value of **Thermal Conductance**, **Conductance per Area**, **Thermal Resistance**, or **Thermal Impedance** and choose the units from the drop-down menu to the right of the text box.
   - For the *Thickness* resistance type:
     a. Specify the **Thickness** and choose the unit from the drop-down menu to the right of the text box.
     b. Click the **Material** button to access the *Select Definition* dialog box.
     c. Choose the material from one of the libraries or define a custom material. Then click **OK** to close the *Select Definition* dialog box.
6. Click **OK** to complete the definition and close the *Contact* dialog box.

   The contact appears in the Project Manager:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Setup (Thermal)

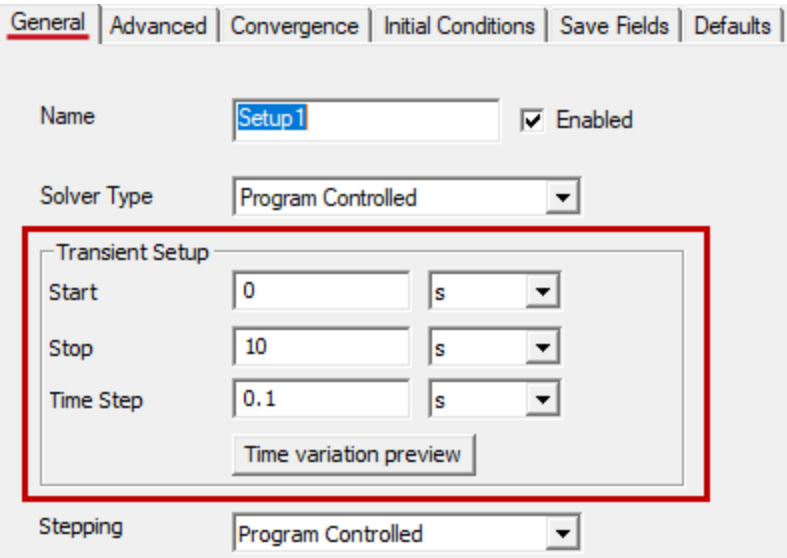
You must add at least one solution setup to each Mechanical design. The solution setup for a *Thermal* analysis includes the following settings:

[General Tab]

- **Name:** The default name is Setup*x*, where *x* is a numeric value incremented for each solution setup that you add to the design. You can customize the name of each setup as desired.
- **Enabled:** This option is selected by default. Deselect the option to prevent the setup from being solved when you click the *Analyze All* command.
- **Solver Type:** Specify the solver to use, or let the program choose the solver. The choices are:
  - **Program Controlled:** This is the default setting, which is recommended for most cases. The application selects the optimal solver.
  - **Direct**
  - **Iterative**

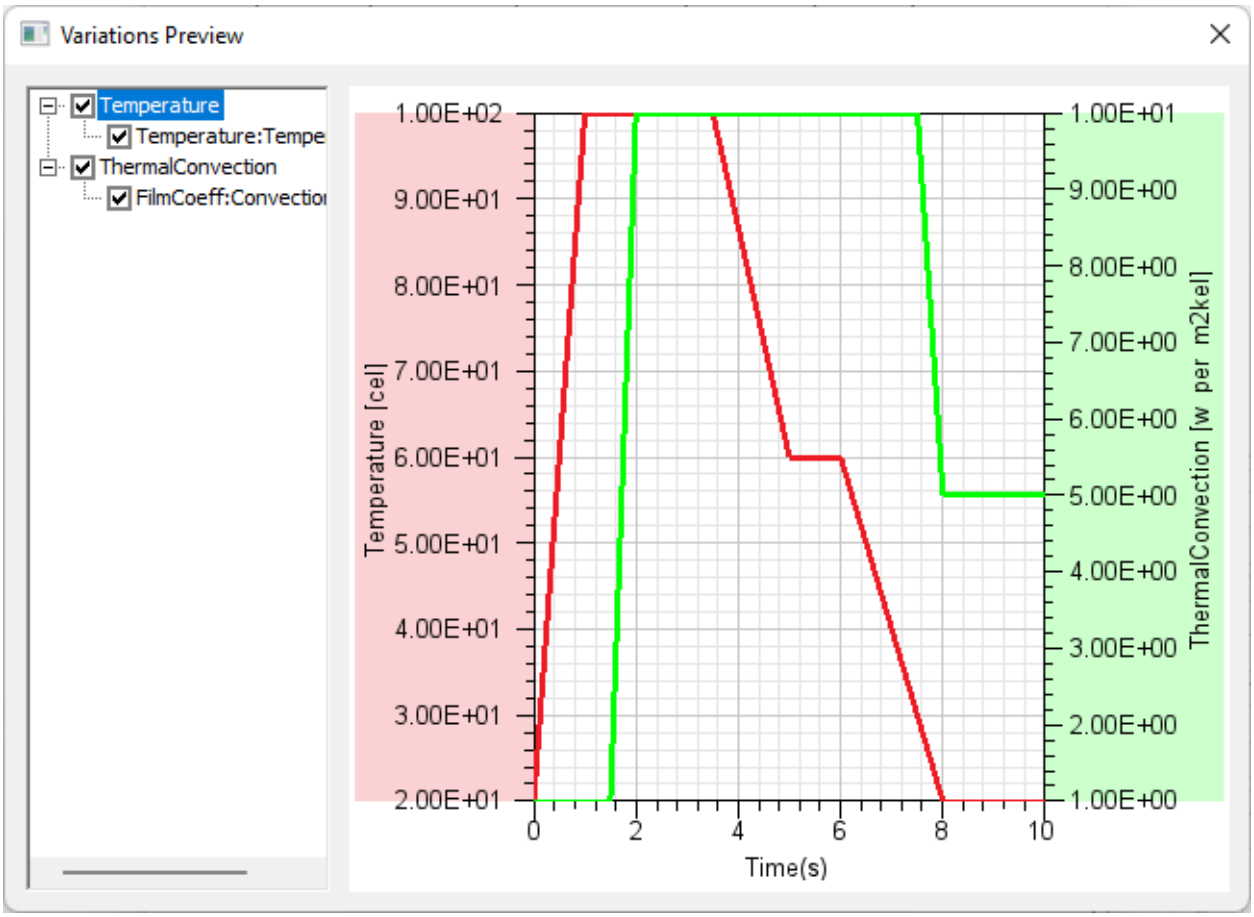
> **Note:**
>
> Consult the Ansys *Mechanical User's Guide* (specifically, *Configuring Analysis Settings* > *Analysis Settings for Most Analysis Types* > *Solver Controls*) for more information about the available solver choices.

- **Transient Setup:** For *Transient Thermal* solutions, you must specify the **Start** time, **Stop** time, and **Time Step** of the simulation in the *Transient Setup* section:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optionally, click **Time variation preview** to see a plot of time-varying boundaries or constraints:



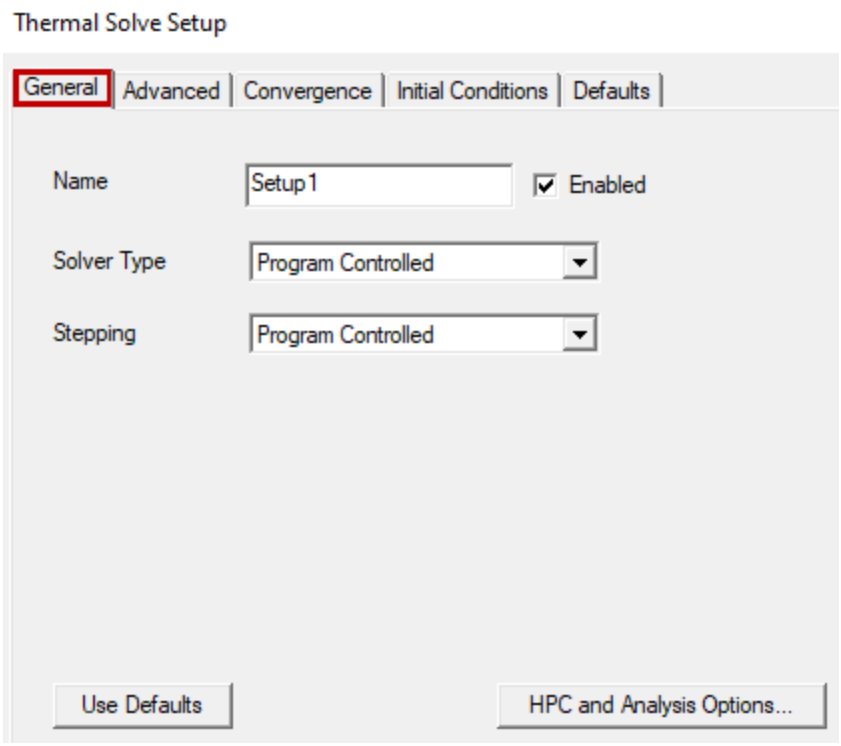
- **Stepping:** Specify whether stepping is enabled, or let the program choose. Stepping is the process of dividing the applied loads (excitations and boundaries) into smaller increments to make solution convergence easier. In the context of steady-state thermal analyses, only the loads are divided, not time increments, since the time to achieve steady-state conditions (that is, thermal equilibrium) is unknown.

  The *Stepping* options are:

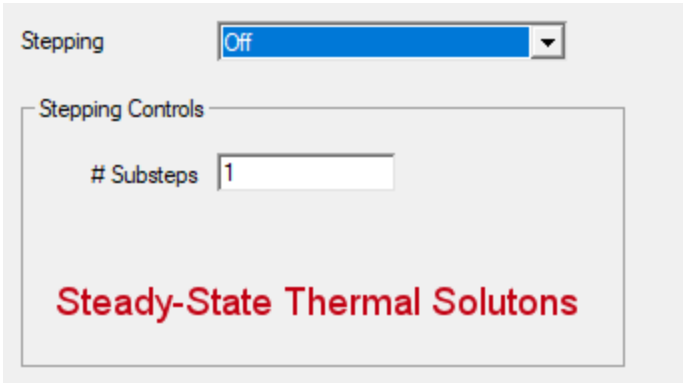
  - **Program Controlled:** This is the default setting, which is recommended for most cases. The application determines the stepping settings.

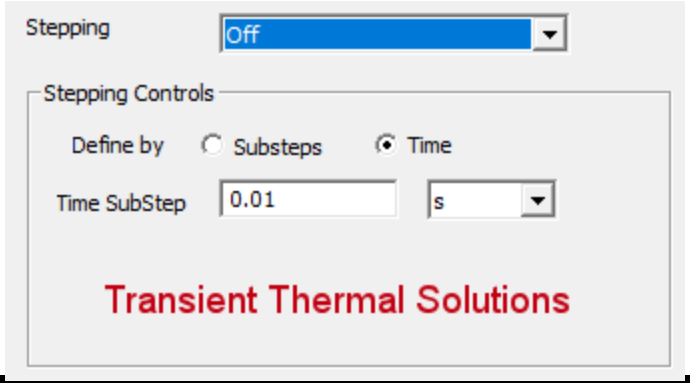
    There are no additional settings to specify in the General tab for *Steady-State Thermal* solutions when this *Stepping* option is selected:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Off:** Program controlled or user-defined automatic stepping is disabled. However, you can still specify a fixed number of calculation **Substeps** (steady-state or transient solutions) or **Time Substeps** (transient solutions only). When this option is selected, the following additional setting appears in the dialog box:





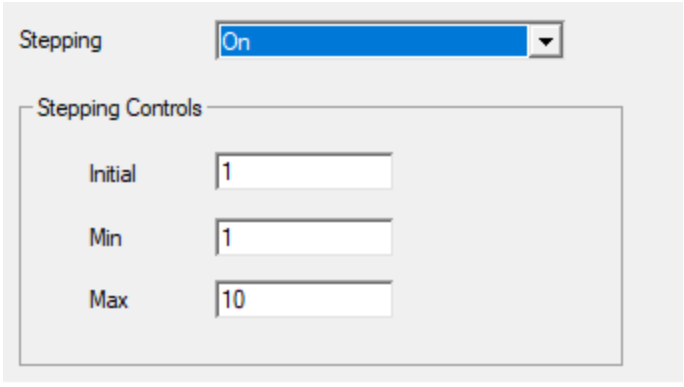
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **# Substeps:** The number of substeps to solve for a steady-state solution or the number to solve per requested time step in a transient solution. In a steady-state solution, you can think of a substep as a load division for all assigned boundaries and excitations. For example, if you apply a 6 watt heat generation excitation, and you specify two substeps, the solver increments the generated heat by 3 watts for each calculation substep. That is, the solution is calculated at 3 W and 6 W of heat input. The applied thermal loads (boundaries and excitations) are divided according to the number you specify and solved as intermediate steps and a final step.

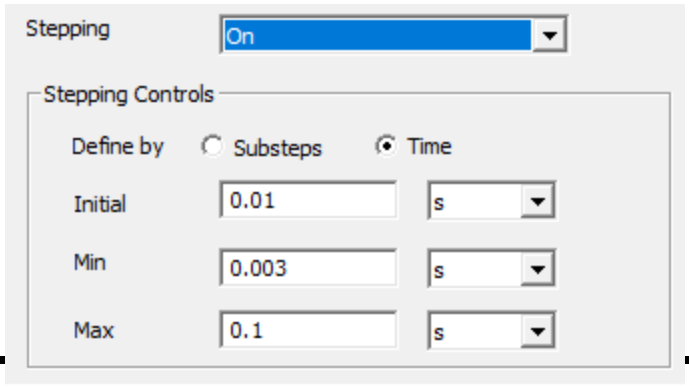
  Substeps can have a similar effect for transient solutions if a thermal source or load is in transition at the time step being divided into smaller pieces. This effect combined with the shorter calculation time increment can improve the ease of solution convergence. (See Note below.)

- **Time Substeps:** In a transient solution, time substeps reduce the duration of the default solution *Time Step* specified in the Transient Setup section. The effect is similar to that of specifying substeps except that you are directly defining the calculation time increment rather than the number of substeps. Any thermal sources or loads that are in transition at the time of a given solution step are analyzed in smaller, partial changes in magnitude. This load change reduction, and the shorter calculation time increment can lead to easier convergence of the solution. (See Note below.)

- **On:** User-defined stepping is enabled. When this option is selected, the following additional settings appear in the dialog box:



Steady-State Thermal Solutions



Transient Thermal Solutions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Initial:** The initial number of substeps to run, or initial substep time increment (for transient solutions only).

  For steady-state solutions: The applied thermal loads (boundaries and excitations) are divided according to the initial number of substeps to represent a "bite-sized" portion of the full thermal load. The default *Initial* substeps value is 1, in which case the full thermal load is applied initially. This number should be greater than or equal to the *Min* number of substeps (next bullet).

  For transient solutions: The default *Initial* time substep is 0.01s. Alternatively, you can choose to specify the initial number of substeps. in which case the default is 100.

- **Min:** The minimum number of substeps to be solved or the shortest substep duration (for transient solutions only). The default is 1 for steady-state solutions and either 0.003s or, alternatively, 10 substeps for transient solutions.

  If no convergence problems are encountered, the solver can reduce the number of substeps to something between the *Initial* and *Minimum* values, essentially increasing the size of the load step. For transient solutions using the *Time* option, the duration of the substep is increased to something between the *Initial* and *Maximum* values to achieve the same effect.

  If you specify *Min* greater than *Initial*, the solver will reset *Min* to be equal to *Initial*. Therefore, the value you enter for *Min* should be less than or equal to the *Initial* number, or duration, of substeps you define.

- **Max:** The maximum number of substeps to solve or the maximum substep duration (for transient solutions only). The default is 10 for steady-state solutions and either 0.1s or, alternatively, 100 substeps for transient solutions.

  More substeps than specified by the *Initial* setting (or substeps with a greater duration) are solved only if convergence problems are encountered. A greater number of calculation substeps (or a shorter substep duration) corresponds to smaller load steps, making convergence easier.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- While results are produced by the solver for each calculation substep, these intermediate results are not available for reporting in the Ansys Electronics Desktop application. Only the final solution results (for a steady-state solution) or results at the requested *Time Step* interval (for transient solutions) are available for post processing (see Save Fields Tab below).
- In the context of a steady-state thermal analysis, you are not dividing the analysis into periods of time. You are only specifying load substeps, each of which has a steady-state solution of unknown time.
- In the absence of nonlinear materials, radiation (which is highly nonlinear), and time-dependent or temperature-dependent excitations or boundaries, thermal analyses is likely to converge in a small number of steps without difficulty. Therefore, we recommend that you use the **Program Controlled** option, which allows up to ten substeps and as few as one.

[Advanced Tab]

Use this tab to set up Mesh Linking when desired.

[Convergence Tab]

This tab contains optional solver controls and tolerance values for temperature and heat convergence.

[Initial Conditions Tab]

Use this tab to define the initial object temperature the solver is to use for the analysis. An appropriate initial temperature can shorten the solution time.

[Save Fields Tab] (Transient Thermal Solutions only)

Use this tab to choose whether to save fields data and at what frequency to save the data when wanted.

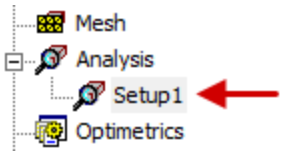
How to Set Up a Thermal Solution:

1. Use one of the following three methods of accessing the thermal solution settings:
   - On the **Simulation** ribbon tab, click **Setup**.
   - Right-click **Analysis** in the Project Manager and choose **Add Solution Setup**.
   - Using the menu bar, click **Mechanical** > **Analysis** > **Add Solution Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Optionally, change the **Name** and any of the other default general settings (**Solver Type**, **Stepping** or **Transient Setup** options) that have been previously described in this topic.
3. Optionally, click **HPC and Analysis Options** to change any of your high performance computing configurations or options.
4. Optionally, specify settings for importing a mesh from another design or project. (See Mesh Linking.)
5. Optionally, specify user-defined options for temperature and heat convergence.
6. Optionally, specify an initial condition (starting temperature) for the solution.
7. Optionally, for transient thermal solutions, specify whether to save fields and how frequently to save them.
8. Click **OK**.

   The setup appears under *Analysis* in the Project Manager:



Convergence Tab

When solving nonlinear thermal analyses, an iterative procedure (equilibrium iterations) is carried out at each substep. A successful solution is indicated when the out-of-balance loads are less than the specified convergence criteria.

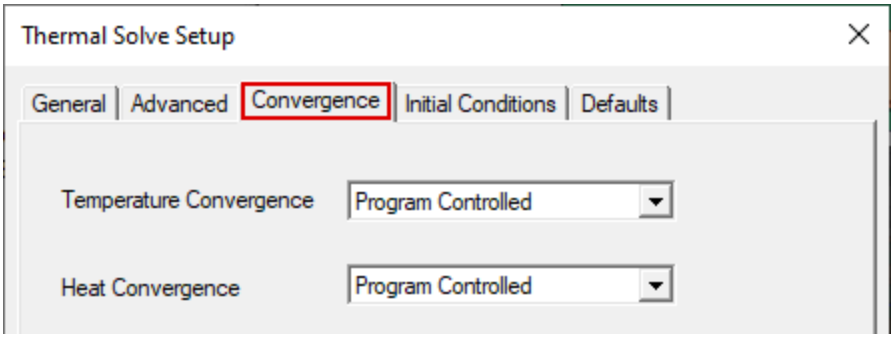
The *Convergence* tab includes solver control settings regarding temperature convergence and heat convergence of the solution. The default option for each (**Program Controlled**) is suitable for the majority of thermal solutions (both those that are fully linear and those with nonlinearities). The alternative settings in the *Convergence* tab provide a means of troubleshooting solutions that fail to converge. The available settings are as follows:

- **Temperature Convergence:** A temperature convergence criterion is enforced for every thermal solution and cannot be disabled.

  The solver calculates a *Reference VALUE* based on the residual of the current iteration. This automatically calculated *Reference VALUE* is used in the determination of the convergence criterion, unless superseded by the **Minimum Reference** value (defined below). The *Reference VALUE* is re-evaluated for each solution iteration regardless of whether program controlled or user-defined options are specified. The available temperature convergence options are:

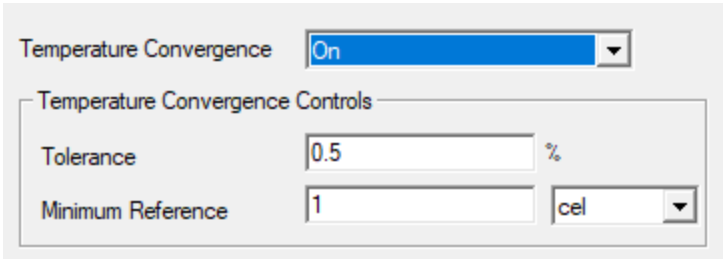
  - **Program Controlled** (default option): The automatically calculated *Reference VALUE*, or alternatively the default **Minimum Reference** value, and the default **Tolerance** (defined below) are used to calculate the temperature convergence criterion for each iteration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **On:** Overrides the automatic setting with the following two user-defined parameters:
  - **Tolerance:** A percentage value that is a factor to be multiplied by either the automatically calculated temperature convergence *reference VALUE* or the specified *Minimum Reference* value (next bullet), whichever is greater. This input is not to be confused with the typical sense of the word "tolerance" (a direct indication of allowable +/- error). In this context, it is a factor that adjusts the tightness of the convergence criterion, not the convergence criterion itself. As with any tolerance, the smaller the value, the tighter the resultant criterion and the more precise the solution must be.
  - **Minimum Reference:** The minimum allowed *reference VALUE*, in temperature units, to be used alternatively for the calculation of the temperature convergence criterion. The *Minimum Reference* value is used when it is greater than the automatically calculated temperature convergence *reference VALUE*.

  (See note below.)



- **Heat Convergence:** Enforcing heat convergence may be enabled or disabled. When disabled, convergence is based solely on the temperature convergence criterion. When enabled, both temperature and heat flow calculations must satisfy their respective convergence criteria.

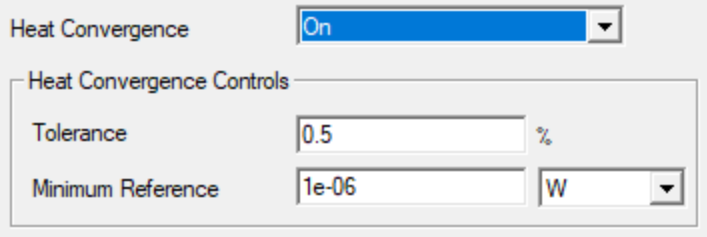
  The solver calculates a *Reference VALUE* based on the residual of the current iteration. This automatically calculated *Reference VALUE* is used in the determination of the convergence criterion, unless superseded by the **Minimum Reference** value (defined

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

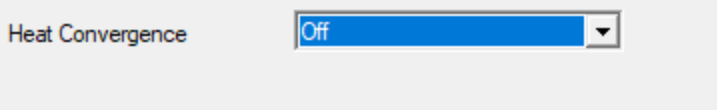
below). The *Reference VALUE* is re-evaluated for each solution iteration regardless of whether program controlled or user-defined options are specified. The available heat convergence options are:

- **Program Controlled** (default option): The automatically calculated *Reference VALUE*, or alternatively the default **Minimum Reference** value, and the default **Tolerance** (defined below) are used to calculate the heat convergence criterion for each iteration.
- **On:** Overrides the automatic setting with the following two user-defined parameters:
  - **Tolerance:** A percentage value that is a factor to be multiplied by either the automatically calculated heat flow convergence *reference VALUE* or the specified *Minimum Reference* value (next bullet), whichever is greater. This input is not to be confused with the typical sense of the word "tolerance" (a direct indication of allowable +/- error). In this context, it is a factor that adjusts the tightness of the convergence criterion, not the convergence criterion itself. As with any tolerance, the smaller the value, the tighter the resultant criterion and the more precise the solution must be.
  - **Minimum Reference:** The minimum allowed *reference VALUE*, in power units (energy/time), to be used alternatively for the calculation of the heat flow convergence criterion. The *Minimum Reference* value is used when it is greater than the automatically calculated heat flow convergence *reference VALUE*.

  (See note below.)



- **Off:** Do not enforce any heat flow convergence criterion.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For both *Temperature Convergence* and *Heat Convergence*, the **greater** of the following two values is used as the convergence criterion:

- <Automatically calculated *reference VALUE*> * **Tolerance**
- **Minimum Reference** * **Tolerance**

Properly set up and well conditioned thermal models will rarely have difficulty converging. Thermal models that may tend to have trouble converging are those that have materials with near-zero thermal conductivity or materials with highly nonlinear temperature-dependent properties. For example, if a material will encounter a phase change, the temperature-dependent properties will change drastically at the melting/fusion or evaporation/condensation temperatures. The **Minimum Reference** input is useful in such cases because it limits how small the convergence criterion can be. Alternatively, a larger **Tolerance** value also relaxes the convergence criterion, making the solution easier. Adjusting either option can resolve solution convergence problems, assuming convergence failure is not due to improper model setup.

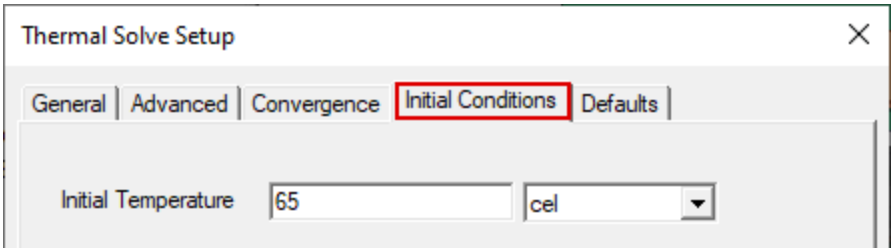
Let the default values guide you concerning what the typical magnitudes should be. For example, if the default Tolerance is 0.5%, try 1% or 2% (not 50%). Similarly, if the default *Minimum Reference* value is 1e-6 W, try 5e-6 W or 1e-5 W (not 0.001). If the convergence criteria are too small, the solution may fail. Conversely, if the criteria are too large, the results may be inaccurate and unsmooth.

Initial Conditions Tab

Under the **Initial Conditions** tab of the *Thermal Solve Setup* dialog box, you can specify a user-defined **Initial Temperature**. This is the global object temperature at the beginning of the solution process. The Initial temperature value is overridden locally by any temperatures boundaries or initial temperatures assigned to the model entities as part of the analysis setup.

- **For Steady-State solutions:** An appropriate initial temperature, approximating the expected prevalent or average temperature results, reduces the number of iterations required to achieve solution convergence. Therefore, the solution time is minimized. This setting has no effect on the steady-state solution's temperature results.
- **For Transient solutions:** Specify the initial temperature that applies at Time = 0. For example, you may wish to start the transient solution at an elevated temperature and analyze the cooling time. Choose a global initial temperature that applies to the majority of model objects to minimize the number of initial temperature assignments needed at selected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The following inputs are supported for the **Initial Temperature**:

- **AmbientTemp:** The predefined variable, which is the **Temperature** value defined in the **Ambient Conditions** tab of the *Mechanical Design Settings* dialog box.
- The name of any **user-defined variable** that contains a temperature value definition.
- A **numeric temperature value** (integer or floating point). Use the adjacent drop-down menu to select the temperature unit for numeric values.

Save Fields Tab

For transient thermal solutions, field results are saved only for the final time step, unless you choose otherwise. Use the settings in this tab to save fields results for every time step, for every n_{th} timestep, or for a varying interval (as defined by a dataset).

- **None:** Only the field results from the final time step will be saved (no intermediate fields). This is the default setting.
- **Every ____ Steps:** Specify the save frequency. A value of **1** will save field results for every timestep. Specifying **2** would save every other timestep, **3** every third, and so on.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additionally, you can specify a pwl (piecewise linear), pwl_periodic (piecewise linear periodic), or pwc (piecewise constant) dataset to vary the fields-saving interval. Use this technique to save field results at finer time increments within periods of interest and coarser increments elsewhere. For more information, see Using Datasets to Define Transient Parameters and Piecewise Constant (pwc) Datasets.

Field results cannot be saved for substeps but only for the time steps specified in the *Transient Setup* section of the *General* tab.

Tip:

Save fields results if you want to animate the fields (such as temperature and heat flux overlays).

If you are mainly interested in how temperatures at specific points vary over time, and you are satisfied with a 2D graph of these results, assign Thermal Monitor points. Saving temperature results at user-specified points is more efficient than saving fields results globally, reducing drive storage requirements and input/output time.

Two-Way Coupling

This feature builds on the ability of Mechanical – Thermal solutions to import EM losses from one or more source designs. The two-way coupling capabilities in Ansys Electronics Desktop (AEDT) allow the temperature results from a Mechanical – Thermal solution to be fed back to an EM loss source design, in which another solution iteration is then performed. Temperature-dependent materials in the source design are affected by the fed-back temperature distribution, thus altering the results of the source solution (and likely changing the EM loss results). A solution controller coordinates the iterative process, next running another iteration of the thermal solution, feeding back the updated temperature distribution to the source design, and so on. The iterative process continues until the user-specified number of coupling iterations has been completed.

Two-way coupling is supported between Mechanical design thermal solutions and any of the design types that can be used as a source of EM loss data. Specifically, the supported design types are as follows:

- **HFSS:** From Modal, Terminal, or Eigenmode solutions – import surface losses from non-solve-inside conducting objects and volume losses from solve-inside lossy objects
- **Maxwell 3D:** Surface losses from *Impedance* boundaries, and volume losses from solve-inside conducting objects.
- **Q3D:** Surfaces losses from conducting objects for the AC field type, and volume losses from lossy objects for the DC field type

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Lossy objects are those with material definitions that include a *Dielectric Loss Tangent* greater than zero.
- For source designs with transient solutions, losses for time steps that are defined as a function (for example, a function of a time) cannot be mapped to a Mechanical thermal solution.

Two-Way Coupling Requirements

Beyond the requirements for importing EM loss results, the following additional requirement must be satisfied in order to perform two-way coupling:

- **For designs acting as EM loss sources:**
  - You must specify a temperature-dependent material for one or more objects.
  - You must select both the **Include Temperature Dependence** and **Enable Feedback** options in the *Temperature of Objects* dialog box.
- **For the Mechanical – Thermal design:**
  - For the coupled EM Loss excitations, you must select both the **Simulate source design as needed** and **Preserve source design solution** options in the *Setup Link* dialog box.
  - You must create a *2-Way Coupling* solution setup (which is the controller for the two-way iterative solution process).
- **Additional restrictions:**
  - The Mechanical – Thermal design is restricted to a single solution setup.
  - Coupled upstream simulations (EM loss source designs) must be driven via the controller in Mechanical. Any simulation you directly initiate at the upstream design will likely cause the next coupled Mechanical simulation to restart from iteration 1.
  - Do not use post processing variables in the upstream designs (for example, post processing variables in the HFSS *Edit Sources* dialog box). When post processing variables are used in upstream designs, the coupled solution can become out of sync with its nominal variation.
  - Do not edit the upstream design during a coupled simulation.
  - An HFSS, Maxwell 3D and Q3D design can only couple with AEDT Mechanical via the AEDT controller. These source designs can be coupled with Ansys Workbench–Mechanical when they are *not* coupled with AEDT Mechanical. A Maxwell 3D design can be coupled with AEDT Mechanical when it is *not* already coupled via Command Line System Coupling or Workbench.
  - Q3D solution setups with more than one problem type enabled are *not* supported.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Frequency sweeps in Q3D solutions are not supported. Imported EM losses are only available at the *Solution Frequency* specified under the General tab of the *Solve Setup* dialog box.

Two-Way Coupling Workflow

Within the following procedure, the terms *upstream* and *downstream* are used to identify the designs or solutions. *Upstream* refers to the design that is the source of the EM loss results. *Downstream* refers to the Mechanical – Thermal design into which the EM loss data is imported and resultant temperatures are computed.

1. Create two designs using identical geometry and object names. One will be a Mechanical – Thermal design and the other must be a design type supported as an EM loss source (HFSS, Maxwell 3D, or Q3D).

   > **Note:**
   >
   > Vacuum or air regions needed for field calculations in the source design may or may not be applicable to your thermal analysis. All thermal design objects must have non-zero thermal conductivity. When you import or copy a vacuum object into a Mechanical – Thermal design, if a nonzero thermal conductivity has not previously been specified in the vacuum material's properties, the material *air* is automatically reassigned to the object. Otherwise, if air or vacuum regions are not relevant to the thermal solution, you can delete them from the Mechanical design or set them as non-model parts. However, note that you will not be able to import the source design's mesh after removing or deactivating any objects within the Mechanical design (the geometry will no longer be identical to the source design).

2. In the upstream design, assign **temperature-dependent** material properties to one or more parts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

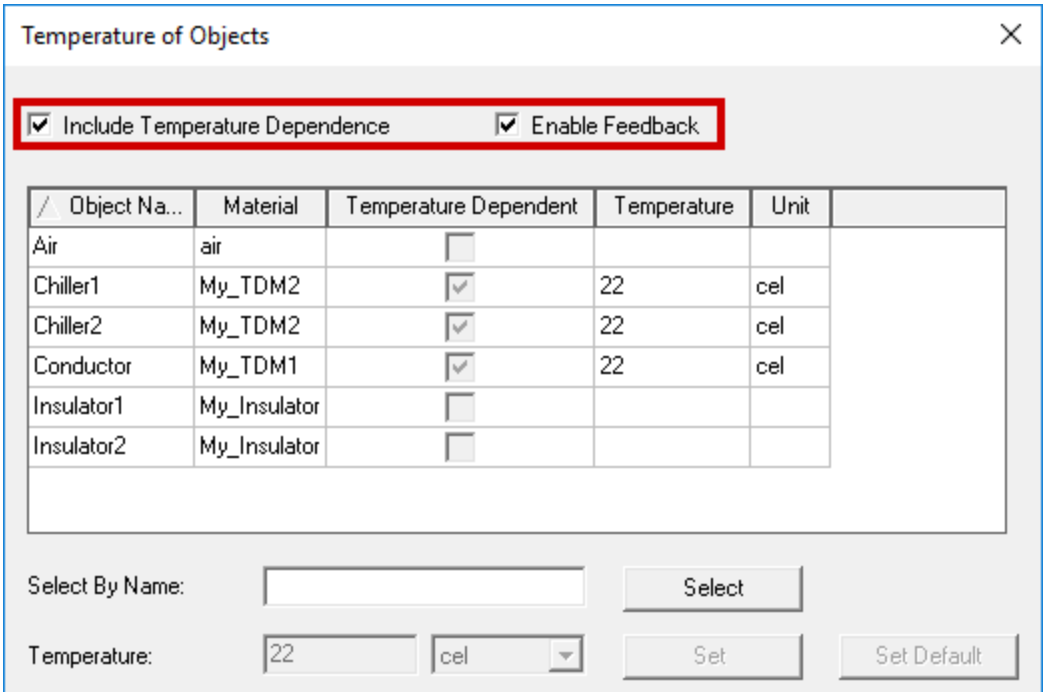
For information concerning the definition of temperature-dependent material properties, consult the following references:

- Search the applicable product's help for **"specifying thermal modifiers"**. (The quotation marks limit search results to instances of the full phrase.)
- For information concerning supported operators and constants that you might want to use in a thermal modifier expression, search applicable products for **"defining an expression"** and also follow the **Related Topics** links on the *Defining an Expression* page.

3. In the Project Manager, right-click the ***upstream design*** and choose **Set Object Temperature** from the shortcut menu.

   Alternatively, you can click the product menu (**HFSS, Maxwell 3D,** or **Q3D** and choose **Set Object Temperature**.

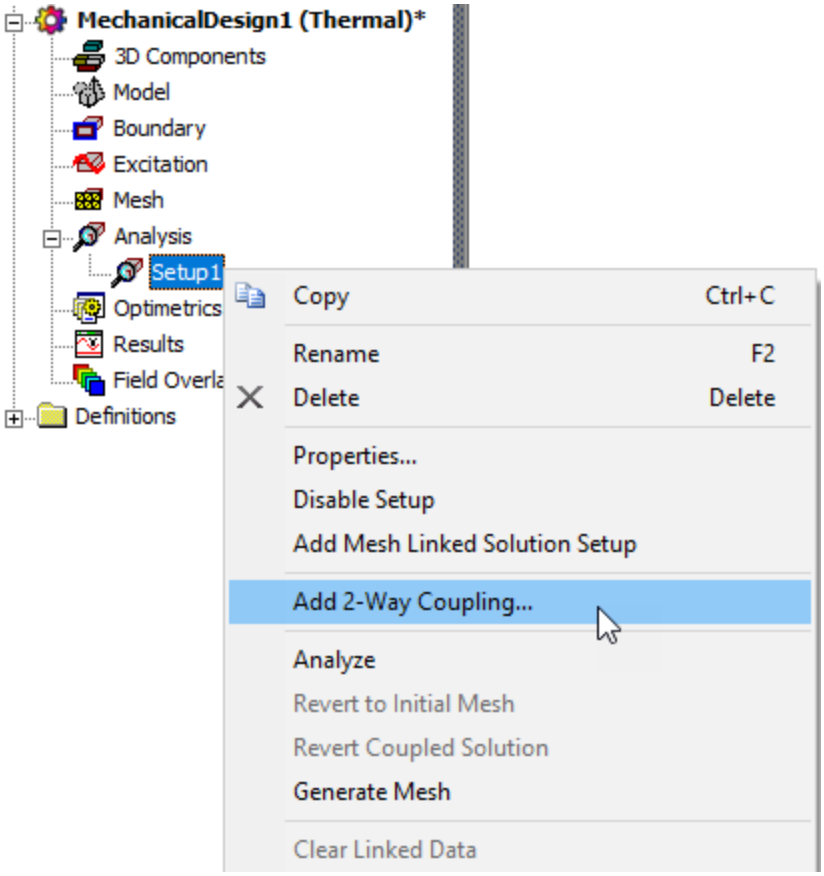
4. In the *Temperature of Objects* dialog box that appears, select the following two options:
   - **Include Temperature Dependence**
   - **Enable Feedback**



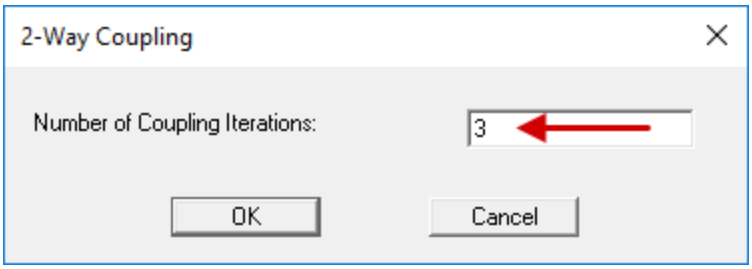
The check boxes in the *Temperature Dependent* column indicate which objects have temperature-dependent material properties assigned to them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the Mechanical – Thermal design, assign an EM Loss excitation to one or more objects, setting up a link to the source design and ensuring that the imported loss is correctly assigned as a *Surface* or *Volume* loss.
6. Ensure that required boundaries, excitations, and solution setups are fully defined for both the upstream and downstream designs.
7. In the Mechanical design, right-click the ***analysis setup*** in the Project Manager and choose **Add 2-way Coupling**:



8. In the *2-Way Coupling* dialog box that appears, specify the desired **Number of Coupling Iterations** and click **OK**. This value must be a positive integer.



9. When all coupled iterations have been completed, right-click the ***mechanical solution setup***, under **Analysis** in the Project Manager, and choose **Profile**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Solutions* dialog box appears with the *Profile* tab selected.

10. Compare the reported **Total Loss on Source Object** values for the last two iterations:

Task	Real Time	CPU Time	Memory	Information
Start				Time: 04/26/2019 09:51:54; Host: CHQMZHAN1; Processor: 20; OS: NT 10.0; Mechanical 2019.3.0
				Executing from D:\AnsysDev\current_2\build_output\64Debug\MechanicalComEngine.exe
				HPC Enabled
Design Validation				Elapsed time : 00:00:01 , MechanicalComengine.exe ComEngine Memory : 101 M
				Perform full validations
Mesh TAU4	00:00:03	00:00:04	72.3 M	16392 tetrahedra "Meshing" Step
Mesh TAU (Coarsening)	00:00:01	00:00:01	72.3 M	12828 tetrahedra
Mesh Post	00:00:04	00:00:06	74 M	16571 tetrahedra
Mesh ACMO	00:00:01	00:00:00	0 K	
Coupling Iteration 1				
Get Heat Source : HeatSource1	00:00:24	00:00:00	0 K	Source: This Project*, HFSSDesign1 - Setup1 : LastAdaptive
				Total Loss on Source Object 'slug1': 0.0282586W, Scaling Factor: 1.05971.
				Total Loss on Source Object 'slug2': 0.0125514W, Scaling Factor: 1.04826.
Solve	00:00:09	00:00:06	173 M	
Temperature Mapping	00:00:01	00:00:00	0 K	
Coupling Iteration 2				
Get Heat Source : HeatSource1	00:00:08	00:00:00	0 K	Source: This Project*, HFSSDesign1 - Setup1 : LastAdaptive
				Total Loss on Source Object 'slug1': 0.0301015W, Scaling Factor: 1.05971.
				Total Loss on Source Object 'slug2': 0.013349W, Scaling Factor: 1.04826.
Solve	00:00:08	00:00:06	173 M	
Temperature Mapping	00:00:01	00:00:00	0 K	
Coupling Iteration 3				
Get Heat Source : HeatSource1	00:00:08	00:00:00	0 K	Source: This Project*, HFSSDesign1 - Setup1 : LastAdaptive
				Total Loss on Source Object 'slug1': 0.0302208W, Scaling Factor: 1.05971.
				Total Loss on Source Object 'slug2': 0.0134006W, Scaling Factor: 1.04826.
Solve	00:00:08	00:00:06	173 M	
Temperature Mapping	00:00:01	00:00:00	0 K	
Solution Process				Elapsed time : 00:01:24 , MechanicalComengine.exe ComEngine Memory : 113 M
Total	00:01:17	00:00:29		Time: 04/26/2019 09:53:19, Status: Normal Completion

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can judge whether you've run a sufficient number of coupling iterations by comparing the *Total EM Loss* magnitudes between the last two iterations (highlighted in the preceding *Profile* image). Convergence occurs when the EM losses and resultant temperatures stabilize (that is, when the variation in these results between adjacent iterations becomes insignificant). However, in certain cases a solution may not stabilize, and you may experience an oscillatory or divergent (runaway) solution. If the total EM losses oscillate (that is, continue to repeat an increase/decrease cycle), the solution still may be acceptable if the magnitude of the change is relatively small.

11. Depending on the comparative *Total Loss on Source Object* results, choose one of the following three actions:
    - If the changes in the total loss results are insignificant between the last two completed iterations, consider the coupled solution converged. The two-way coupled analysis is finished. Continue to step 12.
    - If the total loss results significantly differ between the last two completed iterations, increase the specified *Number of Coupling Iterations* and resume the two-way coupled analysis as follows:
        a. In the downstream design, under **Analysis** > ***mechanical solution setup*** in the Project manager, double-click **2-Way Coupling**.
        b. Increase the value in the **Number of Coupling Iterations** text box, and click **OK**.
        c. Right-click the ***mechanical solution setup*** in the Project Manager and choose **Analyze**.

        > **Note:**
        >
        > - When you restart the solution, it proceeds immediately to the next iteration using the field results from the last one completed.
        > - The *Profile* retains the history of all previously run and new iterations.

        d. Go to step 9 and continue again from there.
    - If the solution is diverging or the results are oscillating significantly, recheck boundaries, excitations, material properties, and mesh quality. Divergent or oscillatory solutions are more likely when one or more materials in the source design have a highly nonlinear conductivity or loss tangent. Tightening the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

convergence tolerance and increasing the maximum number of passes in the source solution setup may be helpful.

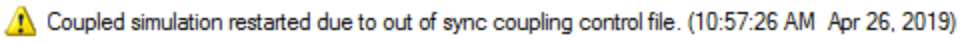
After revising the source design, return to the target design, clear the linked data, and rerun the coupled analysis.

12. Complete the post processing phase (create desired reports and field overlays).

Solution Invalidation

Previously solved results are invalidated when you delete the *2-Way Coupling* setup or when you right-click the solution setup in the Project Manager and choose **Revert Coupled Solution** from the shortcut menu.

The controller automatically detects “out of sequence" simulations and restarts the coupled analysis. The following message is posted in the *Message Manager* window when a coupled simulation needs to be restarted:



Examples of out of sequence simulation situations include:

- Source design edits have been made that invalidate the source design solution.
- Target design edits have been made that invalidate the controller data, including edits that invalidate the imported *EM Loss* data. Examples include changes in mesh and geometry.

Results (Thermal)

Thermal analysis results presentations fall into two basic categories: *Fields Reports* and *Field Overlays*.

- **Fields Report:**

  There are several different display types available for fields reports, including 2D (Rectangular) Plots, 3D Plots, Data Tables, and more. One convenient way to generate a report of thermal field results is to draw a **Polyline** running along the faces and/or through the interior volume of your model. This polyline, which can consist of multiple straight and curved segments, defines a path along which you wish to plot a particular thermal result.

  For **Steady-State Thermal** solutions: A polyline is required for reporting all *predefined* thermal field results with the exception of *Design Volume*. The sweep is the *Distance* along the specified polyline and is therefore in the spatial domain.

  You can also plot user-defined *Named Expressions* along a polyline or create named expressions that do not need a polyline for plotting.

  For **Transient Thermal** solutions: You can plot thermal results using two different types of geometry:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Polyline:** Plot the desired thermal result versus the distance along a specified polyline (spatial domain) in the same manner as previously described for steady-state solutions.
- **Point:** Plot the desired thermal result **versus time** at the specified point location. In this case, the sweep is in the time domain rather than the spatial domain.

Regardless of the display type, the following predefined thermal results are available for reporting:

- **Temperature:** For steady-state or transient solutions, plot temperature versus the distance along a specified polyline
- **Mag_HeatFlux:** Heat flux magnitude versus the distance along a specified polyline.
- **Surface_Loss_Density:** The surface-based electromagnetic (EM) loss density (in units of power per unit area) imported from another design as a source of heat in the thermal solution. It can be plotted versus the distance along a specified polyline.
- **Volume_Loss_Density:**The volumetric electromagnetic (EM) loss density (in units of power per unit volume) imported from another design as a source of heat in the thermal solution. It can be plotted versus the distance along a specified polyline.
- **Design Volume:** The volume of the objects comprising the model.
- **Linked_Heat_Transfer_Coefficient:** For steady-state or transient solutions in which you have chosen the *Non-Uniform* option for a *Convection* boundary's *Film Coefficient*. Heat transfer coefficients imported from an Icepak solution are plotted versus distance along a specified polyline.
- **Thermal_Conductivity_X**, **Thermal_Conductivity_Y**, and **Thermal_Conductivity_Z:** These results are applicable only to models with layout components. The three variants report the mapped thermal conductivity in the X, Y, and Z directions, respectively.
- ***Named Expression*:** A custom result previously defined using the Fields Calculator (see the Fields Calculator Note on the *Creating a Report* page).

- **Thermal Monitor**: Plot the global maximum temperature vs. time and the temperature vs. time at user-defined monitor points assigned before running the analysis.

- **Monitor Report**: Similar to a fields report, but specifically for plotting or tabulating temperature versus time (for *Transient Thermal* analyses only)

- **Field Overlay**:

  You can overlay fields on selected solid or sheet objects, faces, or polylines. The following six results are available:

  - **Temperature:** Color contour overlay of resultant temperatures.
  - **Mag_HeatFlux:** Color contour overlay of heat flux magnitudes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **HeatFlux:** Color *vector* plot indicating the magnitude and direction of heat flux.
- **Thermal Conductivity X**, **Thermal Conductivity Y**, and **Thermal Conductivity Z:** These results are applicable only to models with layout components. The three variants overlay the mapped thermal conductivity in the X, Y, and Z directions, respectively.
- **Surface Loss Density:** Color contour overlay of surface-based electromagnetic (EM) losses imported from a coupled source design.
- **Volume Loss Density:** Color contour overlay of volumetric electromagnetic (EM) losses imported from a coupled source design.
- **Linked Heat Transfer Coefficient:** Color contour overlay of the heat transfer coefficients imported from an Icepak solution when you have chosen the *Non-Uniform* option for a *Convection* boundary's *Film Coefficient*.
- ***Named Expression*:** A custom result previously defined using the Fields Calculator (see the Fields Calculator Note on the *Creating a Report* page).

- **Fields Summary:**

  Provide a Summary based on resultant fields data at boundaries, objects (volumes or surfaces), faces (from face lists), or along global or relative coordinate system planes. Summary information includes details such as the minimum, maximum, and mean field values, the standard deviation of the results, and the normal direction vector (for surfaces).

These reports, overlays, and fields summary are described more fully, along with the instructions for creating them, in the subsections that follow.

Fields Report

The following display types are available for fields reports in Mechanical – Thermal solutions:

- 2D (Rectangular) Plot
- Stacked Plot
- Data Table
- 3D Plot
- 2D Contour Plot

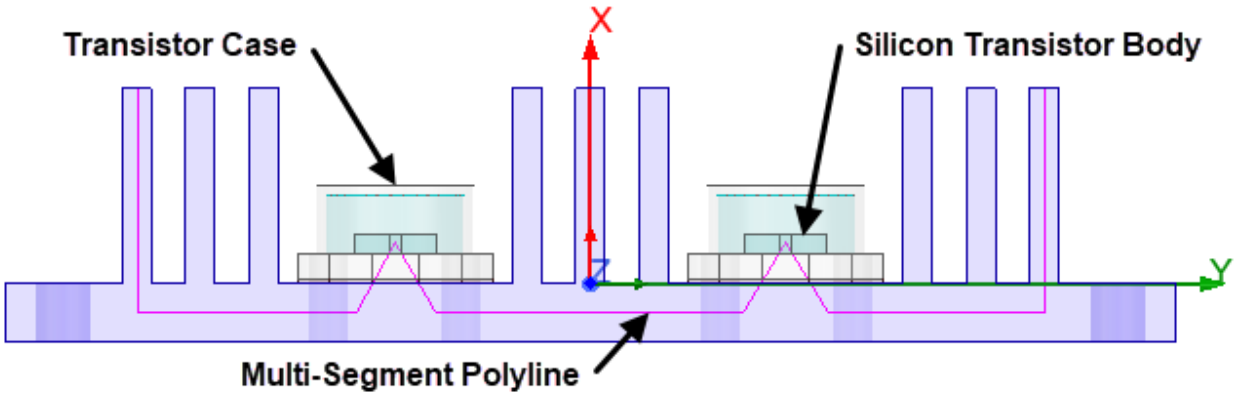
In this topic, we will focus on the display types that are most likely to be used for thermal reports (specifically 2D plots, Stacked plots, and Data Tables). The procedure for creating other types of plots is similar. The subtopic, Creating a Fields Report, details how to generate plots and data tables of thermal fields results.

In order to create a fields report of the *predefined* thermal results, an arc, line, or series of straight and/or curved *Polyline* segments is needed. You can draw the line segments manually or select edges of your model and convert them to line objects. Then, specify this polyline geometry as the path along which the predefined thermal result values will be calculated and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plotted (results versus distance along the path). Alternatively, for *Transient Thermal* solutions, you can also draw a *Point* and use it for plotting thermal results at the point location versus *Time*.

For example, the following image is a pair of complimentary transistors mounted with insulators on a common aluminum heat sink. A multi-segment polyline has been drawn that will be used to show the temperature variation between the+X ends of the outermost fins, with the line segments passing through the base of the heat sink, insulators, transistor cases, and the silicon transistor bodies within the cases:



Note:

Whether you draw the polyline before or after running the solution, it's a good idea to create it as a **non-model** part. However, the solver will ignore line and sheet objects even if the **Model** option is selected in the object properties. If you create a polyline after solving, you will be prompted to create a non-model object, or the solution results will be invalidated.

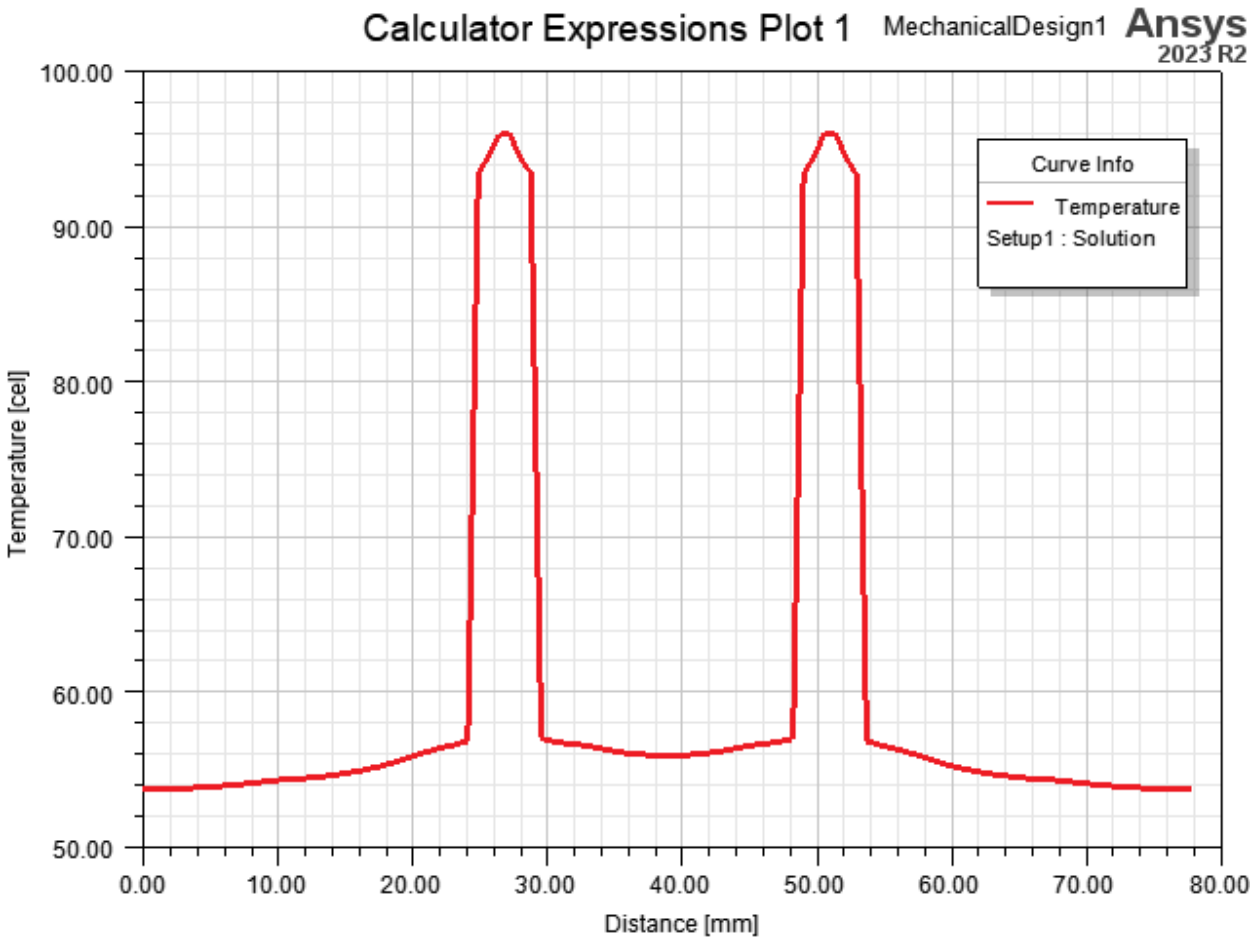
For *Transient Thermal* solutions: When using a point to plot a result in the time domain, you can add the point at any time with no concern of invalidating the results. Points are exclusively non-model entities.

The available *predefined* thermal results are listed and described under Fields Report in the *Results* topic. Examples of 2D plots, Stacked plots, and Data Tables follow:

- **2D (Rectangular) Plot:** This type of plot can contain one trace or multiple traces, each of which is plotted within the same Cartesian grid.

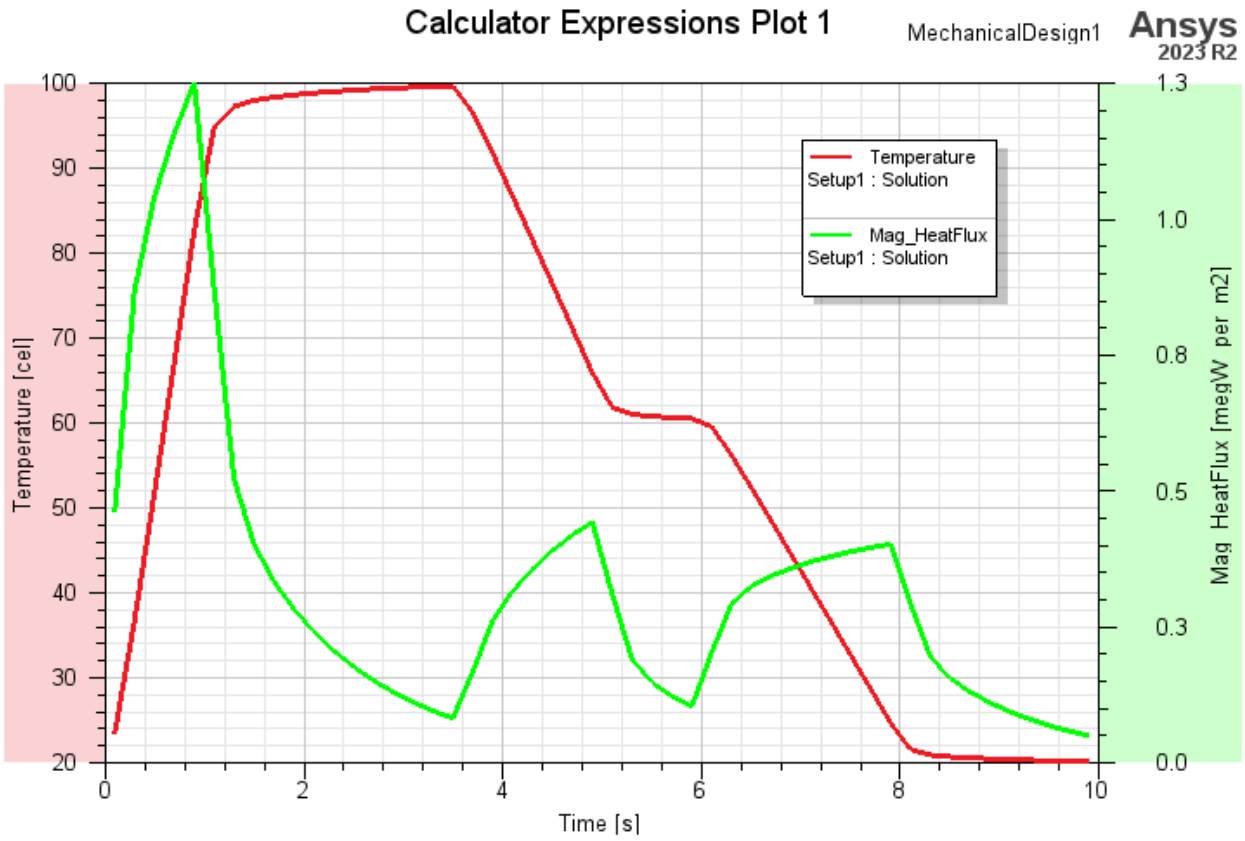
  The following example shows the *Temperature vs. Distance* 2D (rectangular) plot based on the preceding model and polyline example. The steepest temperature gradients occur as the line passes through the insulators, which is expected due to the relatively poor thermal conductivity compared to aluminum and silicon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following example shows temperature and heat flux magnitude versus time for a point inside the solid geometry of a transient thermal analysis model:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

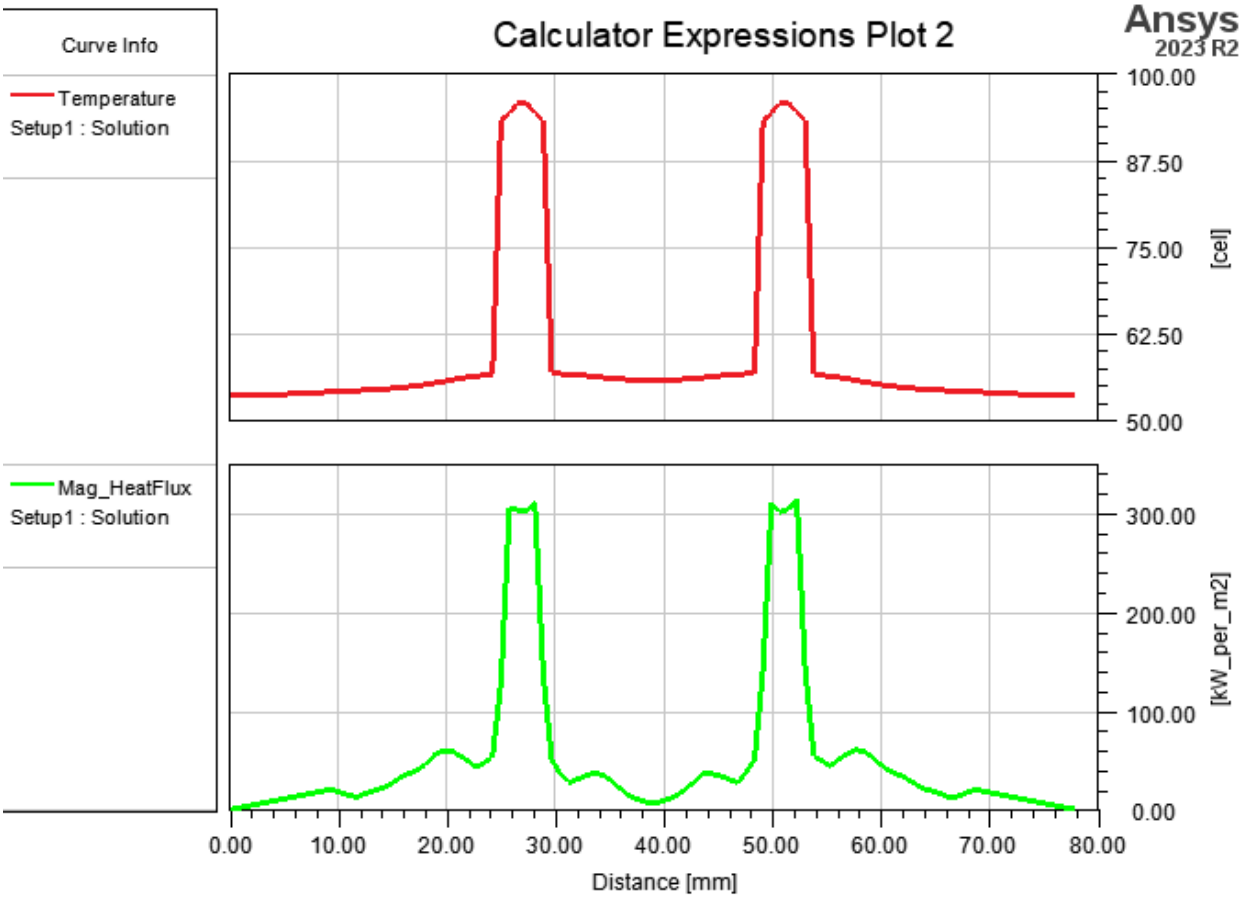


- **Stacked Rectangular Plot:** This type of plot is intended for multiple traces. Unlike the 2D plot previously described, this type puts every individual trace in a separate Cartesian grid stacked vertically. The X scale is shared by all traces, but the Y scale is adjusted to suit each individual trace.

  As with the 2D plot, you must create a polyline object along which the *predefined* thermal result values will be calculated and plotted.

  The following image is an example of a two-curve stacked rectangular plot from the same model. The top trace is the *Temperature* result along the polyline, and the bottom trace is the *Heat Flux Magnitude* plotted along the same polyline. For both traces, the X axis is the distance along the polyline:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Data Table:** Tabulation of any of the available fields report results as an alternative to a plotted curve.

  The example that follows lists the heat flux magnitude and temperature versus the distance along a polyline running through an electric induction motor. The polyline extends radially outward from the centroid of the rotor to the outside diameter of the stator, passing through a winding slot in the stator on its way:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Expressions Table 1 MechanicalDesign1 Ansys 2023 R2

	Distance [mm]	Mag_HeatFlux [kW_per_m2] Setup1 : Solution	Temperature [cel] Setup1 : Solution
1	0.000000	0.019202	98.503994
2	2.286000	0.035474	98.505136
3	4.572000	0.063153	98.508544
4	6.858000	0.092246	98.512481
5	9.144000	0.122258	98.520463
6	11.430000	0.150987	98.531189
7	13.716000	0.136450	98.539332
8	16.002000	0.066917	98.544246
9	18.288000	0.015711	98.547584
10	20.574000	0.049754	98.545639
11	22.860000	0.096425	98.541175
12	25.146000	0.142022	98.535069
13	27.432000	0.181580	98.525015
14	29.718000	0.221502	98.514021
15	32.004000	0.258385	98.500004
16	34.290000	0.294615	98.484300
17	36.576000	0.329524	98.466524
18	38.862000	0.363814	98.446877
19	41.148000	0.397333	98.425377
20	43.434000	0.430202	98.401550

In this case, you would click and drag the scroll bar on the right vertically to see all 101 data points that were tabulated.

The following example shows temperature and heat flux vs. time for a point inside the solid geometry of a transient thermal analysis model:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Expressions Table 1 MechanicalDesign1 Ansys 2023 R2

	Time [s]	Temperature [cel] Setup1 : Solution	Mag_HeatFlux [megW_per_m2] Setup1 : Solution
30	6.0	60.50	0.096
31	6.2	57.99	0.260
32	6.4	54.31	0.311
33	6.6	50.44	0.335
34	6.8	46.52	0.352
35	7.0	42.59	0.365
36	7.2	38.64	0.375
37	7.4	34.68	0.385
38	7.6	30.72	0.392
39	7.8	26.75	0.399
40	8.0	22.78	0.405
41	8.2	21.26	0.232
42	8.4	20.90	0.174
43	8.6	20.74	0.143
44	8.8	20.62	0.121
45	9.0	20.53	0.103
46	9.2	20.45	0.088
47	9.4	20.39	0.075
48	9.6	20.33	0.065
49	9.8	20.28	0.055
50	10.0	20.24	0.047

Creating a Fields Report

The following is the general procedure for creating any type of Fields Report. It includes drawing a polyline to use as the path along which thermal results will be plotted or tabulated. This step is necessary for the predefined temperature, heat flux, and loss results. However, you can plot or tabulate *Design Volumes* and *Named Expressions* you create using the *Fields Calculator* (see note below) without creating any polyline geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The *Fields Calculator* is a powerful and flexible tool for post processing your analysis results. Use it to perform simple or complex mathematical operations on the raw results data. With an appropriate named expression defined, you can generate a report that does not require a polyline as its basis. For example, you could define a face list and create a named expression that integrates the heat flow rate for all faces in the list. Then, in a parametric analysis, you could base the plot's X axis on a design parameter. Finally, base the plot's Y axis on your named expression, thus creating a plot of total heat flow rate of the faces versus the swept parameter. There are many other possibilities as well.

You can access the *Fields Calculator* by right-clicking **Field Overlays** in the Project Manager and selecting **Calculator** from the shortcut menu. For more information, search the Ansys Electronics Desktop help for "using the fields calculator" (include the quotation marks to limit search results to instances of the full phrase). Also, refer to the *Related Topics* links at the bottom of the *Using the Fields Calculator* page.

1. Optionally, if you will be plotting thermal results in the **spatial domain**, create a polyline path to use as the basis of the table's or plot's X data using *one* of the following two methods:
    - **Draw a polyline manually:**
        a. On the **Draw** ribbon tab, click **Draw line**.
        b. If prompted to create a non-model object (because a solution already exists), click **Yes**.
        c. Draw a line or series of lines along which you wish to determine the thermal results. The line segments should pass through or along the surfaces of the bodies comprising the model. Use one of the following three drawing methods:
            - Specify the line start and end points graphically (by clicking in the Modeler window). You can snap to model vertices (endpoints), midpoints or edges, grid points, and more.
            - Specify the points by typing the coordinates into the **X, Y,** and **Z** text boxes in the status bar.
            - Draw an arbitrary line and edit it's properties afterward (in the docked *Properties* window or the *Properties* dialog box).

              You can switch between straight and curved segments by right-clicking in the Modeler window and choosing the desired **Set Edge Type** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. After clicking the last endpoint, right-click in the Modeler window and choose **Done** to terminate the *Draw Line* command.

   Notice the name of the line object that has been added to the History Tree. The default name for the first line object is *Polyline1*, and you can change the name in the docked *Properties* window if desired.

e. Continue to step **3**.

- **Convert model edges to a polyline:**
  a. In *Edge* selection mode, select a model edge or a contiguous series of model edges to select them.
  b. On the **Draw** ribbon tab, click **Edge** > **Create Object From Edge**.
  c. While the objects are still selected, click **Unite** on the **Draw** ribbon tab.

     Notice the name of the united object that has been added to the History Tree. You can change the name in the docked *Properties* window if desired.

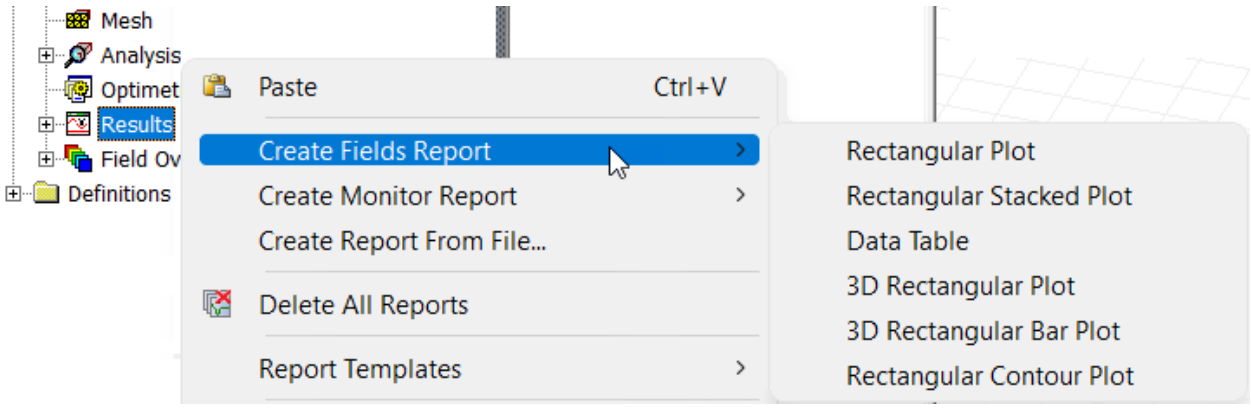
  d. Continue to step **3**.

2. Optionally, if you will be plotting thermal results in the **time domain** (transient solutions only), create a point to use as the basis of the table's or plot's Y data, as follows:
   a. On the **Draw** ribbon tab, click **Draw Point**.
   b. Click a suitable grid point or snap point on the model (such as the midpoint of an edge, quadrant point on a circular edge, center of a face, or vertex) to place a point there.

      Alternatively, tab into the coordinate entry text boxes at the bottom of the program window, type the exact coordinates of the desired point, and press **Enter**.

3. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Fields Report** and choose the ***desired display type*** (such as, **2D**, **Data Table**, or **Stacked**).
   - Using the menu bar, click **Mechanical** > **Results** > **Create Fields Report** > ***desired display type***.
   - Right-click **Results** in the Project Manager and choose **Create Fields Report** > ***desired display type***:

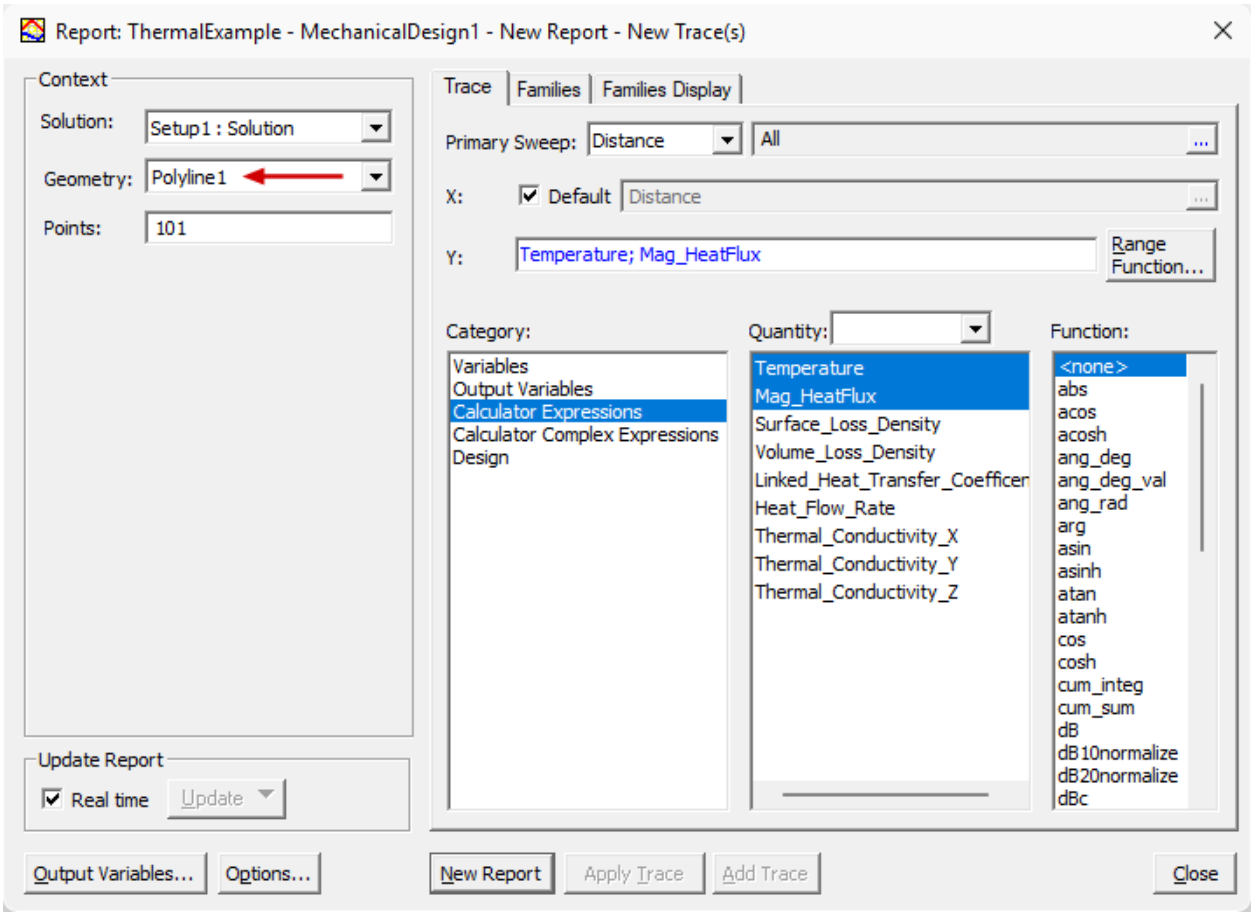
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



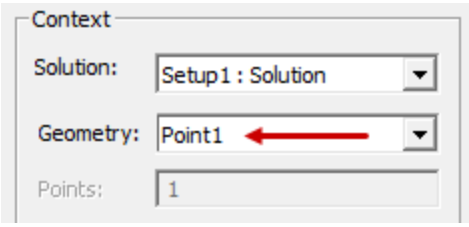
4. Specify the following settings in the *Report* dialog box:
   a. Choose the desired solution setup from the **Solution** drop-down menu (if you solved more than one setup).
   b. Choose the polyline or point you created previously from the **Geometry** drop-down menu.
   c. Optionally, for a polyline, specify a different number of **Points** to calculate along the specified polyline. (The default number of points is 101, which results in one hundred trace segments between the first and last data points.)
   d. In the **Category** list, select **Calculator Expressions**, if it is not already selected.
   e. Select the desired **Quantity** or **Quantities** to tabulate or plot. (You do not need to specify a *Function* for thermal results.)

   The following report setup example shows the settings to tabulate or plot *Temperature* and *Mag_HeatFlux* results along *Polyline1*:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



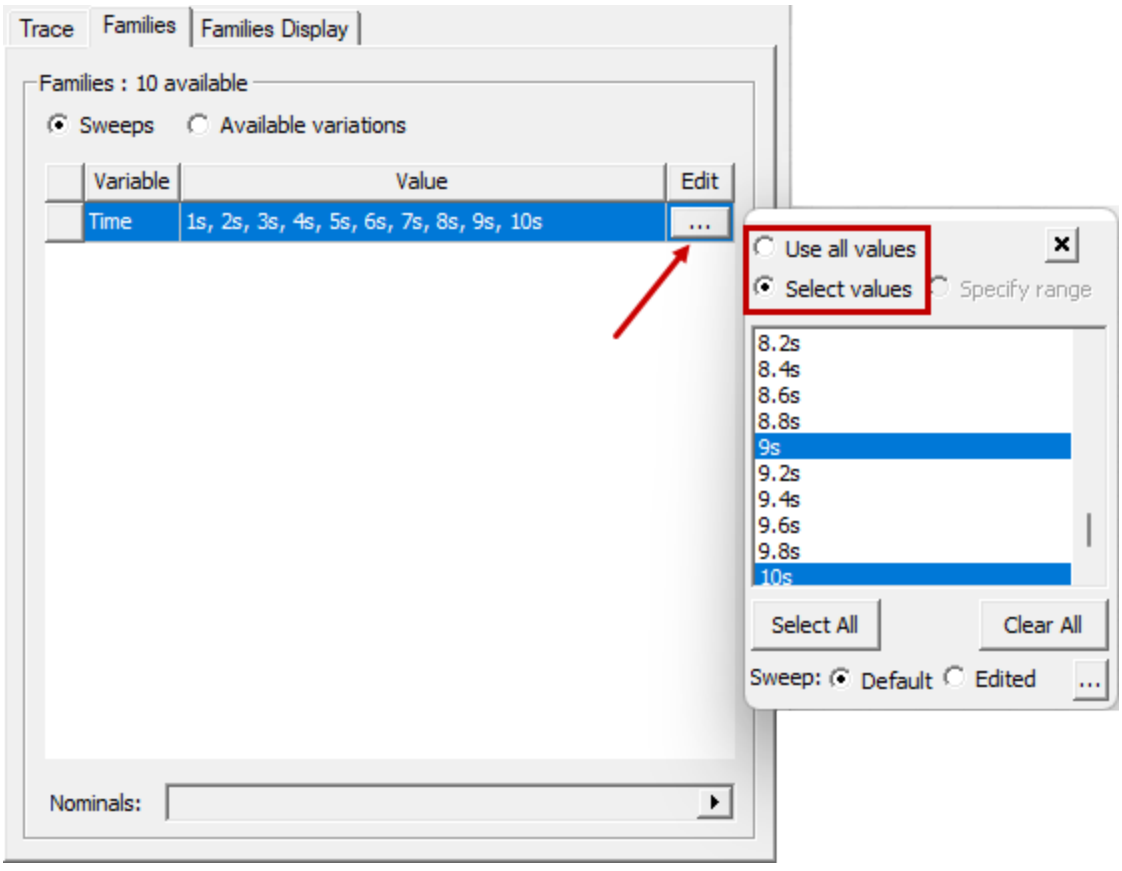
To tabulate or plot *Temperature* and *Mag_HeatFlux* at a point versus *Time* (transient solutions only), the setup would be the same as the preceding example, except for the **Geometry** selection:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can also choose polyline geometry for plotting the results of a transient solution. However, the plot with be in the spatial domain, with the distance along the polyline as the X axis. A separate curve will be added to the plot for every time value selected under the **Families** tab.

5. Optionally, click **Range Function** to apply a mathematical operation to the results. Then:
   a. In the *Set Range Function* dialog box, choose the desired operation from the **Function** drop-down menu (such as *sum, mean, max* or *variance*).
   b. Click **OK** to return to the *Report* dialog box.
6. Optionally, if there are multiple design variants (such as in a Parametric analysis) or multiple time steps (in a transient solution), click the **Families** tab to choose the ones to plot. Then:
   a. Click the **ellipsis button** in the **Edit** column:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Choose **Use all values** or choose **Select values** and pick individual values to include in the plot.

7. Click **New Report**.

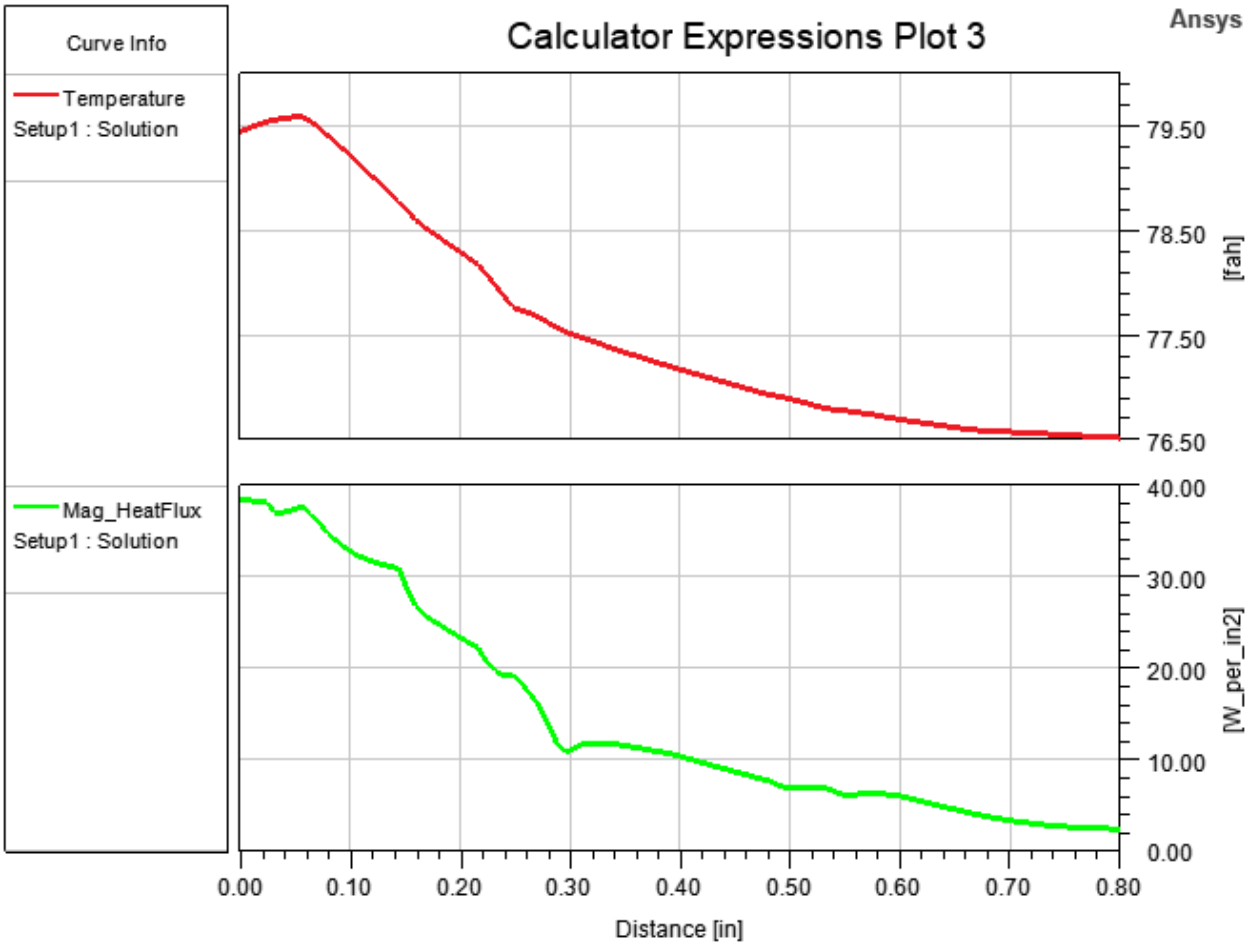
   The plot or data table appears in a new window.

8. Optionally, if you created a plot, you can double-click inside the legend box to access the *Properties* dialog box and customize many plot parameters, such as:
   - Trace colors, style, and width
   - Enable data point symbols and change their color, fill, style, and frequency
   - Grid options
   - Headers
   - X and Y axis parameters and scaling
9. Optionally, if you created a data table, you can customize it in the following ways:
   - Drag the column borders to resize the columns.
   - Click on a column heading and then, in the docked *Properties* window, adjust the **Units, Field Width,** or **Field Precision** values.
10. Optionally, select parameters for one or more additional results to add to the current plot or table and click **Add Trace**.
11. Click **Close** to exit the *Report* dialog box.

Examples:

The following example is a stacked rectangular plot of the *Temperature* and *Mag_HeatFlux* results along *Polyline1*. The polyline was drawn from the center of a transistor/heat sink contact face to one of the bottom corners of the heat sink. The temperature and heat flux decrease with increased distance from the heat source:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following example is a data table listing the volume of the three objects comprising a model. It has been modified to show the volumes in cubic inches, and the columns have been reduced from their default widths:

Design Table 1 MechanicalDesign1 Ansys

	X [in]	Volume(Heat_Sink) [inch3] Setup1 : Solution	Volume(Transistor) [inch3] Setup1 : Solution	Volume(Insulator) [inch3] Setup1 : Solution
1	0.000000	0.255000	0.010603	0.001060

The contents of the **Y** text box in the *Report* dialog box used to produce the preceding table were as follows:

Y: Volume(Heat_Sink); Volume(Transistor); Volume(Insulator)

In this example, an additional column has been added to *Design Table 1*:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Table 1 MechanicalDesign1 Ansys

	X [mm]	Volume(Heat_Sink) [inch3] Setup1 : Solution	Volume(Transistor) [inch3] Setup1 : Solution	Volume(Insulator) [inch3] Setup1 : Solution	Volume(Heat_Sink) + Volume(Transistor) + Volume(Insulator) [inch3] Setup1 : Solution
1	0.000000	0.255000	0.010603	0.001060	0.266663

With the **Volume** of all three parts selected in the **Quantity** list of the *Report* dialog box, the semicolons in the **Y** text box were replaced with plus signs (+):

to produce the total volume column (far right).

Monitor Report

In addition to using the *Thermal Monitor* window to display transient temperature results at monitor points, you can generate a monitor report of global or user-defined monitor point temperatures versus time. This type of report is only available in Mechanical designs of the *Transient Thermal* solution type. The following display types are available for monitor reports:

- 2D (Rectangular)
- Stacked
- Data Table
- 3D
- 3D Rectangular Bar
- 2D Contour

In this topic, we will focus on the display types that are most likely to be used for thermal monitor reports, specifically, Data Tables, 2D (Rectangular) plots, and Stacked plots. The procedure for creating other types of plots is similar. Since this report type is specific to transient solutions, the X axis will be *Time* in all cases.

The available thermal results are listed and described in the Results introduction. For thermal monitors, the global maximum temperature monitor is listed as a *Quantity* when the **Global** *Category* is selected in the *Report* dialog box. User-defined thermal monitor points are listed when the **Monitor** *Category* is selected. Examples of Data Tables, 2D plots, and Stacked plots follow:

- **Data Table:** The same information is available as for curve plots, but the information is presented as tabulated numerical results.

  The following image is an example of a data table listing temperature versus time for the global maximum and two user-defined monitor points. The *Number Format* for the Mode column was changed to integer, the *Field Precision* of the ModeFreq column was reduced to 2 (decimal places), and the column widths were customized:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor Table 1

MechanicalDesign2 Ansys 2024 R2

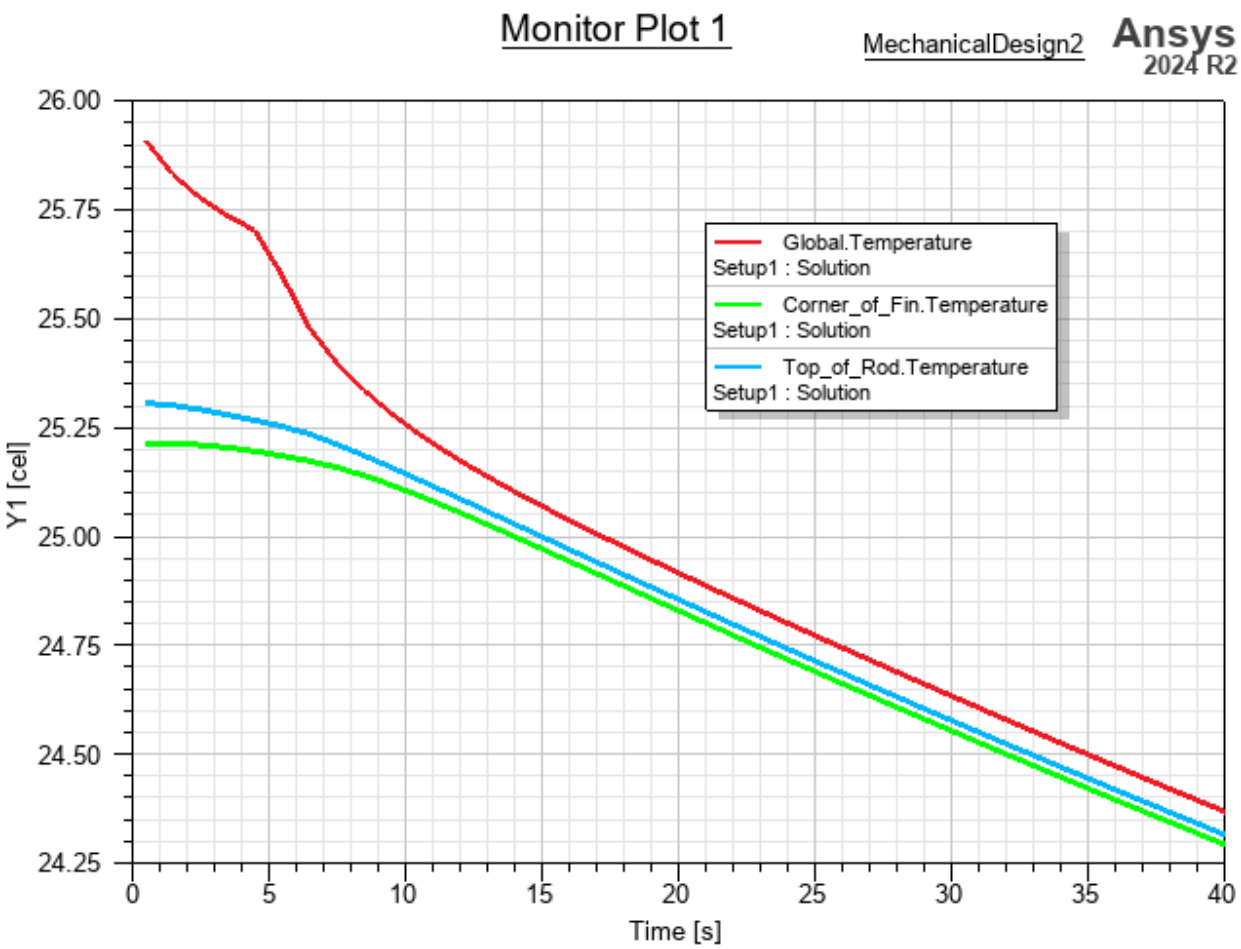
	Time [s]	Global.Temperature [cel] Setup1 : Solution	Corner_of_Fin.Temperature [cel] Setup1 : Solution	Top_of_Rod.Temperature [cel] Setup1 : Solution
1	0.000000	nan	nan	nan
2	0.500000	25.908360	25.212770	25.305290
3	1.500000	25.829080	25.212390	25.300560
4	2.500000	25.775730	25.209700	25.290720
5	3.500000	25.734450	25.203790	25.278760
6	4.500000	25.700960	25.195240	25.265740
7	5.500000	25.598290	25.184780	25.252050
8	6.500000	25.477250	25.172790	25.234770
9	7.500000	25.398240	25.158070	25.211440
10	8.500000	25.334980	25.139560	25.185340
11	9.500000	25.281650	25.117810	25.157930
12	10.500000	25.235210	25.093730	25.129770
13	11.500000	25.193620	25.068040	25.101150
14	12.500000	25.155510	25.041250	25.072260
15	13.500000	25.119880	25.013750	25.043230
16	14.500000	25.086060	24.985800	25.014160
17	15.500000	25.053570	24.957580	24.985110
18	16.500000	25.022050	24.929220	24.956120
19	17.500000	24.991270	24.900810	24.927230
20	18.500000	24.961060	24.872420	24.898450

Tables are also well suited for presenting results on which you have performed mathematical operations (such as reporting the maximum, minimum, or average values).

- **2D (Rectangular) Plots:** This display type produces continuous traces by default. You can plot one quantity or multiple quantities using this format, with all of the results sharing the same Cartesian grid and axes.

  The following image is based on the same solution results as the preceding data table example:

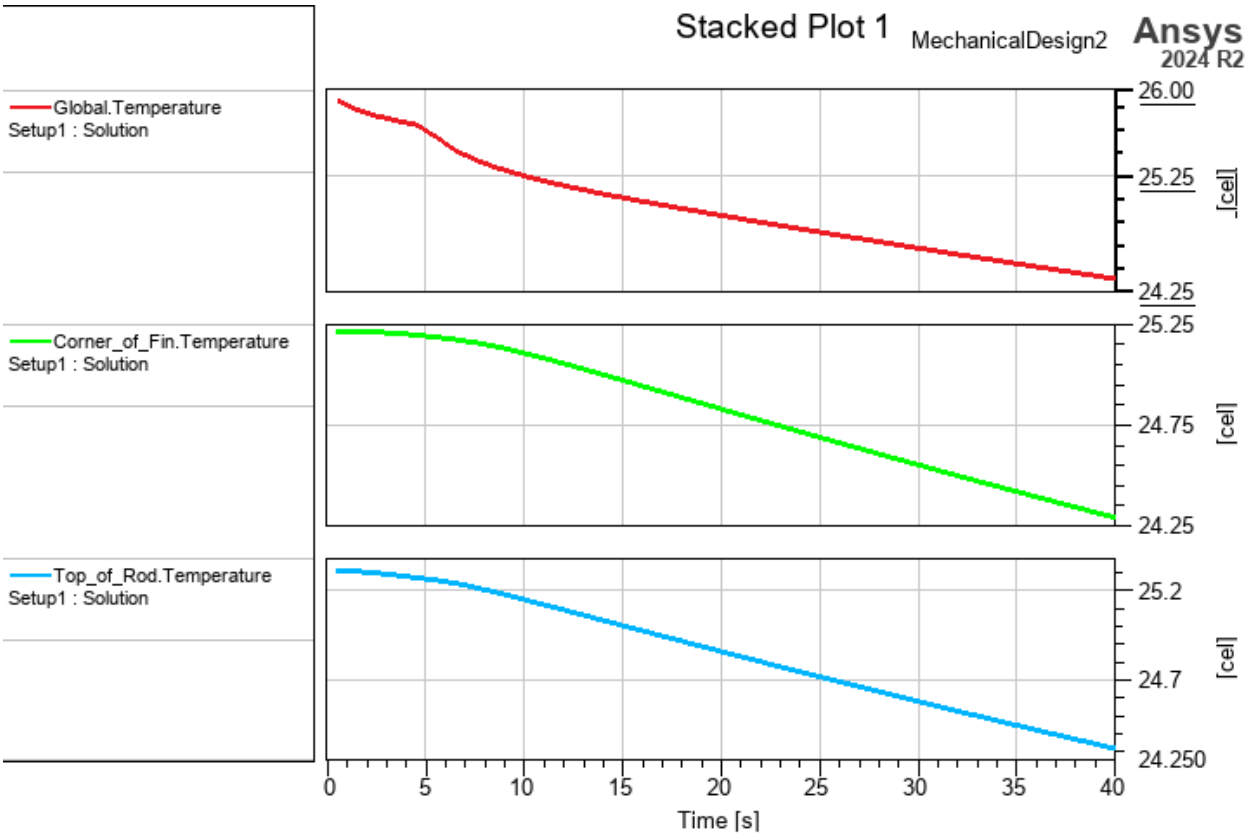
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Stacked Rectangular Plots:** This display type is intended for multiple traces. Unlike the 2D plot previously described, this type puts every individual trace in a separate Cartesian grid stacked vertically. The X scale is shared by all traces, but the Y scale is adjusted to suit each individual trace.

  The following image is based on the same solution results as the preceding data table and 2D plot examples:

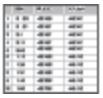
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See the subtopic, Creating a Monitor Report, for step by step instructions on creating and customizing thermal monitor reports.

Creating a Monitor Report

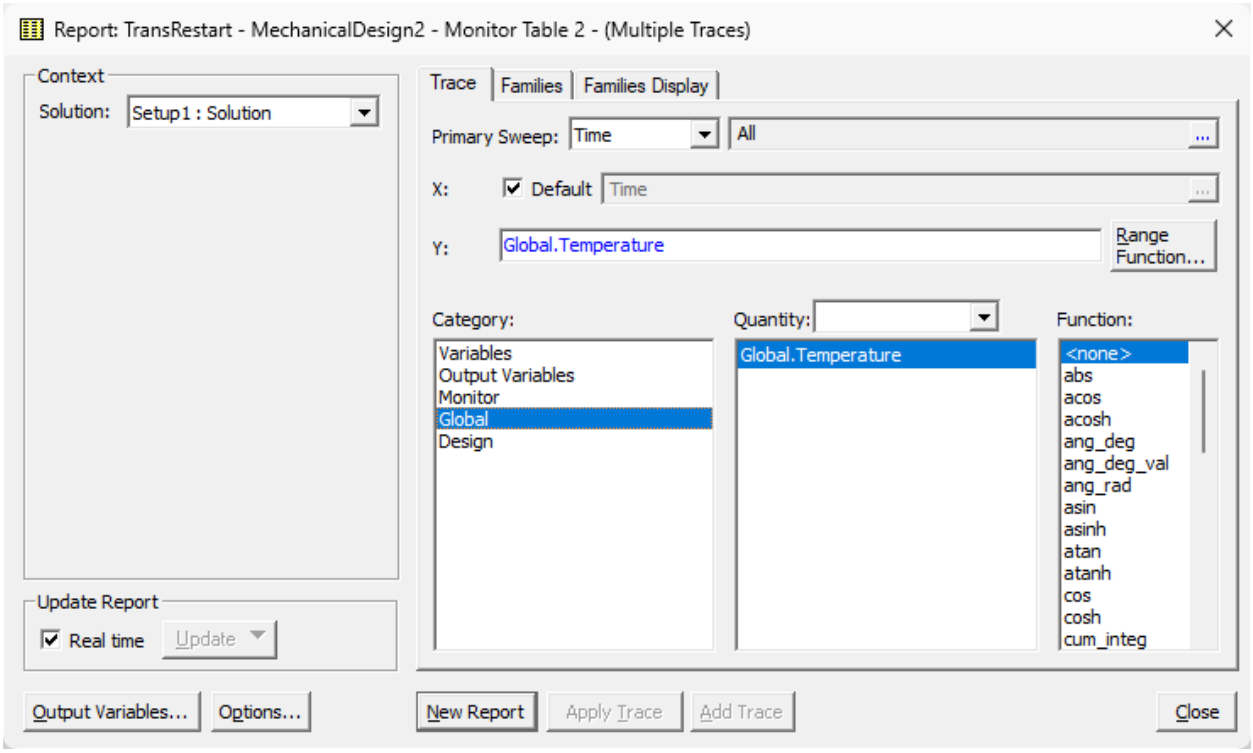
Data Table Example:

1. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Monitor Report** >  **Data Table**.
   - Using the menu bar, click **Mechanical** > **Results** > **Create Monitor Report** > **Data Table**.
   - Right-click **Results** in the Project Manager and choose **Create Monitor Report** > **Data Table** from the shortcut menu.
2. Select the desired **Solution, Category,** and **Quantity** or **Quantities** to tabulate.

   The global maximum temperature (*Global.Temperature*) is listed as an available *Quantity* when you select the **Global** *Category*. User-defined monitor points are shown in the *Quantity* list when you select the **Monitor** *Category*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following report setup example shows the settings to tabulate two user-defined monitor points:



3. Click **New Report** but keep the *Report* dialog box open.

   The table appears in a new window. The following data table example is consistent with the *Report* dialog box settings shown under step 2:

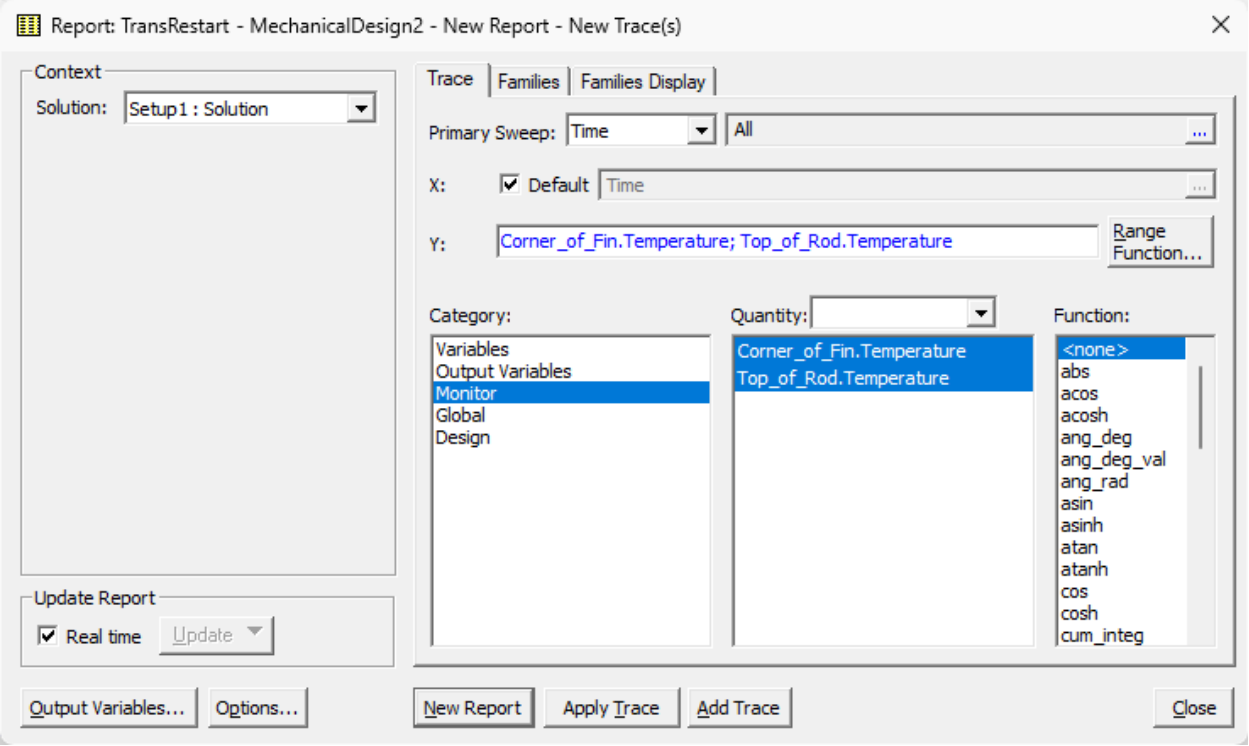
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor Table 1 MechanicalDesign2 Ansys 2024 R2

	Time [s]	Global.Temperature [cel] Setup1 : Solution
1	0.000000	nan
2	0.500000	25.908360
3	1.500000	25.829080
4	2.500000	25.775730
5	3.500000	25.734450
6	4.500000	25.700960
7	5.500000	25.598290
8	6.500000	25.477250
9	7.500000	25.398240
10	8.500000	25.334980
11	9.500000	25.281650
12	10.500000	25.235210
13	11.500000	25.193620
14	12.500000	25.155510
15	13.500000	25.119880
16	14.500000	25.086060
17	15.500000	25.053570
18	16.500000	25.022050
19	17.500000	24.991270
20	18.500000	24.961060

4. Change the Category from Global to Monitor and then select the desired user-defined monitor points, as shown in the following image:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Add Trace** and then **Close**.

   A column is added to the table for each selected quantity.

6. Optionally, customize the table by dragging column widths or selecting a column and changing parameters in the docked *Properties* window (for example, to change the *Field Precision* of the tabulated numbers).

   The following table results from the previously shown settings. The column widths, field widths, and field precision have been customized for all columns:

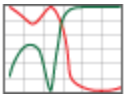
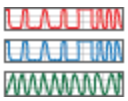
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor Table 1 MechanicalDesign2 Ansys 2024 R2

	Time [s]	Global.Temperature [cel] Setup1 : Solution	Corner_of_Fin.Temperature [cel] Imported	Top_of_Rod.Temperature [cel] Imported
1	0.0	nan	nan	nan
2	0.5	25.91	25.21	25.31
3	1.5	25.83	25.21	25.30
4	2.5	25.78	25.21	25.29
5	3.5	25.73	25.20	25.28
6	4.5	25.70	25.20	25.27
7	5.5	25.60	25.18	25.25
8	6.5	25.48	25.17	25.23
9	7.5	25.40	25.16	25.21
10	8.5	25.33	25.14	25.19
11	9.5	25.28	25.12	25.16
12	10.5	25.24	25.09	25.13
13	11.5	25.19	25.07	25.10
14	12.5	25.16	25.04	25.07
15	13.5	25.12	25.01	25.04
16	14.5	25.09	24.99	25.01
17	15.5	25.05	24.96	24.99
18	16.5	25.02	24.93	24.96
19	17.5	24.99	24.90	24.93
20	18.5	24.96	24.87	24.90

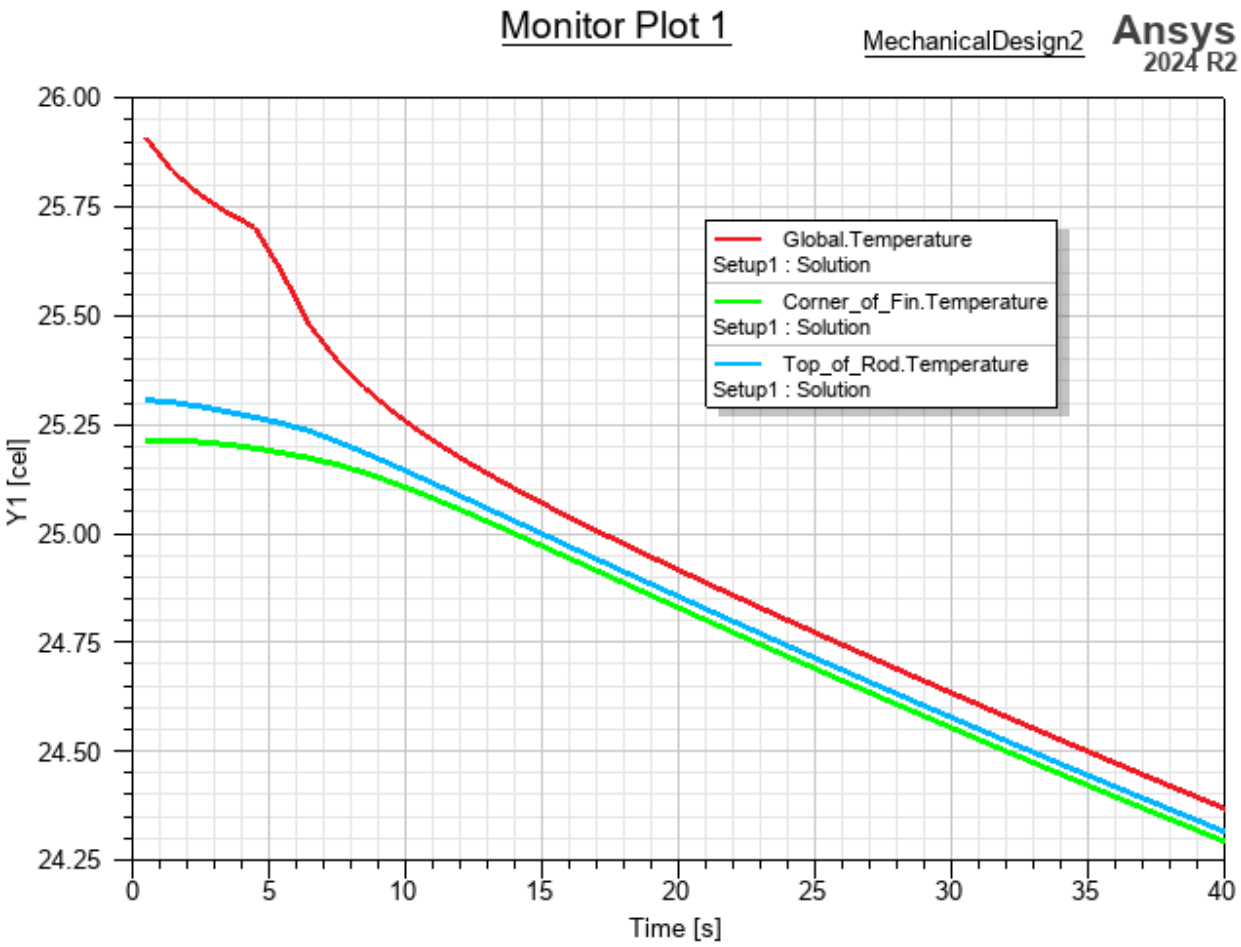
2D (Rectangular) or Stacked Plot Example:

This procedure is exactly the same as for creating a data table of thermal monitor points, except that a different report type is selected in the first step.

1. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Monitor Report** >  **2D** or  **Stacked**.
   - Using the menu bar, click **Mechanical** > **Results** > **Create Monitor Report** > **Rectangular Plot** or **Rectangular Stacked Plot**.
   - Right-click **Results** in the Project Manager and choose **Create Monitor Report** > **Rectangular Plot** or **Rectangular Stacked Plot** from the shortcut menu.
2. Follow steps 2 through 5 of the preceding data table creation procedure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the same monitor results as the data table example, but this time in the form a a 2D plot:



Field Overlay and Animation

In thermal analyses, field overlays provide a visual indication of temperature and heat flux results, each of which can be overlaid on selected objects or faces:

The available *predefined* thermal field overlay results are listed and described under Field Overlay in the *Results (Thermal)* topic. Examples of 2D plots, Stacked plots, and Data Tables follow:

To Produce a Field Overlay Plot and Animate it:

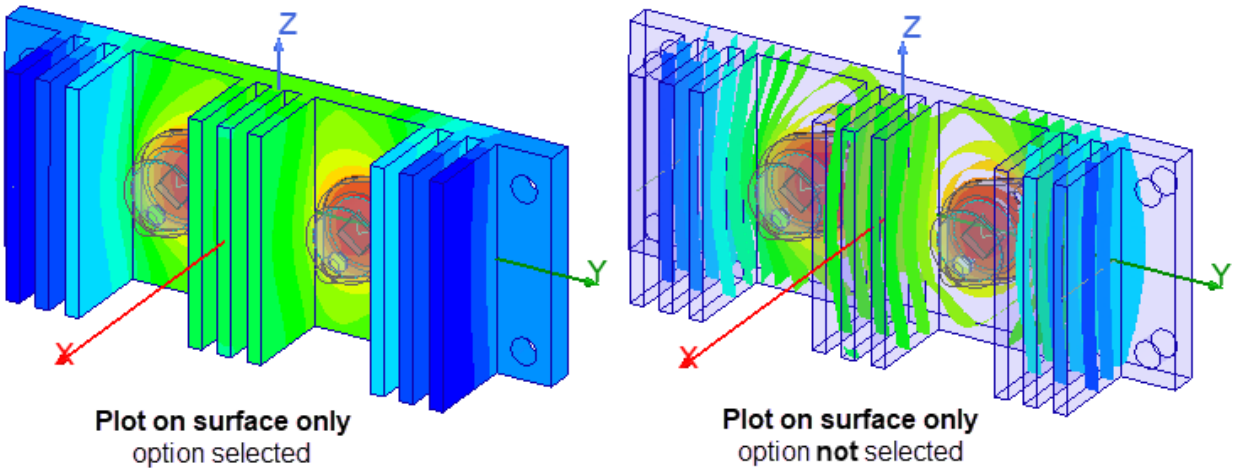
1. Select the *Faces* or *Objects* on which you want to overlay thermal results.
2. Access the *Create Field Plot* dialog box using *one* of the following three methods:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the menu bar, click **Mechanical** > **Fields** > **Plot Fields** > ***desired result***.
- Right-click **Field Overlays** in the Project Manager and choose **Plot Fields** > ***desired result*** from the shortcut menu.
- Right-click in the *Modeler* window and choose **Plot Fields** > ***desired result***.

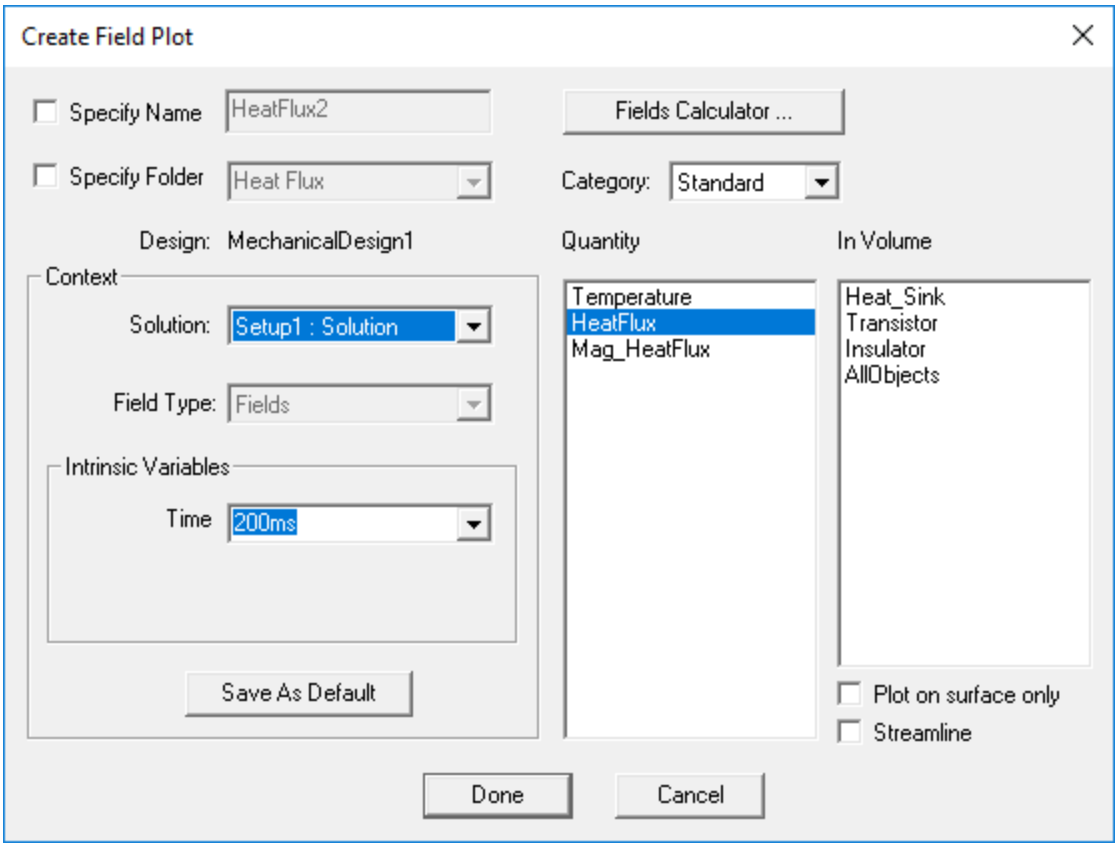
3. If you selected *Named Expression* in the previous step, click the desired named expression in the *Selecting calculated expression* dialog box that appears. Then, click **OK**.
4. In the *Create Field Plot dialog* box that appears, select the desired setup from the **Solution** drop-down menu (if you solved more than one analysis setup).
5. Optionally, if solid objects were selected for the overlay (as opposed to faces), you can clear the **Plot on surface only** option to view temperatures and heat flux magnitudes that occur in the interior of the solid objects.

   When this option is selected, the results are rendered on the model surfaces only. If you clear this option, the results are rendered for the full model volume, but they are rendered as IsoValSurfaces. Rather than the full volume of the model being rendered in varying colors, IsoValSurfaces connect points of the same resultant value and are rendered for each scale division specified in the plot settings (15 values by default). This plot style enables you to see the results on the interior of solids without creating clipping planes. The following heat sink temperature images compare the two plot styles:



   The following field plot setup example produces the heat flux vectors throughout the volume of the selected object or objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Choices appear in the *Intrinsics* section only if multiple variants or timesteps are available for plotting (for example, in parametric or transient analyses).

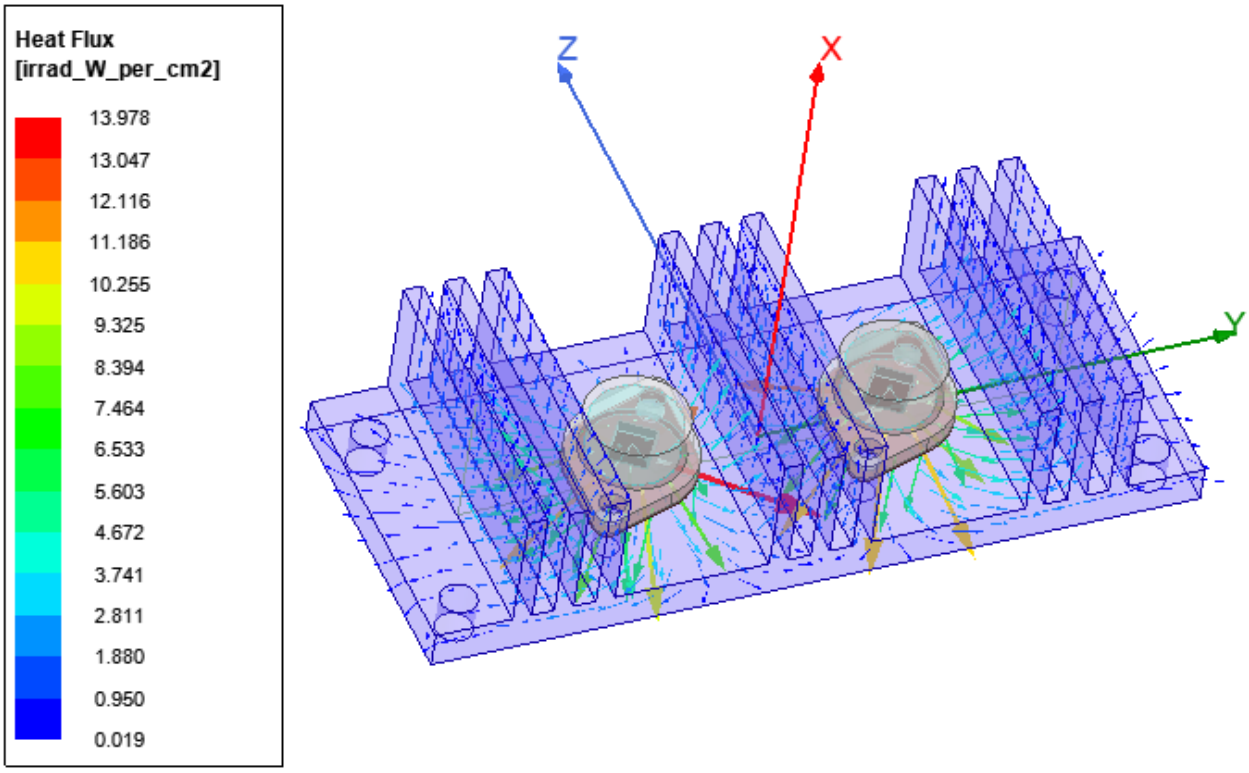
6. Click **Done**.

   The overlay and result legend appear on the model in the *Modeler* window.

7. Optionally, double-click the plot legend to access and modify the plot properties (scale, color map, and so on).

   The following image is a *HeatFlux* (vector) overlay. The legend's number format and units (W/cm^2) have been customized:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. If your solution has multiple sets of field results (such as when you run a transient solution or a parametric analysis to vary a design parameter), you can animate your thermal field results.
    a. Under *Field Overlays* in the Project Manager, right-click ***your field overlay*** and choose **Animate** from the shortcut menu.

       The *Create Animation Setup* dialog box appears. However, if your solution lacks multiple field result sets, an error message will appear indicating that the overlay cannot be animated.

    b. Optionally, change the **Name** and/or type a **Description** for the animation setup.
    c. Using the two drop-down menus in the *Swept Variable(s)* tab, choose the setup or variable you want to use as the basis of the animation.

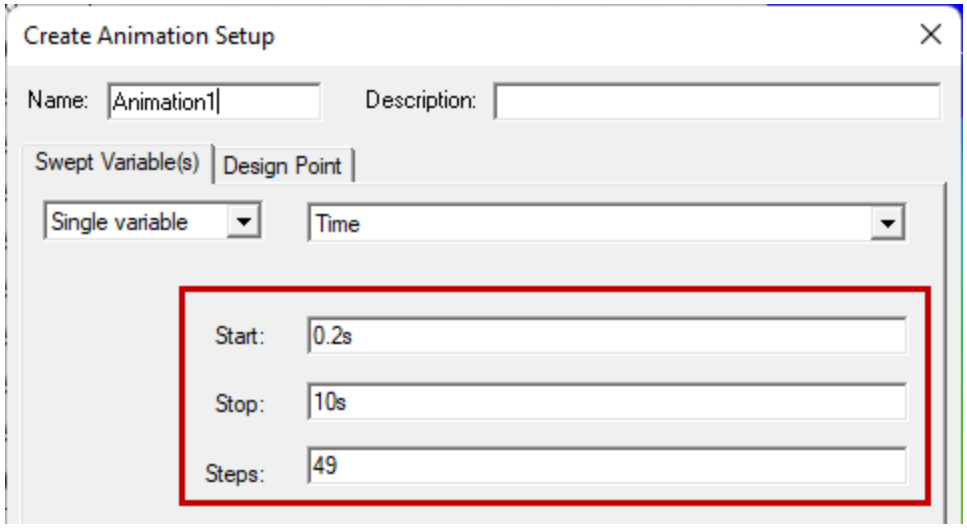
       For example:

       - If you want to base the animation on a single design variable, choose ***Single Variable*** and the ***variable name***.
       - If you want to base the animation on a parametric setup, choose ***Parametric Setup*** and the ***parametric setup name***.

    d. For transient solutions, choose the start, stop and step values in time units.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following example shows the setup for a 10 second transient solution with a timestep size of 0.2 seconds and every step saved:

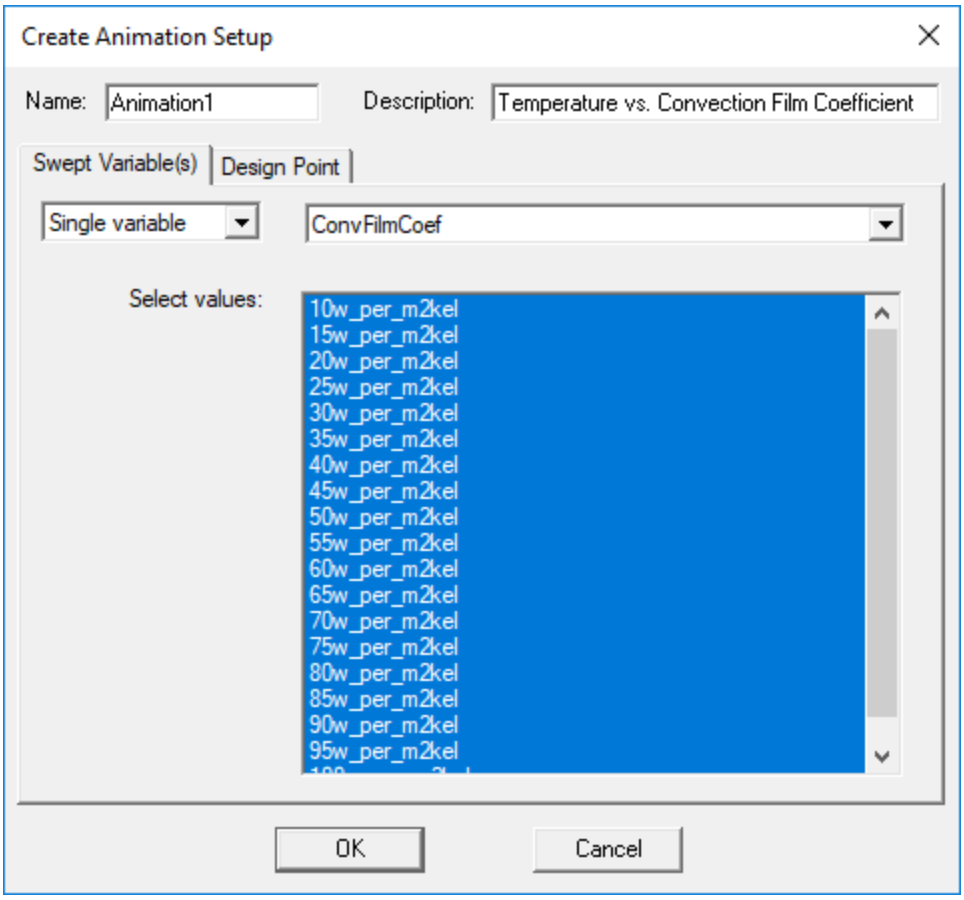


> **Note:**
>
> For transient solutions, you must choose the **Start**, **Stop**, and **Steps** values carefully. If the time values determined for every step do not correlate with the time values of the actual transient solution timesteps, blank overlays will occur for the animation frames. Also, remember to subtract **1** from the number of timesteps or animation frames to get the correct number of **Steps**.
>
> The **Start**, **Stop**, and **Steps** values are filled in automatically when you create an animation setup. The default is to include all available timesteps as an animation frame. If you later adjust the solution's transient setup and rerun it, the animation setup values will not be updated. In this case, the easiest solution is to delete the old animation setup and create a new one.

e. When applicable, select the variations you want to include in the animation.

   The following example shows all variations of a convection film coefficient variable (*ConvFilmCoef*) selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



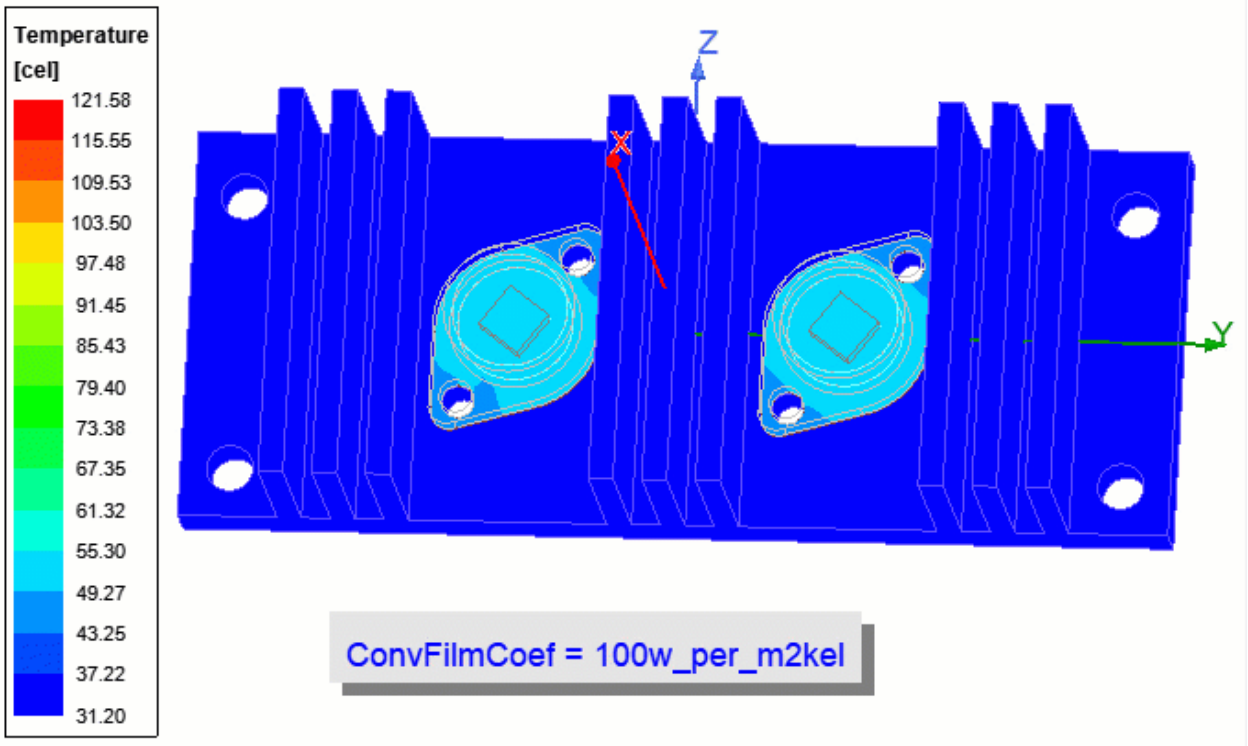
f. Click **OK**.

   The *Animation* dialog box appears and the overlaid results are animated.

g. Use the animation controls to stop, restart, reverse, increase/decrease the speed, or to go to a particular animation frame.

   An animation based on the preceding parametric example is shown below. The animation was reversed (to progress from the greatest to the smallest convection film coefficient) to show the frames in order of increasing temperature:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Fields Summary

A fields summary computes basic or derived field quantities and displays the data in a table in the **Fields Summary** dialog box. Currently, all results are reported in SI units (m, W, °C).

The temperature, heat flux magnitude, imported surface and volume EM loss densities, imported heat transfer coefficients, and mapped thermal conductivities (for layout components) that are available for plotting in reports and fields overlays are also available in the *Fields Summary* dialog box. Additionally, the following derived quantity is only available in the *Fields Summary* dialog box:

- **Heat Flow Rate:** This derived quantity is applicable to *Convection* boundaries, *Contact*, and faces (specified in face lists). It integrates the normal component of heat flux over the area of the selected faces or boundaries to determine the *Total* heat rate (in watts) flowing through the area.

The minimum (*Min*), maximum (*Max*), *Mean*, and standard deviation (*StDev*) values are reported along with the Area/Volume (of specified faces or solid objects, respectively). Additionally, *Normal* unit-vector components are reported for specified flat faces and flat boundaries. If multiple faces are included in a single boundary or calculation, the normal vector is only reported if all faces are flat and parallel and if the sign of the normal vector is the same.

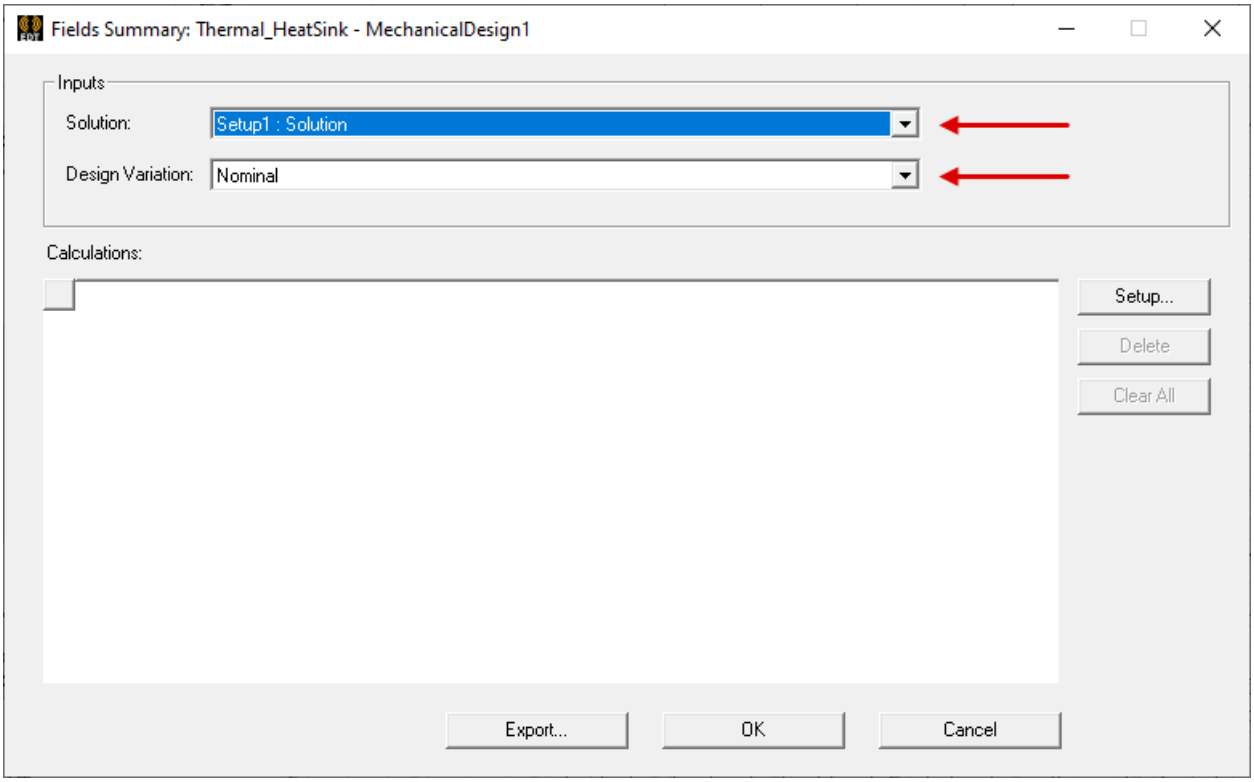
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fields Summary Procedure

1. Use one of the following two methods to begin creating a fields summary:
   - From the **Mechanical** menu, select **Fields** > **Create Fields Summary**.
   - In the Project Manager, right-click **Field Overlays** and select **Create Fields Summary** from the shortcut menu.

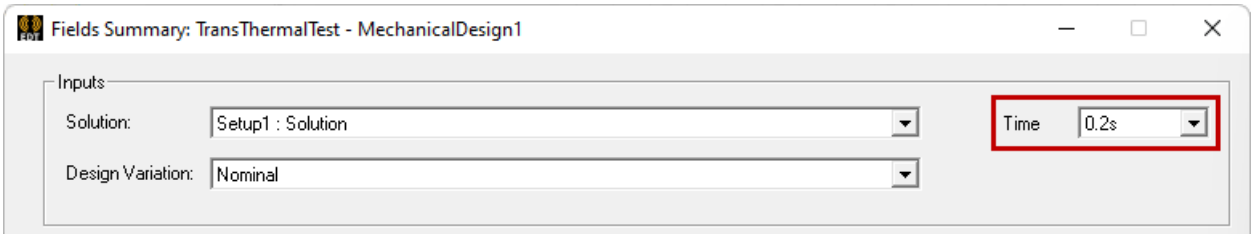
   The *Fields Summary* and *Setup Calculation* dialog boxes are displayed.

2. Under *Inputs* in the *Fields Summary* dialog box, select the **Solution** and **Design Variation** for which you want to create a fields summary.

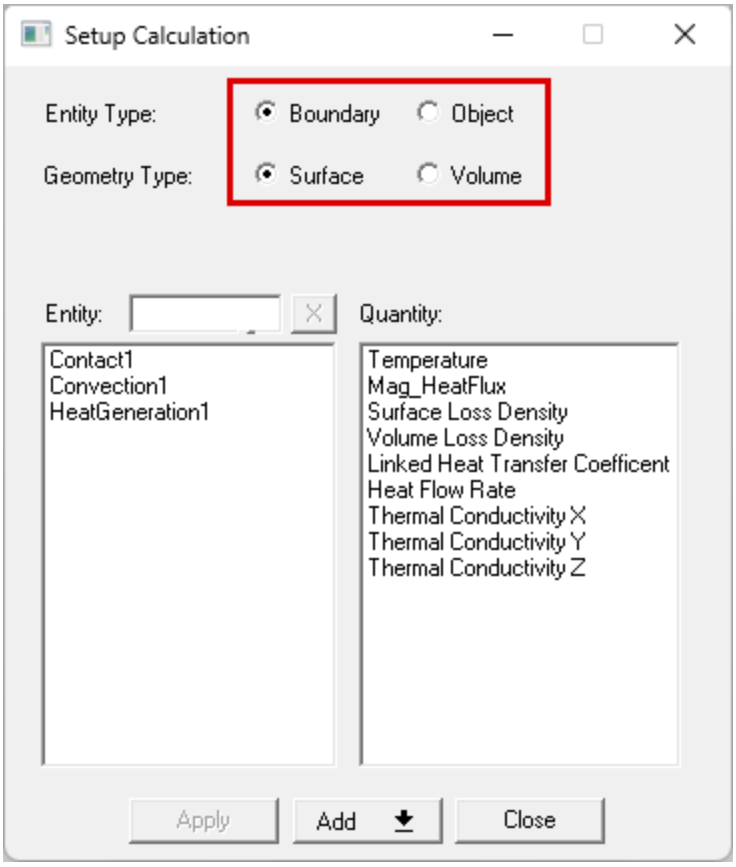


3. Optionally, for transient solutions, the dialog box also includes a **Time** drop-down menu. Choose the time for which to summarize the results:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. In the *Setup Calculation* dialog box, select the **Entity Type** and **Geometry Type**.



- **Entity Type:** Select **Boundary** (thermal boundary or excitation) or **Object** (geometry)
- **Geometry Type:** Select **Volume** or **Surface**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When **Boundary** and **Volume** are selected:

- Only thermal boundaries and excitations that are assigned to a solid object are available under **Entities** (for example, Temperature, Heat Generation, EM Loss).
- Though Convection boundaries can be assigned to objects, they are actually assigned to all **faces** of a selected assignment object, not the volume, and are therefore not available for volumes in the *Field Summary* dialog box.
- Similarly, though Rotating Fluid boundaries are assigned to objects, the assignment also includes a face (the Band Surface). Therefore, this boundary type is also not applicable to volumes in the *Field Summary* dialog box.

When **Boundary** and **Surface** are selected:

- Only thermal boundaries and excitations that are assigned to faces are available under **Entities** (for example, Contact, Convection, EM Loss, Heat Flux).

When **Object** and **Surface** are selected, the following **Entities** are available:

- All solid objects (selecting a solid object is equivalent to selecting all of its faces)
- All shell objects
- Global and relative coordinate system planes (XY, YZ, and XZ)
- Face lists

5. Select the **Entity** and **Quantity** you want to display in the fields summary.

   Depending on your choices in step 3, complete one of the following three actions to define the **Entity**:

   - **Boundary:** Select one or more boundaries or excitations.
   - **Object** and **Volume:** Select one or more solid geometry objects.
   - **Object** and **Surface:** Select one or more solid or shell objects, planes, or face lists.

   Then, choose the **Quantity** to summarize.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

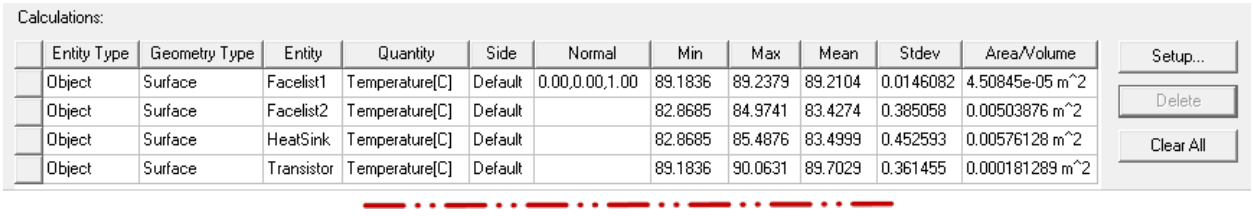
When **Surface** is selected:

- **Heat Flow Rate**, unlike other quantities, is not a field result that is directly available for plotting or overlaying on the model. Rather, it is a calculated quantity based on heat flux results. Specifically, it is the integration of the surface-normal heat flux components along the specified boundaries, faces, or planes. Unlike heat flux, which is the thermal power flow **per unit area**, Heat Flow Rate is in units of **power**.

When **Volume** is selected:

- Only quantities that produce nodal values are available (for example, Temperature, Mag_HeatFlux, Volume Loss Density).

6. From the **Add** drop-down menu, select **Add As Single Calculation** or **Add As Multiple Calculations** to add rows for data.
   - **Add As Single Calculation** adds one row displaying the quantity for all selected entities collectively.
   - **Add As Multiple Calculations** adds a separate row of the selected quantity for each selected entity.

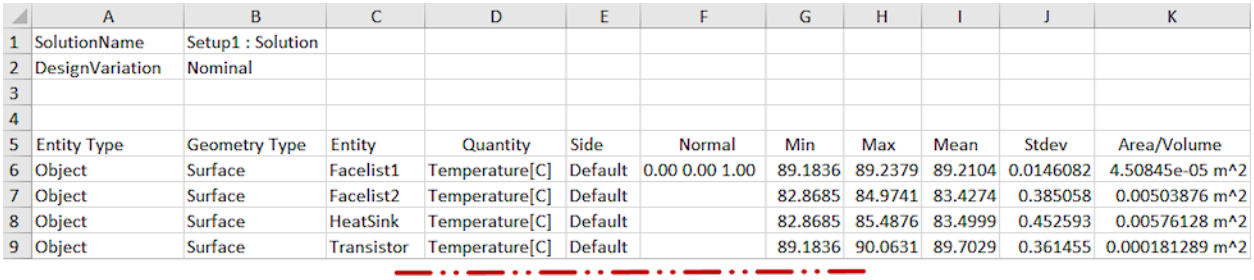


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can double-click a row in the fields report to open the **Setup Calculation** dialog box. You can then edit the row's definition and click **Apply** to change it.
>
> If the *Setup Calculation* dialog box has been closed and you want to add another summary, click **Setup** to reopen it.

7. Optionally, click **Export** (or **Apply and Export**, if the settings haven't been saved yet) to open the **Save As** dialog box. Then, save a comma-separated values (.csv) file of the fields summary data to the desired folder on your computer or network.

   The .csv file looks like the following image when opened in a spreadsheet program, such as Excel (with column widths and text alignment adjusted):



	A	B	C	D	E	F	G	H	I	J	K
1	SolutionName	Setup1 : Solution									
2	DesignVariation	Nominal									
3											
4											
5	Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev	Area/Volume
6	Object	Surface	Facelist1	Temperature[C]	Default	0.00 0.00 1.00	89.1836	89.2379	89.2104	0.0146082	4.50845e-05 m^2
7	Object	Surface	Facelist2	Temperature[C]	Default		82.8685	84.9741	83.4274	0.385058	0.00503876 m^2
8	Object	Surface	HeatSink	Temperature[C]	Default		82.8685	85.4876	83.4999	0.452593	0.00576128 m^2
9	Object	Surface	Transistor	Temperature[C]	Default		89.1836	90.0631	89.7029	0.361455	0.000181289 m^2

8. Choose one of the following two actions:
   - Click **OK** to save the settings and close the *Fields Summary* dialog box. When you open the dialog box again, the data for the saved settings is displayed.
   - Click **Cancel** to close the **Fields Summary** dialog box without saving the settings or summary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - Structural Solutions

Use the *Structural* solution type to determine stresses and deformation induced as a result of thermal expansion or contraction of solid objects. Thermal deformation occurs as the material temperature increases or decreases relative to the environment temperature, which is also the temperature at which the materials are strain-free and stress-free. The amount of expansion is determined by the change in temperature (relative to the environment) and the material's coefficient of thermal expansion (CTE).

Stresses occur when the thermal expansion of a material is physically impeded, either by a support boundary restricting motion locally or due to two or more materials with differing CTEs or temperatures being bonded to each other. In the latter case, the two materials are trying to expand or contract in differing amounts but are forced to deform equally along the bonded interface between them. An example is the interface between a semiconductor pad or pin and the solder. Another example is the interface between the solder and the copper traces of a printed circuit board.

> **Important:**
>
> Product licensing for Structural solutions differs from other design and solution types within the Ansys Electronics Desktop application. See the Mechanical Licensing Requirements topic for details.

This section contains information regarding the material assignment, setup options, and results that are available for a *Mechanical* design, *Structural* solution type. Specifically, the following topics are included:

- Assigning Materials
- Boundaries
- Excitations
- Solution Setup
- Results

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The *Thermal Expansion Coefficient* must be defined for the selected materials.
- A temperature must be assigned to at least one object to produce stress and deformation results. It can be a uniform body temperature (using a *Thermal Condition* excitation) or imported temperature results from a separate Thermal solution.
- All objects within the model must be statically stable. That is, supports must be sufficient to prevent rigid body motion in all three global directions. Supports should be assigned judiciously to make the model statically stable without significantly impeding the expected thermal deformation of the assembly. Otherwise, stresses at supports will be exaggerated.

Assigning Structural Materials

When choosing a material for a structural analysis, you will see a very different set of materials and material properties compared to an electronic circuit or electromagnetics analysis. Instead of properties such as Electrical Conductivity and Relative Permittivity, the physical properties of interest for structural analyses are the material's Young's Modulus, Mass Density, Poisson's Ratio, Shear Modulus, and Thermal Expansion Coefficient. Only materials with four of these five properties defined will appear in the *Select Definition* dialog box, where you choose the default material or assign an object material.

The one property that typically will be undefined is the Shear Modulus. For isotropic materials, the *Shear Modulus* (G) is calculated from the Young's Modulus (E) and Poisson's Ratio (v), according to the following equation:

$$G = E / [2 * (1 + v)]$$

Therefore, the shear modulus is listed as *Undefined* for the library materials, which are isotropic solids. You can specify shear moduli in three directions for *anisotropic* materials.

Anisotropic Properties

Materials for structural solutions can have simple isotropic properties (that is, behaving quantitatively the same way in every direction), or anisotropic properties (exhibiting different behavior in three orthogonal directions). Specifically, the Young's Modulus, Poisson's Ratio, Shear Modulus, and Thermal Expansion Coefficient can vary in each of three orthogonal directions. These directions are defined by the **Orientation** property of each object that uses the material (indicating a global or user-defined coordinate system).

All predefined library materials are of the simple (isotropic) type. You must define your own custom material to use anisotropic properties.

See the Defining Anisotropic Tensors topic for more information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature-Dependent Properties

Structural materials can also have temperature-dependent properties. Specifically, the Mass Density, Young's Modulus, Poisson's Ratio, and Thermal Expansion Coefficient can vary as a function of temperature.

> **Note:**
>
> The following restrictions exist:
>
> - If you define a thermal modifier for **Young's Modulus (E)**, you must also define a thermal modifier for **Poisson's Ratio (v)** and vice versa. Either *both* of these properties must be temperature-dependent or *neither* of them.
> - If you define the thermal modifiers for **E** and **v** using dataset tables, as opposed to general expressions (linear, quadratic, conditional, or so on), you must use the exact same temperature values for both datasets. Otherwise, you will encounter material validation errors.
> - If your design will experience temperatures below 0° C, you must define the thermal modifiers for all applicable properties using manually defined dataset tables. General expressions are automatically converted to tables for the solver, and these automatically generated tables only contain temperature datapoints in the range of 0–1000° C.

All predefined library materials are of the non-temperature-dependent type. You must define your own custom materials to use temperature-dependent properties.

See the Specifying Thermal Modifiers topic for more information.

Default Material and Applicable Libraries

The default material for structural analyses is *steel_1008*, which is a standard AISI low-carbon grade of steel. The following libraries contain materials suitable for Structural solutions:

- *[sys] GRANTA Materials Data for Simulation*
- *[sys] GRANTA Producers Materials Data for Simulation*
- *[sys] Materials*
- *[sys] Vacuumschmelze*

Non-Applicable Materials

Fluid materials are not applicable to structural analyses and are filtered from the libraries.

Material Selection

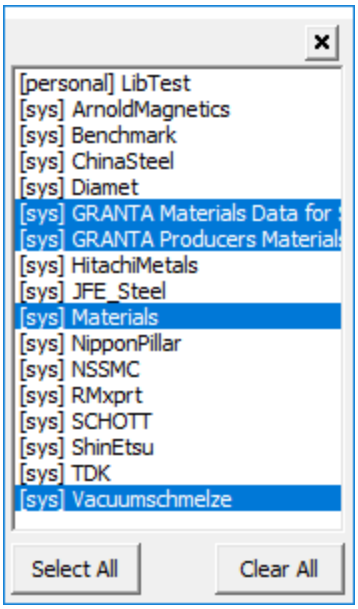
To set the default material for all objects you subsequently create:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. At the far right end of the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.

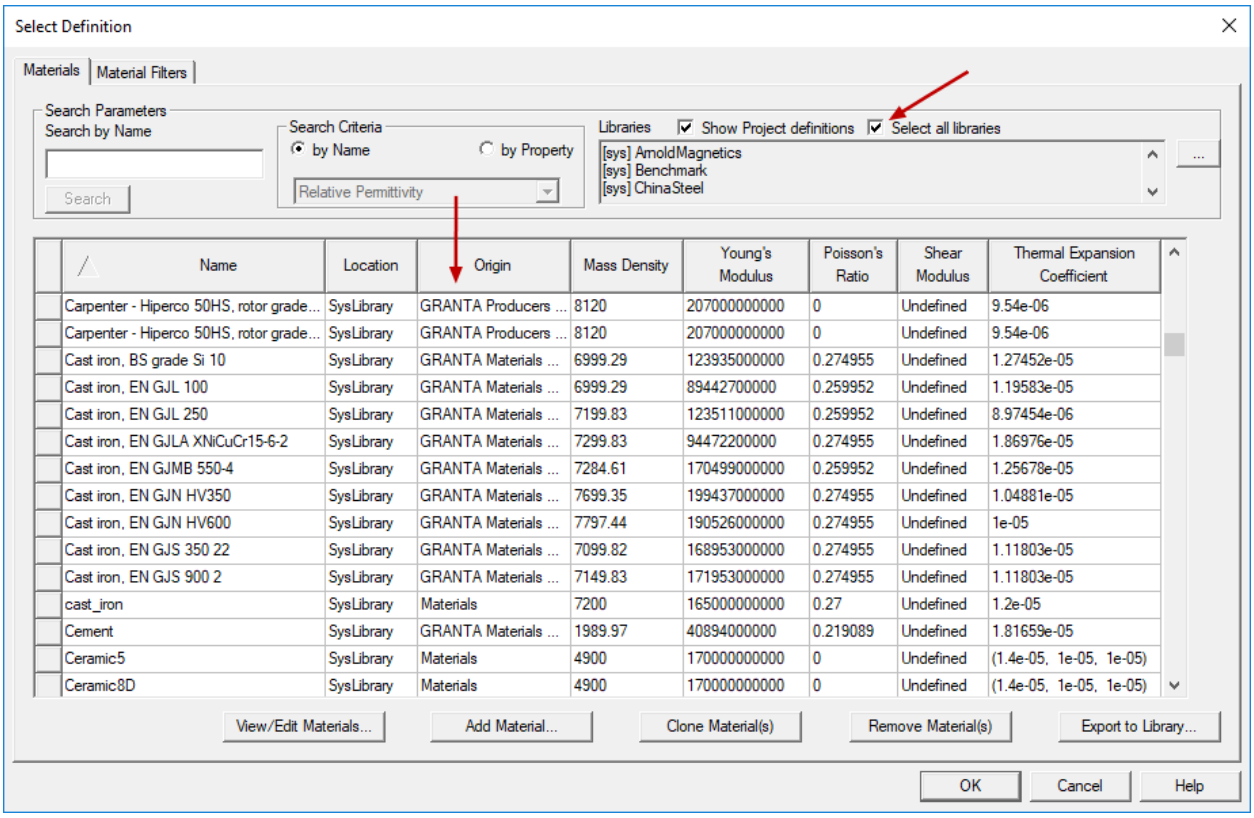
   The *Select Definition* dialog box appears.

2. In the *Libraries* list, ensure that one or more of the previously listed suitable libraries are selected, or a user library with mechanical material properties. Click the ellipsis button (**...**) to the right of the *Libraries* list to access a pop-up dialog box in which you can select which libraries to include in the *Materials* list:



   Alternatively, select the **Select all libraries** option:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

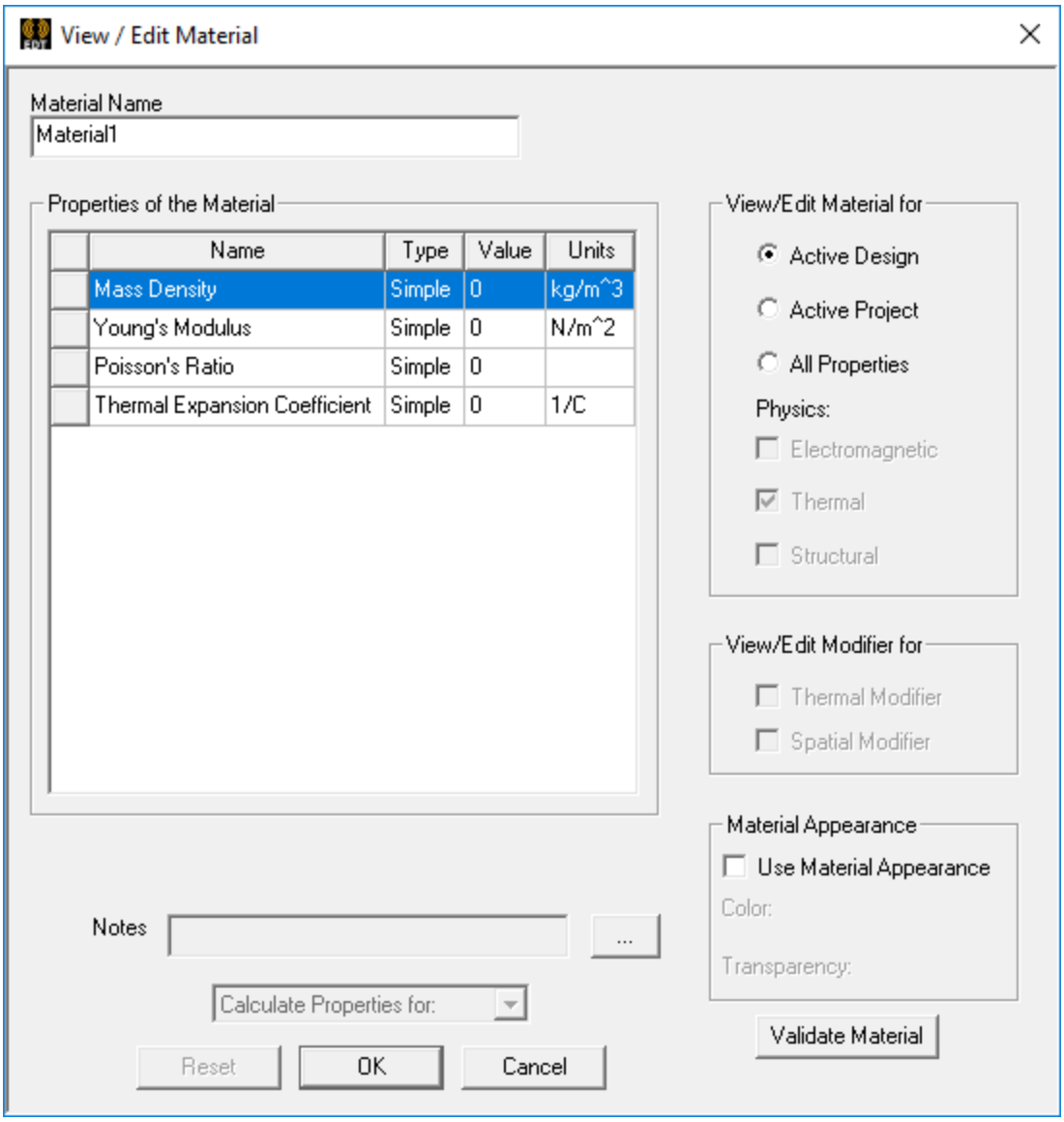


This option is selected by default for new projects.

3. Optionally, type a portion of the material name in the **Search by Name** text box. The list will automatically scroll as needed and the first material matching the search string will be selected. If you scroll the selected material off screen, click **Search** to quickly bring the searched material back into view.
4. Select the desired material, if present.
5. Optionally, to define your own material properties, click **Add Material**.

   The *View / Edit Material* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optionally, to specify anisotropic properties for the **Young's Modulus, Poisson's Ratio, Shear Modulus** or **Thermal Expansion Coefficient** (when properties vary in the three global directions), select **Anisotropic** from the **Type** drop-down menu. Otherwise, the *Simple* option is for isotropic properties (when the properties are the same in every direction).
7. Optionally, if you want to define temperature-dependent material properties, follow the instructions in the Specifying Thermal Modifiers topic.
8. If editing properties, or defining a new material, specify the desired **Material Name, Mass Density, Young's Modulus, Poisson's Ratio** and **Thermal Expansion Coefficient**.

   The following material validation rules apply:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For simple (isotropic) materials:
  - Mass Density (ρ) must be greater than zero: ($\rho > 0$)
  - Young's Modulus (E) must be greater than zero: ($E > 0$)
  - Poisson's Ratio (v) must be in the range: ($-1 < v < 0.5$, non-inclusively)
  - Thermal Expansion Coefficient (α) must be greater than zero ($\alpha > 0$)
- For anisotropic materials:
  - Mass Density (ρ) must be greater than zero: ($\rho > 0$)
  - Young's Modulus (E) must be greater than zero in all three directions: [for E, T(1,1) > 0, T(2,2) > 0, and T(3,3) > 0]

    For *Young's Modulus*, T(1,1), T(2,2) and T(3,3) respectively correspond to E_x, E_y, and E_z (the moduli in the three axis directions, whether the coordinate system is global or user-defined).

  - Shear Modulus (G) must be greater than zero in all three directions: [for G, T(1,2) > 0, T(1,3) > 0, and T(2,3) > 0]

    For *Shear Modulus*, T(1,2), T(1,3) and T(2,3) respectively correspond to G_{xy}, G_{xz}, and G_{yz} (the moduli based on the three axis directions, whether the coordinate system is global or user-defined).

  - Material must be positive definite as defined by the following function of Young's Modulus (E) and Poisson's Ratio (v):

    $(1 - v_{12}^2 * E_2/E_1 - v_{23}^2 * E_3/E_2 - 2 * v_{12} * v_{23} * v_{13} * E_3/E_1) > 0$

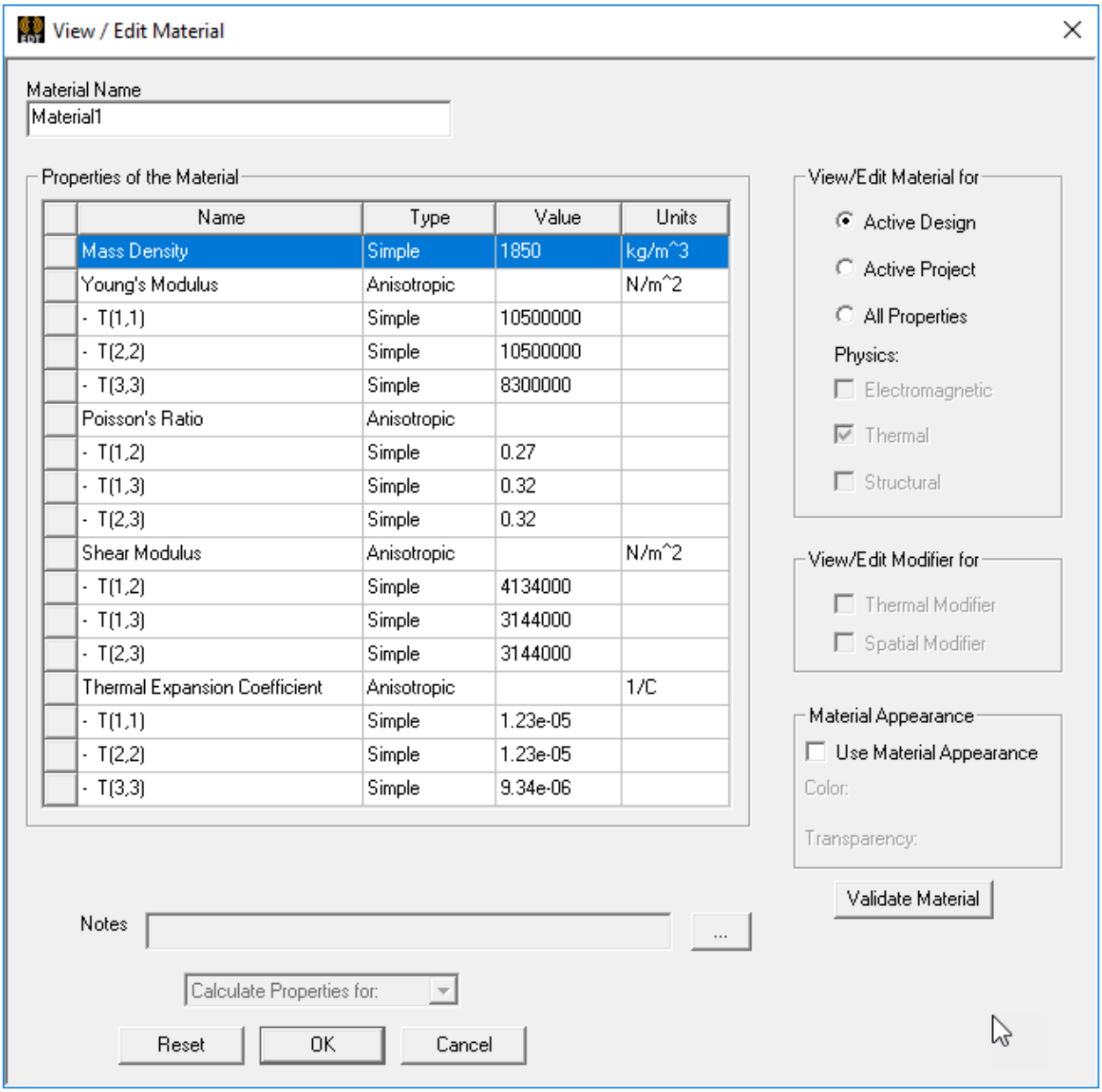
    For *Poisson's Ratio*, T(1,2), T(1,3) and T(2,3) respectively correspond to v_{xy}, v_{xz}, and v_{yz} (the ratios based on the three axis directions, whether the coordinate system is global or user-defined).

  - Thermal Expansion Coefficient (α) must be greater than zero in all three directions: [for α, T(1,1) > 0, T(2,2) > 0, and T(3,3) > 0]

    For *Thermal Expansion Coefficient*, T(1,1), T(2,2) and T(3,3) respectively correspond to α_x, α_y, and α_z (the expansion coefficients in the three axis directions, whether the coordinate system is global or user-defined).

The following is an example of the properties for an anisotropic material:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For temperature-dependent materials:
  - If Young's Modulus (E) has a thermal modifier defined, Poisson's Ratio (v) must also have a thermal modifier defined, and vice versa.
  - When using datasets to define the thermal modifiers for E and v, all temperature datapoints must be identical between the two tables.

9. Optionally, select the **Use Material Appearance** option and specify the desired **Color** and **Transparency** attributes.
10. Optionally, click **Validate Material** to check for errors in the specified properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the properties pass the validation check, a green check mark appears to the right of the *Validate Material* button. Otherwise, a red X appears, and a pop-up message provides the error details.

11. Click **OK** to close the *View / Edit Material* dialog box.
12. Click **OK** to close the *Select Definition* dialog box.

To assign a different material to a selected solid object:

1. Select the object to which you want to assign a material.
2. Use one of the following five methods of accessing the *Select Definition* dialog box:
    - On the **Draw** ribbon tab, click **Material**.
    - Using the menu bar, click **Modeler** > **Assign Material**.
    - Right-click in the **Modeler** window and choose **Assign Material** from the shortcut menu.
    - In the **Attribute** tab of the docked *Properties* window, choose **Edit** from the **Material** *Value* drop-down menu.
    - If the *Properties* dialog box appears automatically when you create an object, select the **Attribute** tab. Then, choose **Edit** from the **Material** *Value* drop-down menu.
3. Complete steps 2 through 6 of the previous procedure.

To add, remove, or edit materials:

1. Use one of the following two methods of accessing the *Edit Libraries* dialog box:
    - Using the menu bar, click **Tools** > **Edit Libraries** > **Materials**.
    - Under *Definitions* in the Project Manager, right-click **Materials** and choose **Edit Library** from the shortcut menu.
2. Select the desired material library from the *Libraries* list, or select **Show all libraries**.
3. Select an existing material or click **Add Material** to define a new one.
4. Optionally, you can click the following buttons once you've selected one or more existing materials:
    - **View / Edit Materials:** To change the name or properties.

      > **Note:**
      >
      > When you change the properties of a *SysLibrary* material, a new *Project* library material is created.

    - **Clone Material(s):** Make copies of the selected materials as a starting point for defining new ones.
    - **Remove Material(s):** Remove the selected materials from the library.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You cannot remove predefined *SysLibrary* materials. You can only delete materials that you've added to the project library or a user library (and only if the material is not being used in the current design).

5. After making the desired changes, click **OK** to close the *View / Edit Material* and/or *Edit Library* dialog boxes.

Boundaries (Structural)

In structural analyses, boundaries are used to represent areas where the model is supported and therefore constrained against motion in one or more directions. The words "support" and "constraint" are frequently used interchangeably.

The following three types of supports are available in Mechanical – Structural analyses. Click the links for more information on each type:

- **Fixed Support**
- **Frictionless Support**
- **Cylindrical Support**

> **Note:**
>
> Solid elements only have translational degrees of freedom (DOF). Individual nodes along a constrained face are prevented from translating in the X, Y, or Z direction but are free to rotate. However, applying a support to a face also indirectly prevents rotation in one or more directions. The reason is that the face cannot rotate without its nodes translating in the X, Y, and/or Z direction.

General Usage Notes Common to all Boundaries

- You can include different boundary types on different faces to best represent the behavior of the structure being modeled. For example, you could have a fixed support on one face, cylindrical supports on other faces (with various radial, tangential, or axial fixity), and a frictionless support on yet another face.
- The goal is to make the model statically stable (that is, to prevent rigid body motion in all three global directions. At the same time, you have to be careful not to overconstrain the model, reducing the accuracy of the stress and deformation results. Be aware that a face with a fixed support assigned to it will not be free to expand or contract when the temperature increases or decreases. This effect causes the stress results at the face to be exaggerated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- While you are permitted to combine two or more boundary types on the same face, there are currently no supports for which it would make sense to do so.
- Any of the available boundary types can be included in the definition of a 3D component.

Excitations (Structural)

The following excitations are available for *Structural* solutions in Ansys Electronics Desktop. Click the links below to see complete information about the excitations and how to apply them:

- **Pressure:** Apply a uniform pressure to selected object faces. The pressure can be normal to each element face, or you can specify pressure components based on any coordinate system.
- **Force:** Apply uniform X, Y, and Z components of force to selected object faces (global and relative coordinate systems are supported). Or, import non-uniform surface or volumetric force distributions from another design's solution.
- **Displacement:** Displace faces or edges a specified distance and direction or constrain faces or edges against displacement in specified directions. Global and relative coordinate systems are supported, and the normal direction can be specified for faces.
- **Thermal Condition:** Apply a specified uniform temperature or non-uniform temperature distribution to selected solid objects.

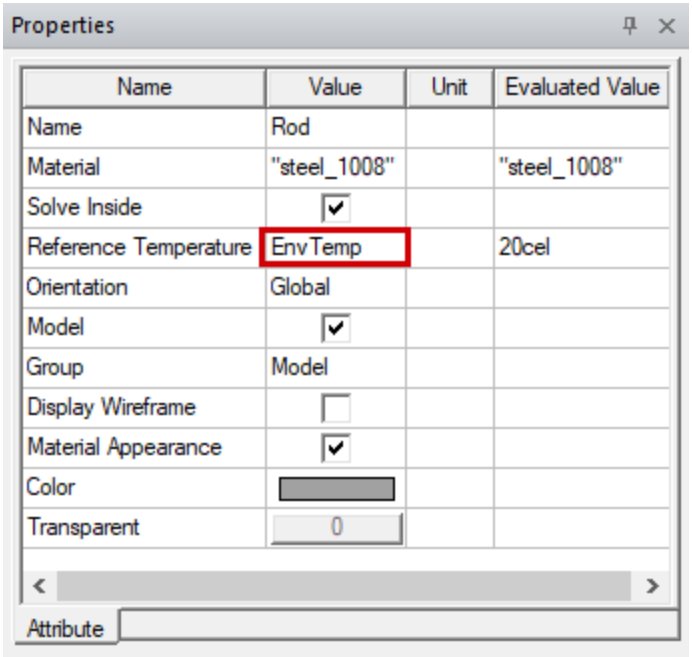
Specifying the Reference Temperature

In order to determine thermal stresses, a stress-free (also known as strain-free) reference temperature must be known for all objects. Resulting stresses are not based on the assigned or resultant temperature of the objects but rather on the difference between the assigned or resultant temperature and the specified reference temperature.

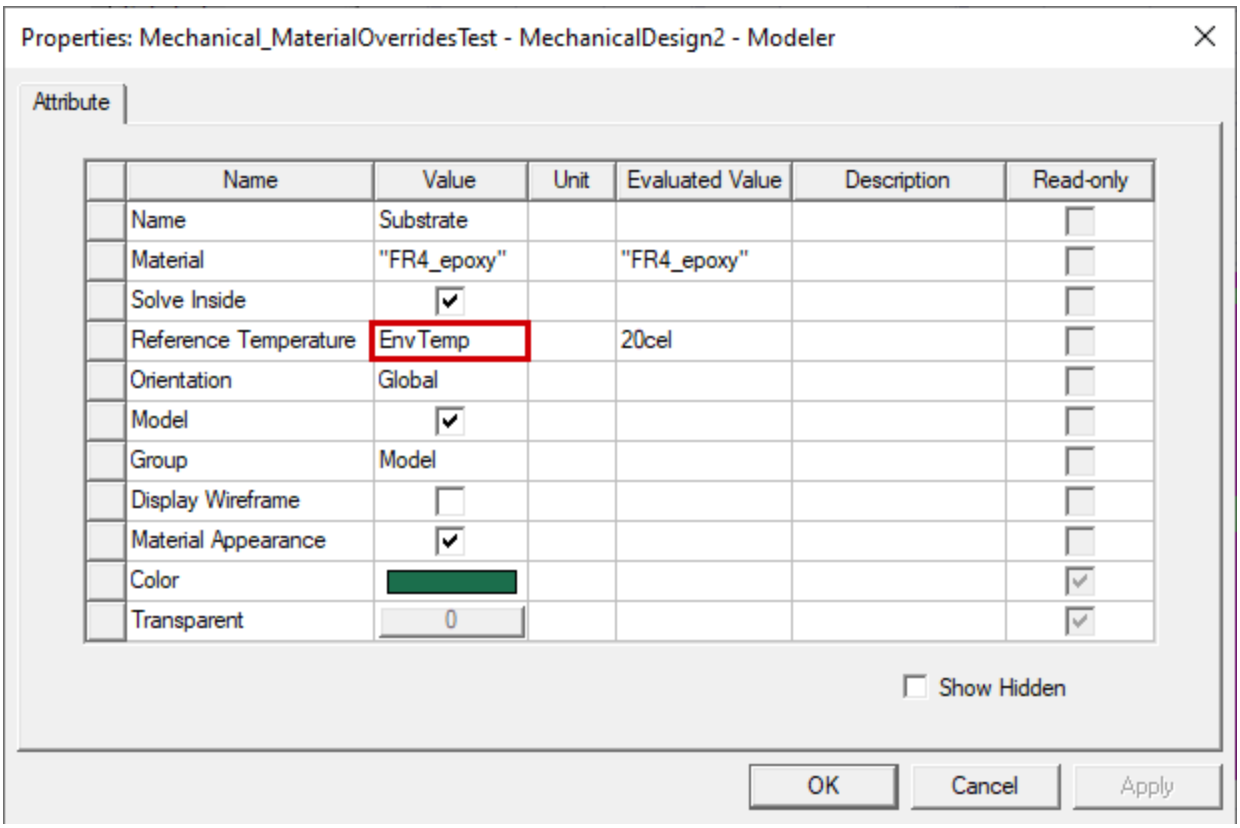
By default, the environment temperature ("*EnvTemp*" variable) is assumed as the reference temperature for all objects. However, in electronic assemblies, it is not uncommon for objects to be in a stress-free state at a temperature higher or lower than that of the environment. Additionally, it is not uncommon for different components to be stress-free at different temperatures. For example, 63% tin / 37% lead eutectic solder melts and freezes at 183° C. After solidification, the solder cools and, in so doing, experiences thermal contraction. This thermal contraction is resisted by the adjacent objects to which the solder is bonded, which may be expanding or contracting at a different rate. This phenomenon results in a state of residual stress when the board is cooled to room temperature. In such cases, the resultant stresses actually decrease as the assembly is heated. Conversely, objects that are mechanically assembled at room temperature, such as by fasteners, will be stress-free at room temperature, and stresses for these objects will increase as the assembly is heated.

The reference temperature is an object attribute and is defined in the docked *Properties* window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Or, alternatively, it can be defined in an object's *Properties* dialog box (select the object, right-click, and choose **Edit** > **Properties**):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

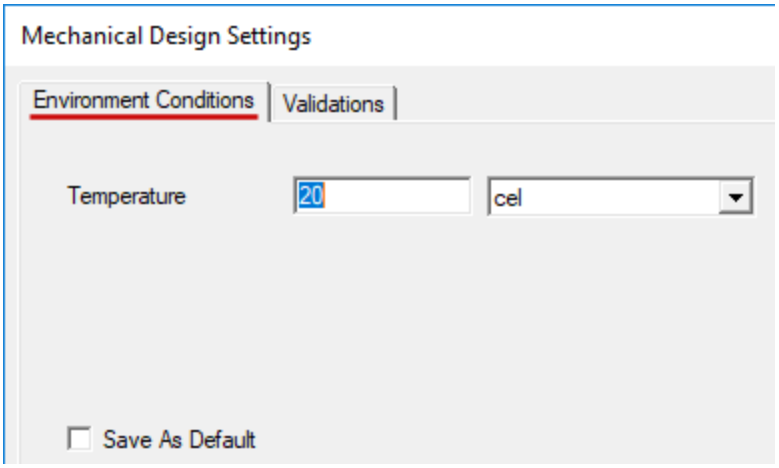
You only need to specify the **Reference Temperature** value when it differs from the default *EnvTemp* value. You can also alter the global **EnvTemp** value if a majority of objects share the same stress-free temperature.

You can specify the **Reference Temperature** using any of the following five forms:

- The default **EnvTemp** variable, which is a global design setting
- A numerical temperature value with units (such as **183cel**)
- A user-defined design variable (such as **Ta**)
- A user-defined project variable (such as **$Tf**)
- An expression comprised of constants, variables (either design or project type), and operators, for example: (Ta-ln($Tf^2))/1.5

To change the global **EnvTemp** value:

1. Access the *Mechanical Design Settings* dialog box using one of the following two methods:
   - In the menu bar, click **Mechanical** > **Design Settings**.
   - Right-click the structural design heading in the Project Manager [**MechanicalDesign*x* (Structural)**, by default]. Then, choose **Design Settings** from the shortcut menu.
2. In the *Environment Conditions* tab, specify the desired **Temperature** value and append the units (or use the adjacent drop-down menu to choose the **Units**).



3. Click **OK**.

Solution Setup (Structural)

You must add at least one solution setup to each Mechanical design. The solution setup for a structural analysis includes the following settings:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Name:** The default name is Setup*x*, where *x* is a numeric value incremented for each solution setup that you add to the design. You can customize the name of each setup as desired.
- **Enabled:** This option is selected by default. Deselect the option to prevent the setup from being solved when you click the *Analyze All* command.
- **Solver Type:** Specify the solver to use, or let the program choose the solver. The choices are:
  - **Program Controlled:** This is the default setting, which is recommended for most cases. The application selects the optimal solver.
  - **Direct**
  - **Iterative**

> **Note:**
>
> Consult the Ansys *Mechanical User's Guide* (specifically, *Configuring Analysis Settings* > *Analysis Settings for Most Analysis Types* > *Solver Controls*) for more information about the available solver choices.

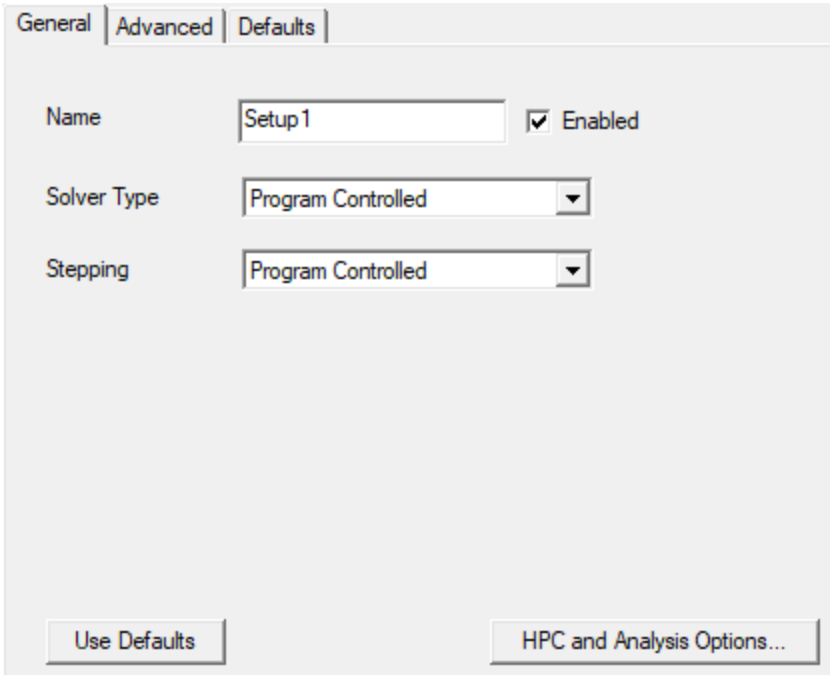
- **Stepping:** Specify whether stepping is enabled, or let the program choose. Stepping is the process of dividing the applied loads (excitations and boundaries) into smaller increments to make solution convergence easier. In the context of structural analyses, only the loads are divided, not time increments, since time is not applicable to static analyses.

  The *Stepping* options are:

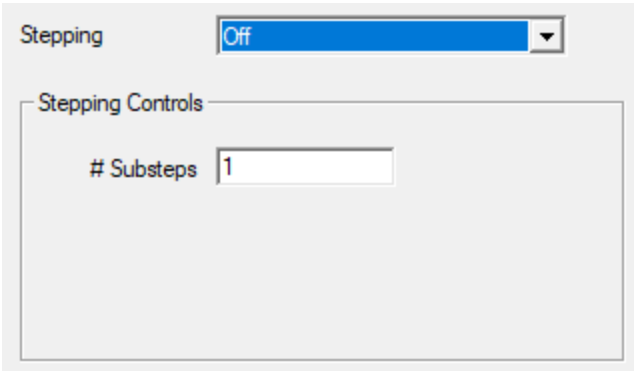
  - **Program Controlled:** This is the default setting, which is recommended for most cases. The application determines the stepping settings.

    There are no additional settings to specify when this *Stepping* option is selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Off:** Program controlled or user-defined automatic stepping is disabled. However, you can still specify a fixed number of calculation steps. When this option is selected, the following additional setting appears in the dialog box:

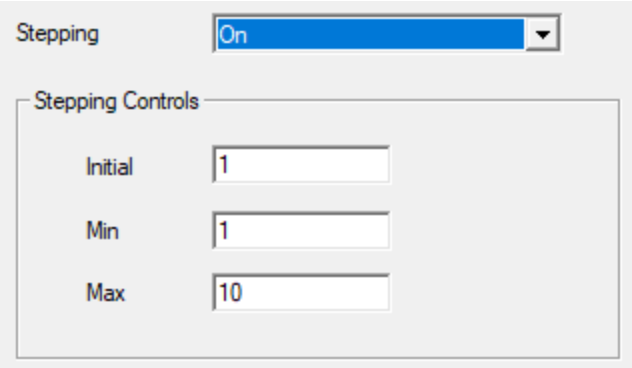


  - **# Substeps:** The number of substeps to solve.

    You can think of a substep as a load division. For example, if you apply a temperature of 100 C in a *Thermal Condition* excitation, and you specify two substeps, the solver increments the change in temperature in two steps. In each step, the temperature increments by half of the difference between the assigned temperature and the *Global* (that is, stress-free) *Temperature*. If the stress-free temperature is 20 C, then Delta T = 80 C total and 40 C per step. Therefore, the body temperatures for steps 1 and 2, respectively, would be 60 C and 100 C. The applied thermal loads (boundaries and excitations) are divided according to the number of steps you specify and solved as intermediate steps and a final step. (See Note below).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **On:** User-defined stepping is enabled. When this option is selected, the following additional settings appear in the dialog box:



  - **Initial:** The initial number of substeps to run. For the first solution increment, the applied thermal loads (boundaries and excitations) are divided according to the initial number of substeps to represent a "bite-sized" portion of the full thermal load. The default *Initial* substeps value is 1, in which case the full thermal load is applied initially. This number should be greater than or equal to the *Min* number of substeps (next bullet).
  - **Min:** The minimum number of substeps to be solved. The default is 1.

    If no convergence problems are encountered, the solver can reduce the number of substeps to something between the *Initial* and *Minimum* values, essentially increasing the size of the load step.

    If you specify *Min* greater than *Initial*, the solver will reset *Min* to be equal to *Initial*. Therefore, the value you enter for *Min* should be less than or equal to the *Initial* number of substeps you define.
  - **Max:** The maximum number of substeps to solve. The default is 10. More substeps than specified by the *Initial* setting are solved only if convergence problems are encountered. A greater number of calculation substeps corresponds to smaller load steps, making convergence easier.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

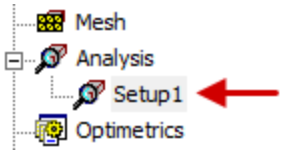
Note:

- While results are produced by the solver for each calculation substep, these intermediate results are not available for reporting in the Ansys Electronics Desktop application. Only the final solution results, based on the full values of the applied boundaries and excitations, are available for post processing.
- In the context of a structural analysis, you are not dividing the analysis into periods of time. You are only specifying load substeps, each of which has a static stress and displacement solution.
- In the absence of nonlinear materials and time-dependent or temperature-dependent excitations or boundaries, a structural analysis is likely to converge in a small number of steps without difficulty. Therefore, we recommend that you use the **Program Controlled** option, which allows up to ten substeps and as few as one.

How to Set Up a Structural Solution:

1. Use one of the following three methods of accessing the structural solution settings:
   - On the **Simulation** ribbon tab, click **Setup**.
   - Right-click **Analysis** in the Project Manager and choose **Add Solution Setup**.
   - Using the menu bar, click **Mechanical** > **Analysis** > **Add Solution Setup**.
2. Optionally, change the **Name** and any of the other default settings (**Solver Type** or **Stepping** options) that have been previously described in this topic.
3. Optionally, click **HPC and Analysis Options** to change any of your high performance computing configurations or options.
4. Optionally, specify settings for importing a mesh from another design or project. (See Mesh Linking.)
5. Click **OK**.

   The setup appears under *Analysis* in the Project Manager:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results (Structural)

Thermal analysis results presentations fall into two basic categories: *Fields Reports* and *Field Overlays*.

- **Fields Report:**

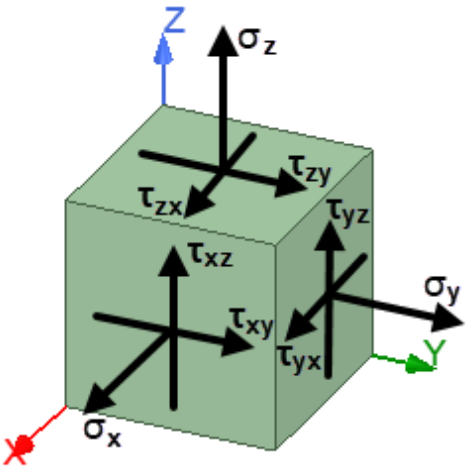
  There are several different display types available for fields reports, including 2D (Rectangular) Plots, 3D Plots, Data Tables, and more. One convenient way to generate a report of thermal field results is to draw a polyline running along the faces and/or through the interior volume of your model. This polyline, which can consist of multiple straight and curved segments, defines a path along which you wish to plot a particular structural result. A polyline is required for all *predefined* structural field results with the exception of *Design Volume*.

  You can also plot user-defined *Named Expressions* along a polyline or create named expressions that do not need a polyline for plotting.

  Regardless of the display type, the following predefined structural results are available for reporting:

  - **Mag_Displacement:** Displacement magnitude versus the distance along a specified polyline.
  - **Equivalent_Stress:** The von Mises equivalent stress result (in units of force per unit area) versus the distance along a specified polyline. The von Mises stress combines the effect of six stress tensors into a single positive value. This equivalent stress can be compared directly to a ductile material's yield strength or endurance limit as a predictor of permanent material deformation or fatigue failure. The following diagram and equations show how the equivalent stress is defined in terms of six stress tensors. The diagram shows nine tensors (three tensile/compressive tensors and six shear tensors). However, among the six shear tensors, there are only three unique values. The first of the two equation lines below shows the pairs of shear tensors with identical values. Therefore, the equivalent stress (σ_{eq}) equation is based on three tensile/compressive tensor values and three shear tensor values:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$\tau_{xy} = \tau_{yx};\quad \tau_{yz} = \tau_{zy};\quad \tau_{zx} = \tau_{xz}$$

$$\sigma_{eq} = \sqrt{0.5[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

(See Note 1.)

- **Temperature:** Temperature versus the distance along a specified polyline. Temperature results, for Structural solutions, are only applicable to values imported from Icepak (using a non-uniform Thermal Condition excitation).
- **Youngs_Modulus_X, Youngs_Modulus_Y,** and **Youngs_Modulus_Z:** (See Note 3.)

  Plotted versus the distance along a specified polyline, Young's Modulus (E) quantifies how much a material resists elongation or compression in the direction it is being strained. It is also referred to as a stress-strain curve, and is essentially linear for isotropic materials operating in the elastic range. The units are force per area, and the value represents the stress required to produce a unit of strain. The higher the Young's Modulus, the stiffer the material.

- **Poisson_Ratio_XY, Poisson_Ratio_YZ,** and **Poisson_Ratio_XZ:** (See Note 3.)

  Plotted versus the distance along a specified polyline, Poisson's Ratio (v) quantifies how the cross-section of a material sample changes as it is strained in the perpendicular direction, such as the way the cross section of a rubber band contracts as the band is streched. Three values are available for plotting, each being the negative of the ratio of transverse strain and axial strain ($-\varepsilon_{transverse}/\varepsilon_{axial}$) resulting from an applied axial load. Specifically, $v_{xy} = -\varepsilon_x/\varepsilon_y$, $v_{yz} = -\varepsilon_y/\varepsilon_z$, and $v_{xz} = -\varepsilon_x/\varepsilon_z$.

- **Thermal_Expansion_X, Thermal_Expansion_Y,** and **Thermal_Expansion_Z:** (See Note 3.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotted versus the distance along a specified polyline, the thermal expansion coefficient of a material, indicates how its length changes with temperature. Its units are 1 per unit temperature, but it is often thought of as delta-length per unit length pre unit temperature, where the length units cancel each other.

- **Shear_Modulus_X, Shear_Modulus_Y,** and **Shear_Modulus_Z:** (See Note 3.)

  Plotted versus the distance along a specified polyline, Shear Modulus (G), also know as the modulus of rigidity, is a measure of the rigidity of a material. Specifically, it is the ratio of the shear stress and shear strain. For the isotropic constituent materials (dielectric and conductor properties), the value of G is calculated from the specified Young's Modulus and Poisson's Ratio according to the relationship, G = E / [2 * (1+v)].

- **Equivalent_Strain:** The von Mises equivalent strain result versus the distance along a specified polyline. This result is dimensionless, though it is often expressed in canceling units of length per unit length. The equivalent strain combines the effect of six strain tensors. Similar to the stress tensors shown in the preceding diagram, there are nine strain tensors—three tensile/compressive tensors (ε_x, ε_y, and ε_z) and six shear tensors. However, as is true for the shear stress tensors, there are also only three unique values for the shear strain tensors (γ_{xy}, γ_{yz}, and γ_{zx}). The following equation shows how the equivalent strain (ε_{eq}) is calculated from the six strain tensor values and the *effective* Poisson's ratio (v'), which equals the specified Poisson's ratio for isotropic materials undergoing elastic strain:

$$\varepsilon_{eq} = \frac{1}{(1+v')\sqrt{2}}\left[(\varepsilon_x-\varepsilon_y)^2+(\varepsilon_y-\varepsilon_z)^2+(\varepsilon_z-\varepsilon_x)^2+\frac{3}{2}(\gamma_{xy}^2+\gamma_{yz}^2+\gamma_{zx}^2)\right]^{\frac{1}{2}}$$

  (See Note 1.)

- **Design Volume:** The volume of the objects comprising the model.
- ***Named Expression*:** A custom result previously defined using the Fields Calculator (see the Fields Calculator Note on the *Creating a Report* page).

- **Field Overlay:**

  You can overlay fields on selected solid or sheet objects, faces, or polylines. The following five results are available:

  - **Mag_Displacement:** Color contour overlay of the displacement magnitude.
  - **Displacement_Vector:** Vector overlay of the displacement.
  - **Equivalent_Stress:** Color contour overlay of the von Mises equivalent stress, which is defined in the preceding diagram and σ_{eq} equation (under *Fields Report*).

    (See Notes 1 and 2.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Equivalent_Strain:** Color contour overlay of the von Mises equivalent strain, which is defined in the preceding ε_{eq} equation (under *Fields Report*).

  (See Notes 1 and 2.)

- **Temperature:** Color contour overlay of the temperature distribution. Temperature results, for Structural solutions, are only applicable to values imported from Icepak (using a non-uniform Thermal Condition excitation). Use this overlay to verify that temperatures have been imported and mapped to the geometry correctly.
- **Youngs_Modulus_X, Youngs_Modulus_Y,** and **Youngs_Modulus_Z:** (See Note 3.)

  Color contour overlay of the effective Young's Modulus (E) values in the three coordinate axis directions. See the full description of **Youngs_Molulus_X, ...** in the above *Fields Report* section.

- **Poisson_Ratio_XY, Poisson_Ratio_YZ,** and **Poisson_Ratio_XZ:** (See Note 3.)

  Color contour overlay of the three effective Poisson's Ratios (v). See the full description of **Poisson_Ratio_XY, ...** in the above *Fields Report* section.

- **Thermal_Expansion_X, Thermal_Expansion_Y,** and **Thermal_Expansion_Z:** (See Note 3.)

  Color contour plot of the thermal expansion coefficients in the three coordinate axis directions.

- **Shear_Modulus_X, Shear_Modulus_Y,** and **Shear_Modulus_Z:** (See Note 3.)

  Color contour overlay of the three effective Shear Modulus (G) values in the three coordinate axis directions. See the full description of **Shear_Modulus_X, ...** in the above *Fields Report* section.

- ***Named Expression*:** A custom result previously defined using the Fields Calculator (see the Fields Calculator Note on the *Creating a Report* page).

- **Fields Summary:**

  Provide a Summary based on resultant fields data at boundaries, objects (volumes or surfaces), faces (from face lists), or along global or relative coordinate system planes. Summary information includes details such as the minimum, maximum, and mean field values, the standard deviation of the results, and the normal direction vector (for surfaces).

  (See Notes 1 and 2.)

These reports, overlays, and fields summary are described more fully, along with the instructions for creating them, in the subsections that follow.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

1. Generally, stress and strain results are averaged among the elements sharing a common node. The averaging is performed for the individual nodal stress and strain tensors before the equivalent stress and strain results are calculated. However, this averaging is *not* performed across object boundaries. Where two objects meet at coincident faces, the nodes are shared between the two objects, and bonded contact behavior exists. Though a node may be shared by two different objects, the nodal averaging is only performed for each set of elements belonging to a single object. Therefore, the nodal values associated with each object will differ from each other at the shared nodes (one unique nodal value for each contacting object). This behavior is important especially when the two objects have significantly different material properties. For any given amount of displacement or strain, the object with the lower stiffness (that is, Young's modulus) will typically have the lower nodal stress values, and the stiffer material will experience the higher nodal stresses. Averaging, in this case, would result in stresses being underestimated for the stiffer part and overestimated for the softer part.
2. The differing stress and strain values along contact faces will be evident in the Fields Summary when maximal or minimal stresses occur at a contact face (as long as the stresses are summarized separately for each object). The differences will also be evident if a field overlay is assigned to only one of two contacting objects and then compared to stresses with the overlay reassigned to the other object.
3. Youngs_Modulus, Poisson_Ratio, Thermal_Expansion, and Shear_Modulus results are only applicable to structural designs that include layout components (in which material properties are mapped to dielectric elements to represent the effect of metal traces and vias). In such cases, the properties can vary throughout the volume of the layout component based on the metal fraction mapped to its grid cells, and they can vary by direction, even though the constituent dielectric and conductor materials are isotropic. Therefore, three separate, directional values are available for plotting for each of these properties.

   Constituent dielectric and conductor materials must have isotropic properties. Anisotropic materials are not currently supported for layout components in Mechanical.

Fields Report

The following display types are available for fields reports in Mechanical – Structural solutions:

- 2D (Rectangular) Plot
- Stacked Plot

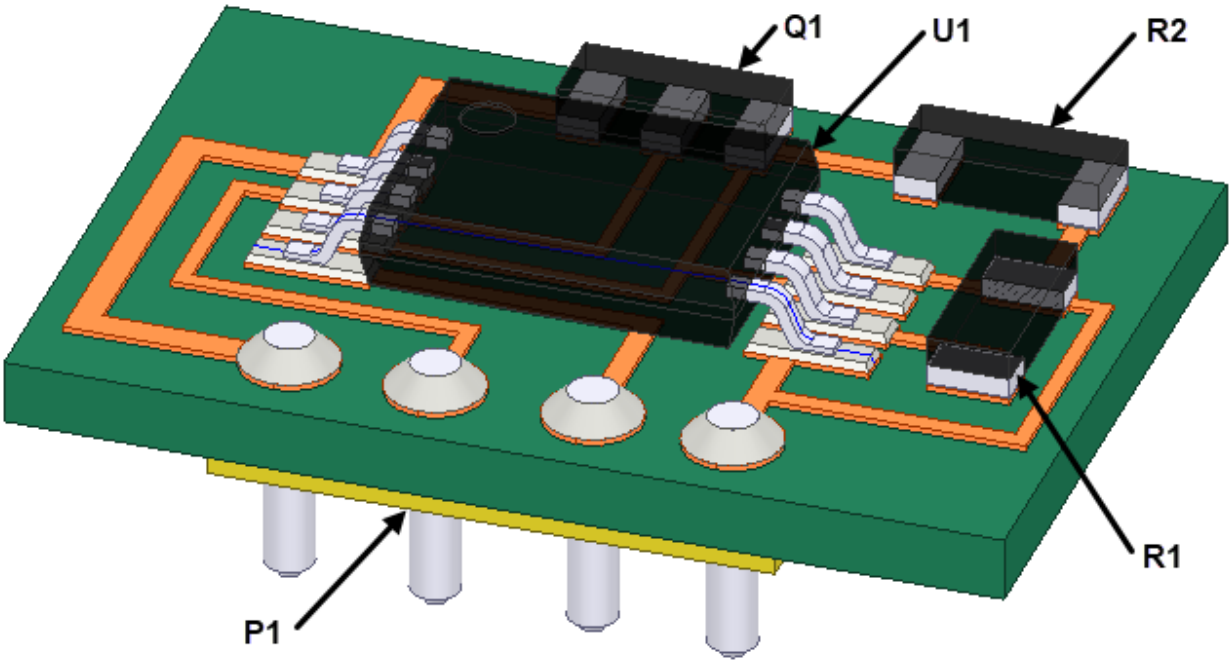
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Data Table
- 3D Plot
- 2D Contour Plot

In this topic, we will focus on the display types that are most likely to be used for structural reports (specifically 2D plots, Stacked plots, and Data Tables). The procedure for creating other types of plots is similar. The subtopic, Creating a Fields Report, details how to generate plots and data tables of structural fields results.

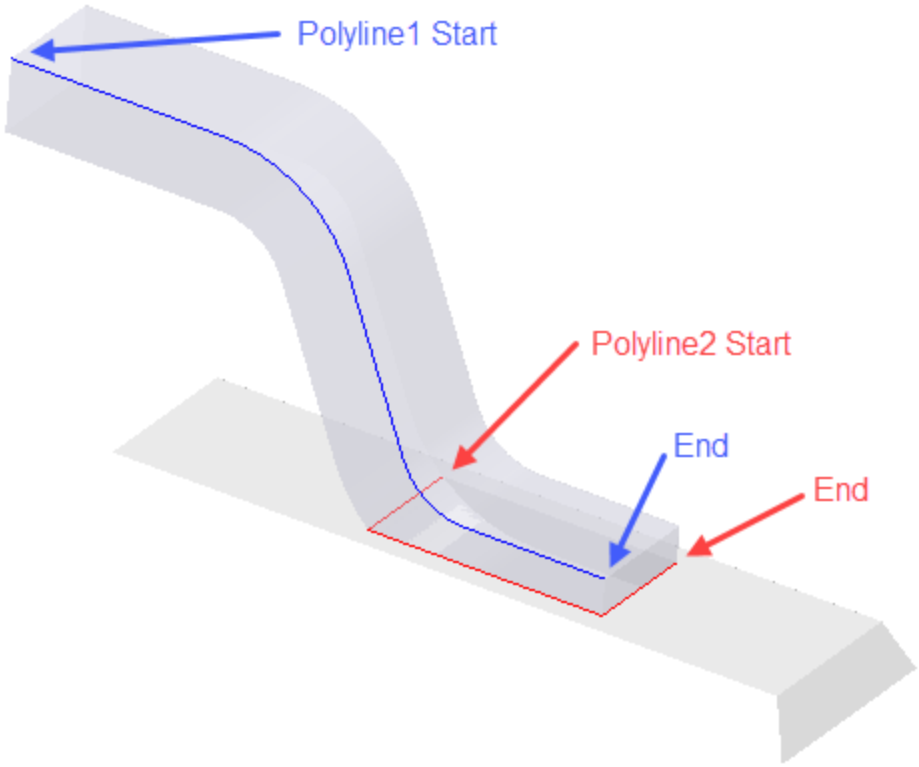
In order to create a fields report of the *predefined* structural results, an arc, line, or series of straight and/or curved polyline segments is needed. You can draw the line segments manually or select edges of your model and convert them to line objects. Then, specify this polyline geometry as the path along which the predefined thermal result values will be calculated and plotted.

For example, the following image is a sample thermal stress analysis model. It consists of a printed circuit board (PCB), pin header, integrated circuit (IC), transistor, two resistors, and the solder used to connect all of the parts to the PCB traces:



Two multi-segment polylines have been drawn, which are used to define paths for plotting the equivalent stresses along edges of interest. Polyline1 is drawn along the edges of Pin 5 of the IC. Polyline2 is drawn along three edges where the top of the solder intersects Pin 5:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

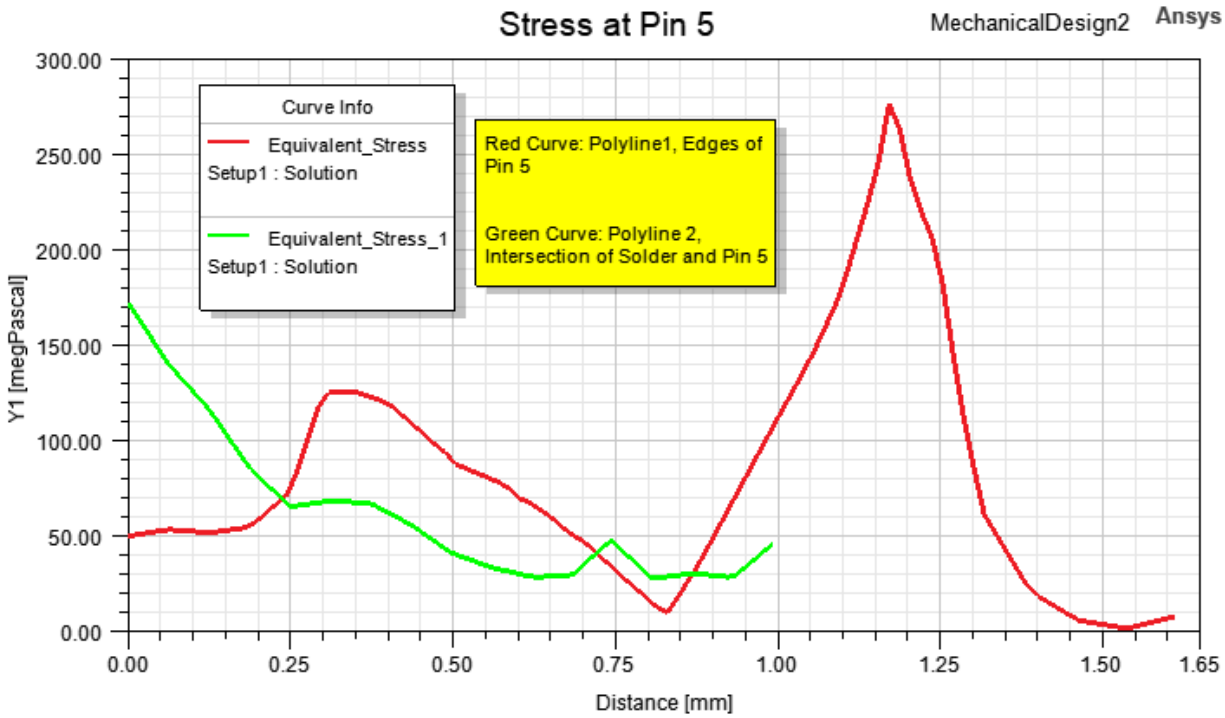
- Whether you draw the polyline before or after running the solution, it's a good idea to create it as a **non-model** part. However, the solver will ignore line and sheet objects even if the **Model** option is selected in the object properties. If you create a polyline after solving, you will be prompted to create a non-model object, or the solution results will be invalidated.
- For the preceding example, the points defining the arc segments along Polyline1 are drawn slightly below the curved faces and edges of Pin 5 (about 0.005 mm). When curved object faces are discretized into a mesh, they become faceted. A pure arc segment would not pass through the elements generated along the corresponding model edges, leaving gaps in any plotted or tabulated results. Moving the curve slightly inward prevents any gaps in the output.

The available *predefined* structural results are listed and described under Fields Report in the *Results (Structural)* topic. Examples of 2D plots, Stacked plots, and Data Tables follow:

- **2D (Rectangular) Plot:** This type of plot can contain one trace or multiple traces, each of which is plotted within the same Cartesian grid.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following image shows the *Equivalent_Stress vs. Distance* 2D (rectangular) plot based on the preceding model and polylines example. A note has been added to identify which curve corresponds to which polyline:

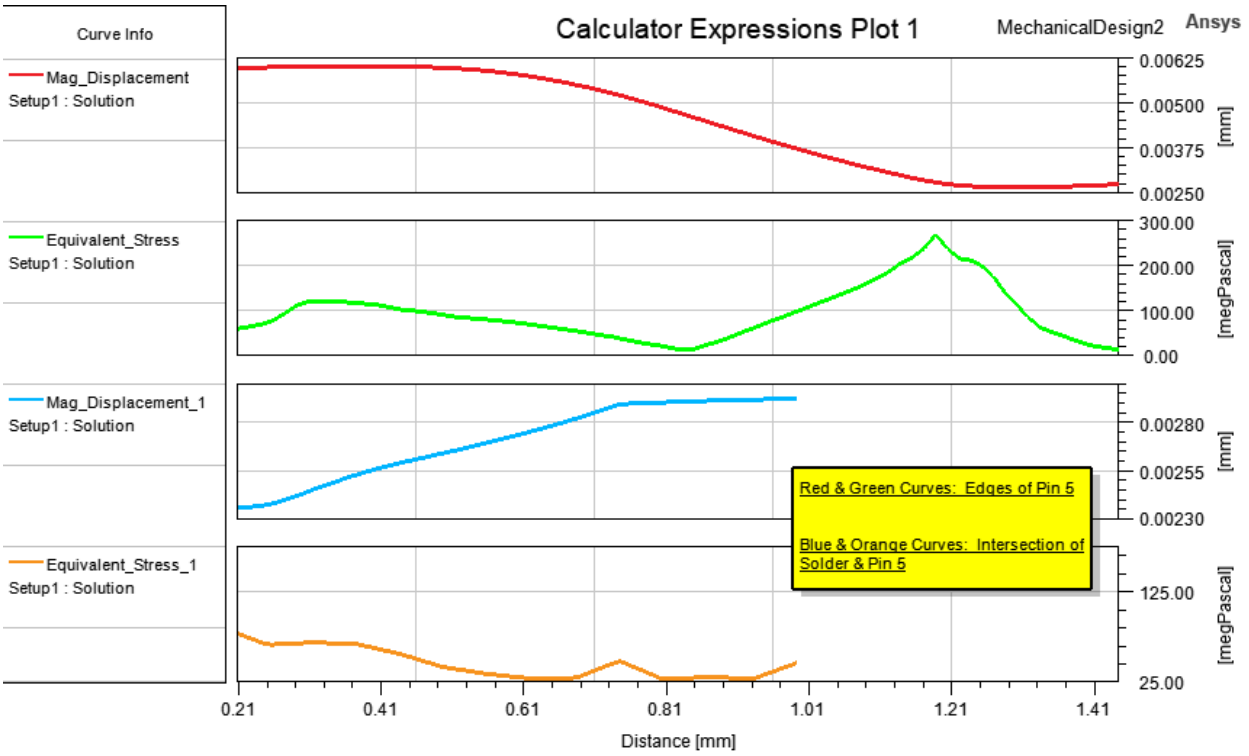


- **Stacked Rectangular Plot:** This type of plot is intended for multiple traces. Unlike the 2D plot previously described, this type puts every individual trace in a separate Cartesian grid stacked vertically. The X scale is shared by all traces, but the Y scale is adjusted to suit each individual trace.

  As with the 2D plot, you must create a polyline object along which the *predefined* structural result values will be calculated and plotted.

  The following image is an example of a four-curve stacked rectangular plot from the same model. The top trace is the *Mag_Displacement* along Polyline1 (the edges of Pin 5), and the second trace is the *Equivalent_Stress* along the same polyline. The third trace is the *Mag_Displacement* along Polyline2 (the intersection of the solder and Pin 5), and the bottom trace is the *Equivalent_Stress* along the same polyline. For all traces, the X axis is the distance along the polylines:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Data Table:** Tabulation of any of the available fields report results as an alternative to a plotted curve.

  The example that follows lists the *Equivalent_Stress* and *Mag_Displacement* results versus the distance along Polyline1 and Polyline2 of the same example model. The results are staggered between columns two and three (Polyline1) and four and five (Polyline2) because the *Distance* calculation points differ between the two lines:

Calculator Expressions Table 1 — MechanicalDesign2 — Ansys

	Distance [mm]	Mag_Displacement [mm] Setup1 : Solution	Equivalent_Stress [megPascal] Setup1 : Solution	Mag_Displacement_1 [mm] Setup1 : Solution	Equivalent_Stress_1 [megPascal] Setup1 : Solution
43	0.260067	0.006011	79.909771		
44	0.268160			0.002388	66.794117
45	0.276321	0.006013	96.590116		
46	0.278092			0.002400	67.182999
47	0.288023			0.002412	67.571882
48	0.292576	0.006017	113.270461		
49	0.297955			0.002425	67.960764
50	0.307887			0.002439	68.349646
51	0.308830	0.006020	120.877602		
52	0.317819			0.002453	68.380102
53	0.325084	0.006023	120.783795		
54	0.327751			0.002466	68.192054
55	0.337683			0.002479	68.004006
56	0.341338	0.006025	120.689988		

Polyline 1 Polyline2

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, you would click and drag the scroll bar on the right vertically to see all 101 data points that were tabulated for each polyline.

Creating a Fields Report

The following is the general procedure for creating any type of Fields Report. It includes drawing a polyline to use as the path along which structural results will be plotted or tabulated. This step is necessary for the predefined stress, strain, temperature, and various material property results. However, you can plot or tabulate *Design Volumes* and *Named Expressions* you create using the *Fields Calculator* (see note below) without creating any polyline geometry.

> **Note:**
>
> The *Fields Calculator* is a powerful and flexible tool for post processing your analysis results. Use it to perform simple or complex mathematical operations on the raw results data. With an appropriate named expression defined, you can generate a report that does not require a polyline as its basis. For example, you could define a face or object list and create a named expression that finds the maximum equivalent stress among the items in the list. Then, in a parametric analysis, you could base the plot's X axis on a design parameter. Finally, base the plot's Y axis on your named expression, thus creating a plot of the maximum equivalent stress among the selected objects or faces versus the swept parameter. There are many other possibilities as well.
>
> You can access the *Fields Calculator* by right-clicking **Field Overlays** in the Project Manager and selecting **Calculator** from the shortcut menu. For more information, search the Ansys Electronics Desktop help for "using the fields calculator" (include the quotation marks to limit search results to instances of the full phrase). Also, refer to the *Related Topics* links at the bottom of the *Using the Fields Calculator* page.

1. Create a polyline path to use as the basis of the table or plot's X axis using *one* of the following two methods:
   - **Draw a polyline manually:**
     a. On the **Draw** ribbon tab, click **Draw line**.
     b. If prompted to create a non-model object (because a solution already exists), click **Yes**.
     c. Draw a line or series of lines along which you wish to determine the structural results. The line segments should pass through or along the surfaces of the bodies comprising the model. Use one of the following three drawing methods:
        - Specify the line start and end points graphically (by clicking in the Modeler window). You can snap to model vertices (endpoints),

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

midpoints or edges, grid points, and more.

- Specify the points by typing the coordinates into the **X, Y,** and **Z** text boxes in the status bar.
- Draw an arbitrary line and edit it's properties afterward (in the docked *Properties* window or the *Properties* dialog box).

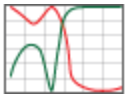
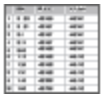
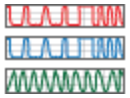
  You can switch between straight and curved segments by right-clicking in the Modeler window and choosing the desired **Set Edge Type** option.

d. After clicking the last endpoint, right-click in the Modeler window and choose **Done** to terminate the *Draw Line* command.

   Notice the name of the line object that has been added to the History Tree. The default name for the first line object is *Polyline1*, and you can change the name in the docked *Properties* window if desired.

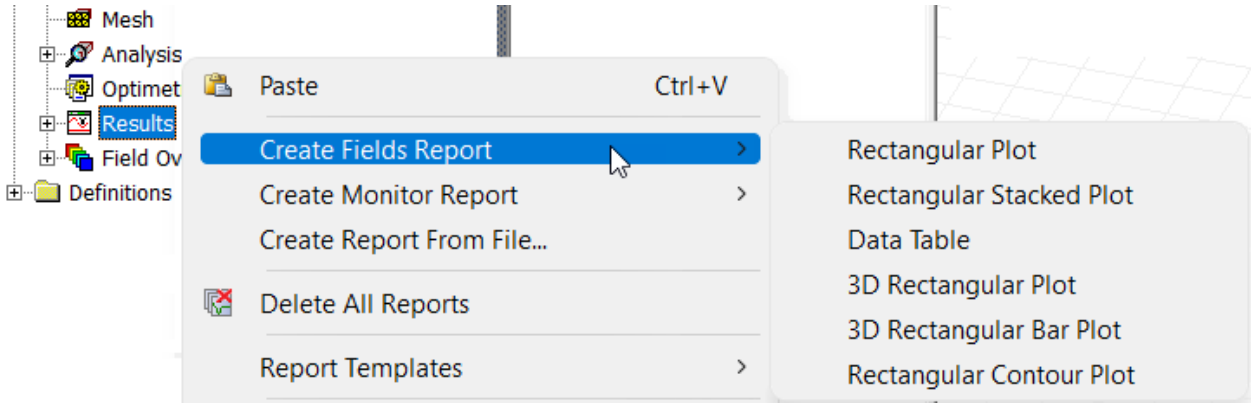
- **Convert model edges to a polyline:**
  a. In *Edge* selection mode, select a model edge or a contiguous series of model edges to select them.
  b. On the **Draw** ribbon tab, click **Edge** > **Create Object From Edge**.
  c. While the objects are still selected, click **Unite** on the **Draw** ribbon tab.

     Notice the name of the united object that has been added to the History Tree. You can change the name in the docked *Properties* window if desired.

2. Access the *Report* dialog box using one of the following three methods:
   - On the **Results** ribbon tab, click **Fields Report** and choose the ***desired display type*** (such as,  **2D**,  **Data Table**, or  **Stacked**).
   - Using the menu bar, click **Mechanical** > **Results** > **Create Fields Report** > ***desired display type***.
   - Right-click **Results** in the Project Manager and choose **Create Fields Report** >

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

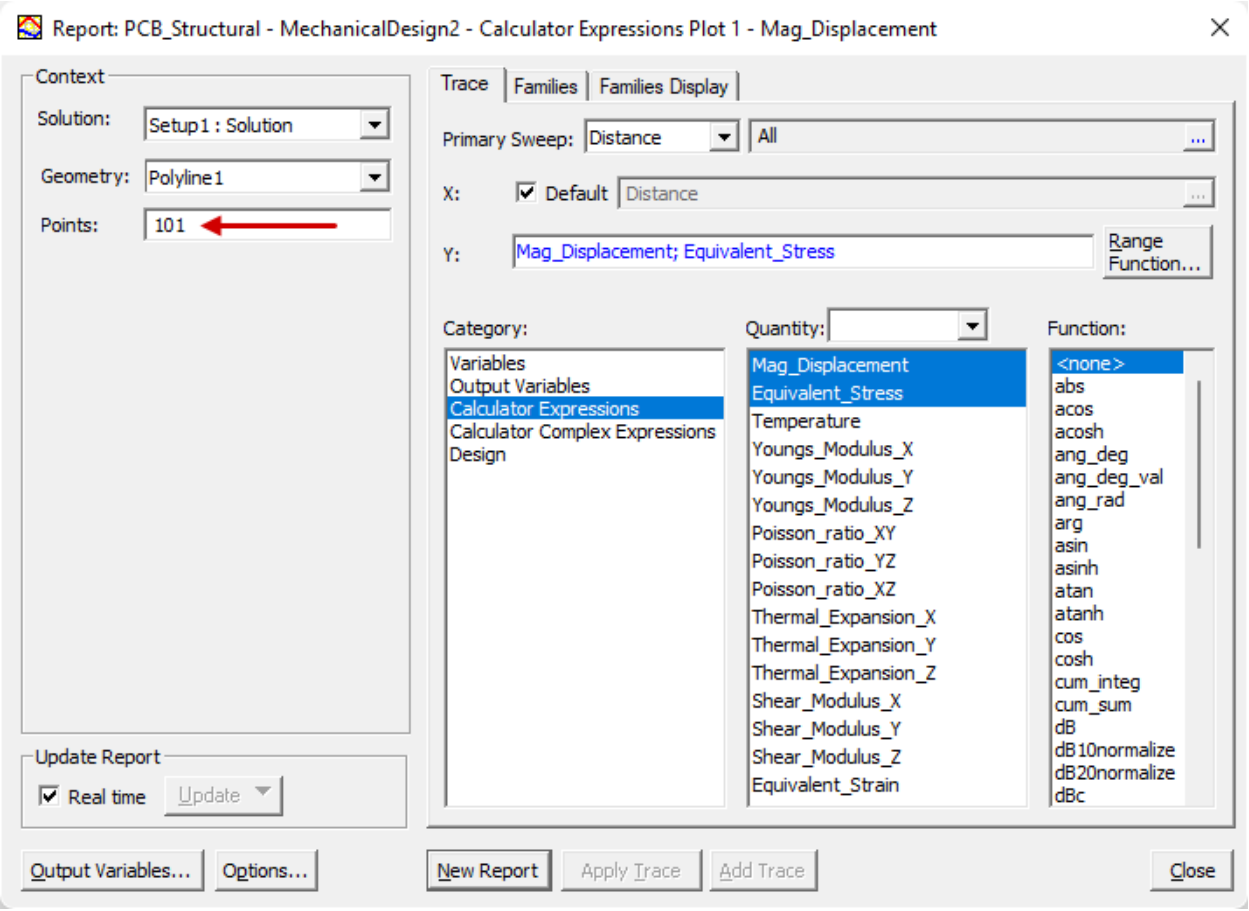
desired display type:



3. Specify the following settings in the *Report* dialog box:
   a. Choose the desired solution setup from the **Solution** drop-down menu (if you solved more than one setup).
   b. Choose the polyline you created previously from the **Geometry** drop-down menu.
   c. Optionally, specify a different number of **Points** to calculate along the specified polyline. (The default number of points is 101, which results in one hundred trace segments between the first and last data points.)
   d. In the **Category** list, select **Calculator Expressions**, if it is not already selected.
   e. Select the desired **Quantity** or **Quantities** to tabulate or plot. (You do not need to specify a *Function* for structural results.)

   The following report setup example shows the settings to tabulate or plot *Mag_Displacement* and *Equivalent_Stress* results along *Polyline1*:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, click **Range Function** to apply a mathematical operation to the results. Then:
    a. In the *Set Range Function* dialog box, choose the desired operation from the **Function** drop-down menu (such as *sum, mean, max* or *variance*).
    b. Click **OK** to return to the *Report* dialog box.
5. Click **New Report**.

    The plot or data table appears in a new window.

6. Optionally, if you created a plot, you can double-click inside the legend box to access the *Properties* dialog box and customize many plot parameters, such as:
    - Trace colors, style, and width
    - Enable data point symbols and change their color, fill, style, and frequency
    - Grid options
    - Headers
    - X and Y axis parameters and scaling
7. Optionally, if you created a data table, you can customize it in the following ways:

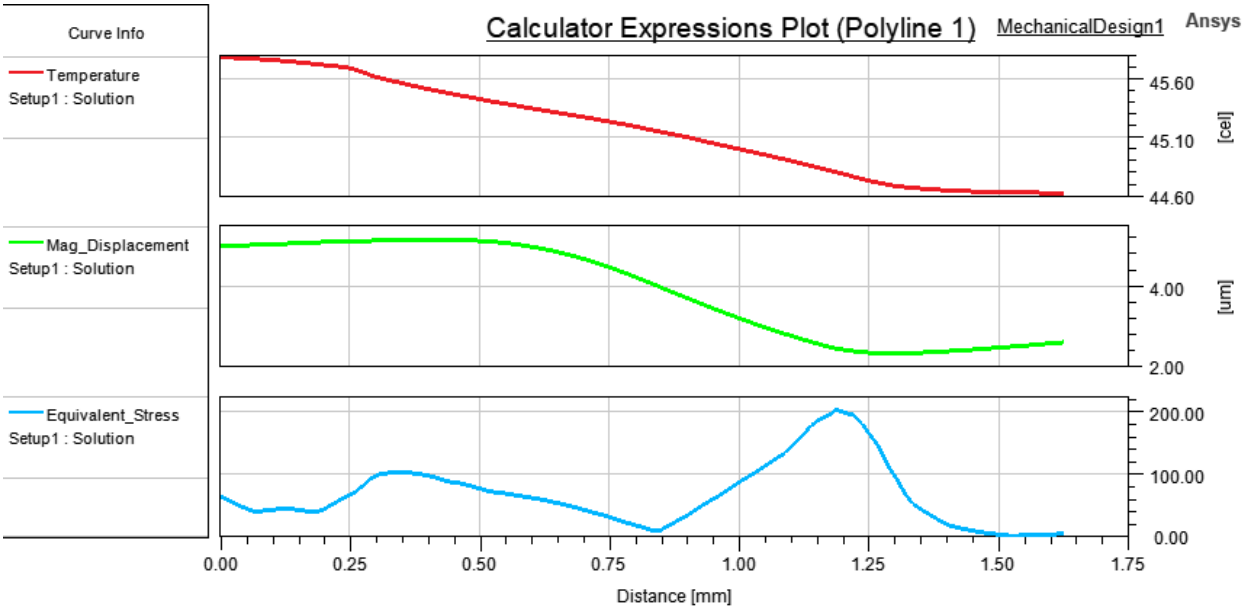
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Drag the column borders to resize the columns.
- Click on a column heading and then, in the docked *Properties* window, adjust the **Units, Field Width,** or **Field Precision** values.

8. Optionally, select parameters for one or more additional results to add to the current plot or table and click **Add Trace**.
9. Click **Close** to exit the *Report* dialog box.

Examples:

The following example is a stacked rectangular plot of the *Temperature, Mag_Displacement,* and *Equivalent_Stress* results along *Polyline1* (based on the example model shown in the structural Fields Report topic):

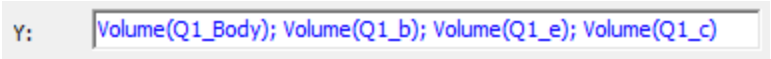


The following example is a data table listing the volume of the four objects comprising transistor Q1. It has been modified to show the volumes in cubic millimeters, and the columns have been reduced from their default widths:

Design Table 1 MechanicalDesign2 Ansys

	X [mm]	Volume(Q1_Body) [mm3] Setup1 : Solution	Volume(Q1_b) [mm3] Setup1 : Solution	Volume(Q1_e) [mm3] Setup1 : Solution	Volume(Q1_c) [mm3] Setup1 : Solution
1	0.000000	1.135941	0.126216	0.126216	0.126216

The contents of the **Y** text box in the *Report* dialog box used to produce the preceding table were as follows:



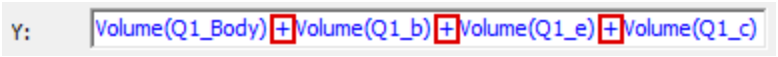
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, an additional column has been added to *Design Table 1*. It shows the total volume of the four objects included in the original table:

Design Table 1 MechanicalDesign2 Ansys

	X [mm]	Volume(Q1_Body) [mm3] Setup1 : Solution	Volume(Q1_b) [mm3] Setup1 : Solution	Volume(Q1_e) [mm3] Setup1 : Solution	Volume(Q1_c) [mm3] Setup1 : Solution	Volume(Q1_Body)+Volume(Q1_b)+Volume(Q1_e)+Volume(Q1_c) [mm3] Setup1 : Solution
1	0.000000	1.135941	0.126216	0.126216	0.126216	1.514589

With the **Volume** of all four parts selected in the **Quantity** list of the *Report* dialog box, the semicolons in the **Y** text box were replaced with plus signs (+):



to produce the total volume column (far right).

Field Overlay and Animation

In structural analyses, field overlays provide a visual indication of the assigned temperatures, equivalent stress or strain results, displacement results, and, in the case of layout components, various material properties. Each of these sets of values can be overlaid on selected objects or faces. Displacements can be overlaid as a vector plot or as color contours of the magnitude. All field overlays are displayed over deformed geometry, based on the shape after thermal expansion or contraction. By default, the deformation is scaled (typically greatly exaggerated) for easy visibility.

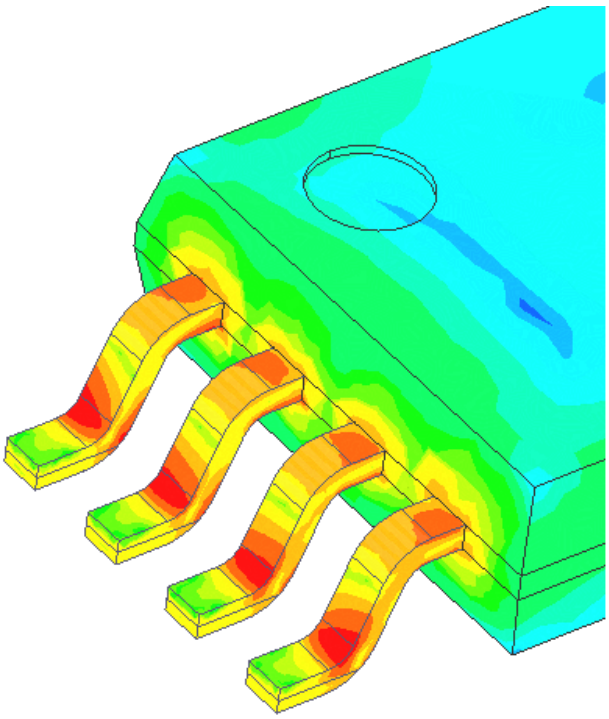
The available *predefined* structural field overlay results are listed and described under Field Overlay in the *Results (Structural)* topic. Examples of 2D plots, Stacked plots, and Data Tables follow:

To Produce a Field Overlay Plot and Animate it:

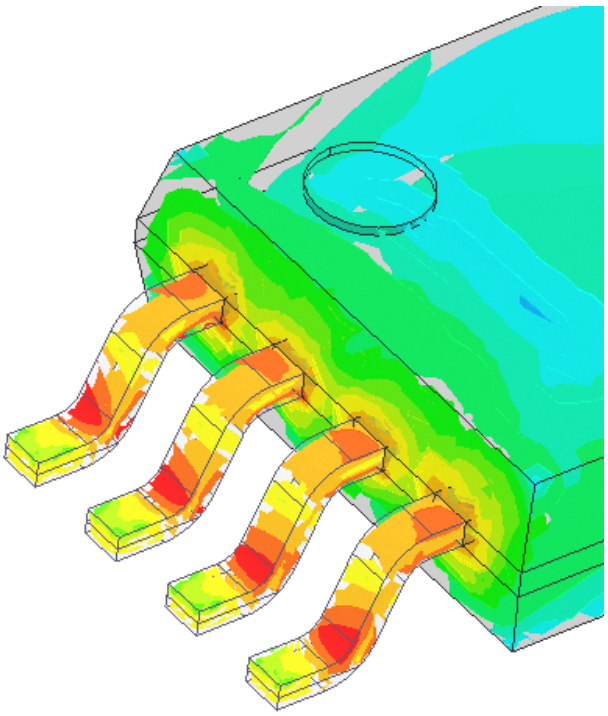
1. Select the *Faces* or *Objects* on which you want to overlay structural results.
2. Access the *Create Field Plot* dialog box using *one* of the following three methods:
   - Using the menu bar, click **Mechanical** > **Fields** > **Plot Fields** > ***desired result***.
   - Right-click **Field Overlays** in the Project Manager and choose **Plot Fields** > ***desired result*** from the shortcut menu.
   - Right-click in the *Modeler* window and choose **Plot Fields** > ***desired result***.
3. If you selected *Named Expression* in the previous step, click the desired named expression in the *Selecting calculated expression* dialog box that appears. Then, click **OK**.
4. In the *Create Field Plot dialog* box that appears, select the desired setup from the **Solution** drop-down menu (if you solved more than one analysis setup).
5. Optionally, if solid objects were selected for the overlay (as opposed to faces), you can clear the **Plot on surface only** option to view results within the interior of the solid objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When this option is selected, the results are rendered on the model surfaces only. If you clear this option, the results are rendered for the full model volume, but they are rendered as IsoValSurfaces. Rather than the full volume of the model being rendered in varying colors, IsoValSurfaces connect points of the same resultant value and are rendered for each scale division specified in the plot settings (15 values by default). This plot style enables you to see the results on the interior of solids without creating clipping planes. The following equivalent stress images compare the two plot styles:



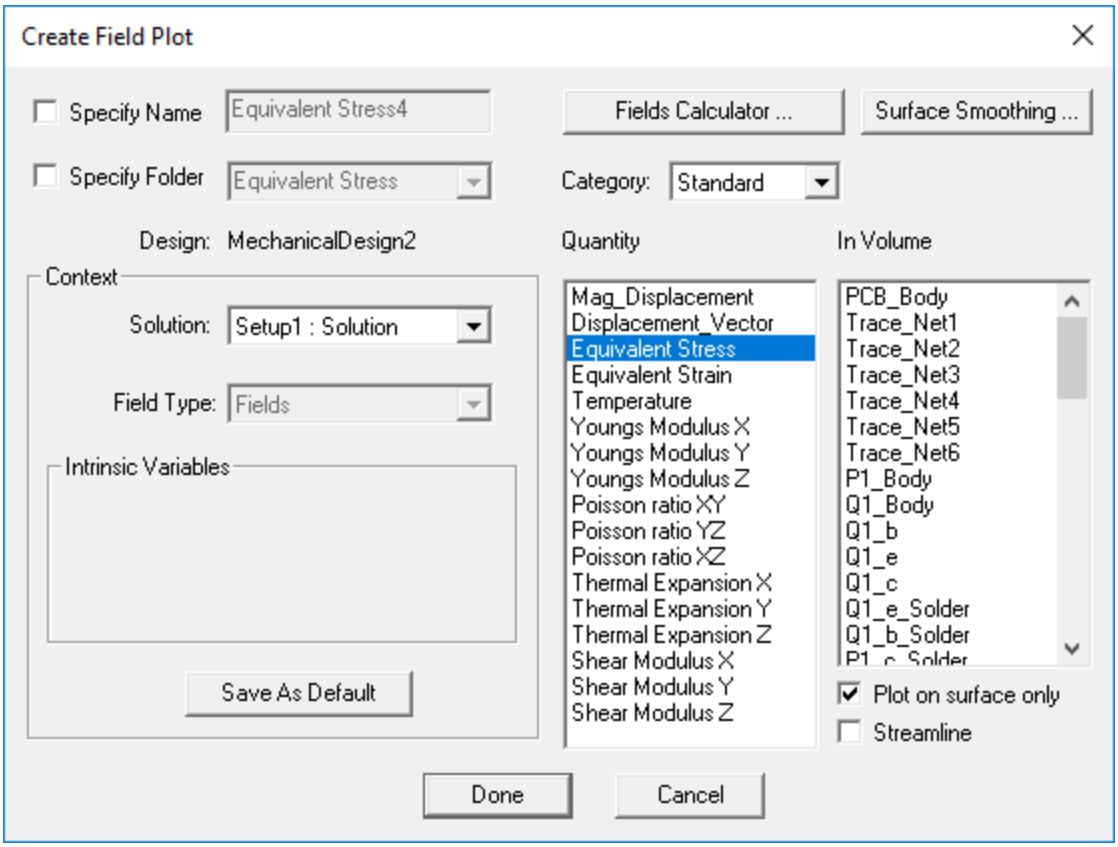
Plot on surface only
option selected



Plot on surface only
option **not** selected

The following field plot setup example overlays the equivalent stresses on the surfaces of selected objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



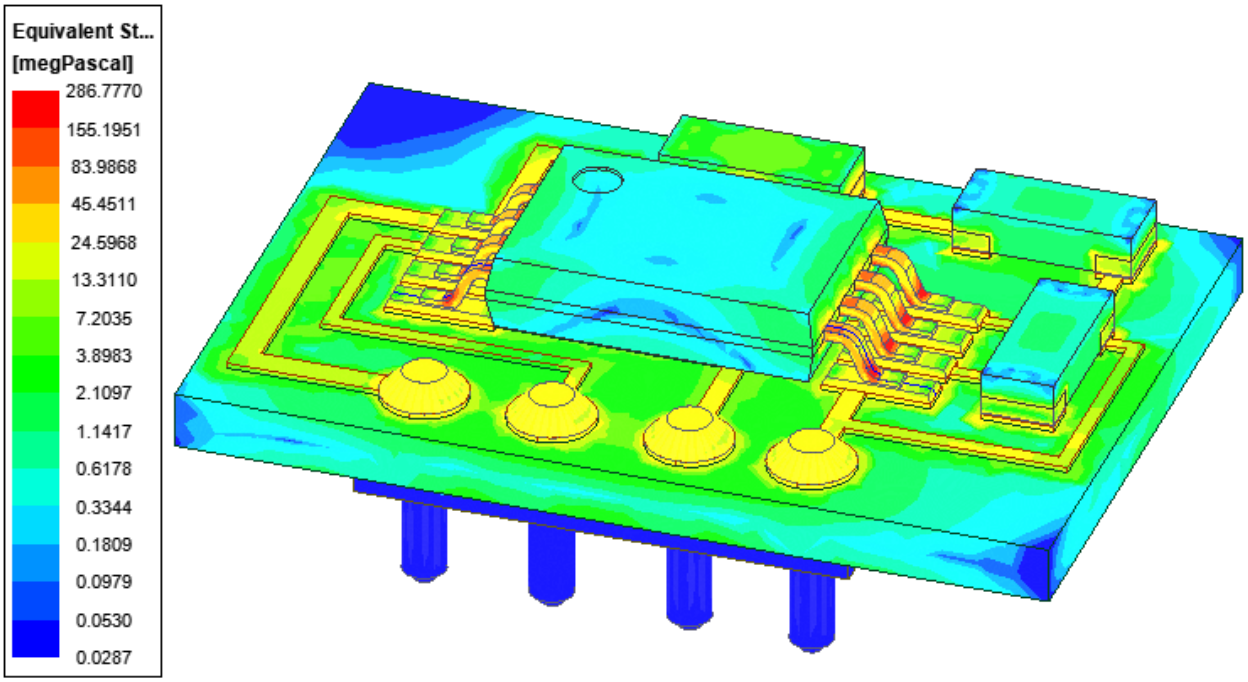
6. Click **Done**.

   The overlay and result legend appear on the model in the *Modeler* window.

7. Optionally, double-click the plot legend to access and modify the plot properties (scale, color map, and so on).

   The following image is an *Equivalent_Stress* overlay. The legend's units (megPascal) have been customized, the scale set to logarithmic, and the **Scale Deformation** option cleared to show the undeformed shape:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. You can animate structural overlays using the **NormalizedDeformation** option for the swept variable. The color contours and deformed shape in the animation vary linearly from zero to the calculated results. Also, if your solution has multiple sets of field results (such as when you run a parametric analysis to vary a design parameter), you can animate your structural field results to show the way the solution changes for each variant.
    a. Under *Field Overlays* in the Project Manager, right-click ***your field overlay*** and choose **Animate** from the shortcut menu.

       The *Create Animation Setup* dialog box appears.

    b. Optionally, change the **Name** and/or type a **Description** for the animation setup.
    c. Using the two drop-down menus in the *Swept Variable(s)* tab, choose the setup or variable you want to use as the basis of the animation.

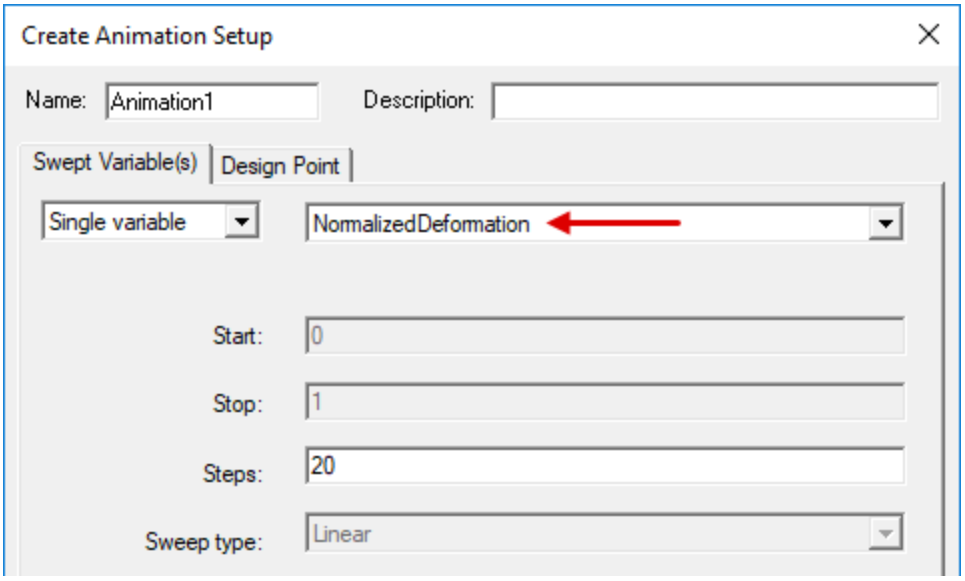
       For example:

       - If you want to linearly animate the results for a normalized deformation multiplier of 0 through 1, choose **Single Variable** and the **NormalizedDeformation** option.
       - If you want to base the animation on a single design variable, choose **Single Variable** and the ***variable name***.
       - If you want to base the animation on a parametric setup, choose ***Parametric Setup*** and the ***parametric setup name***.

    d. Select the variations you want to include in the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following example shows a linear animation setup using the *NormalizedDeformation* option and 20 *Steps*:



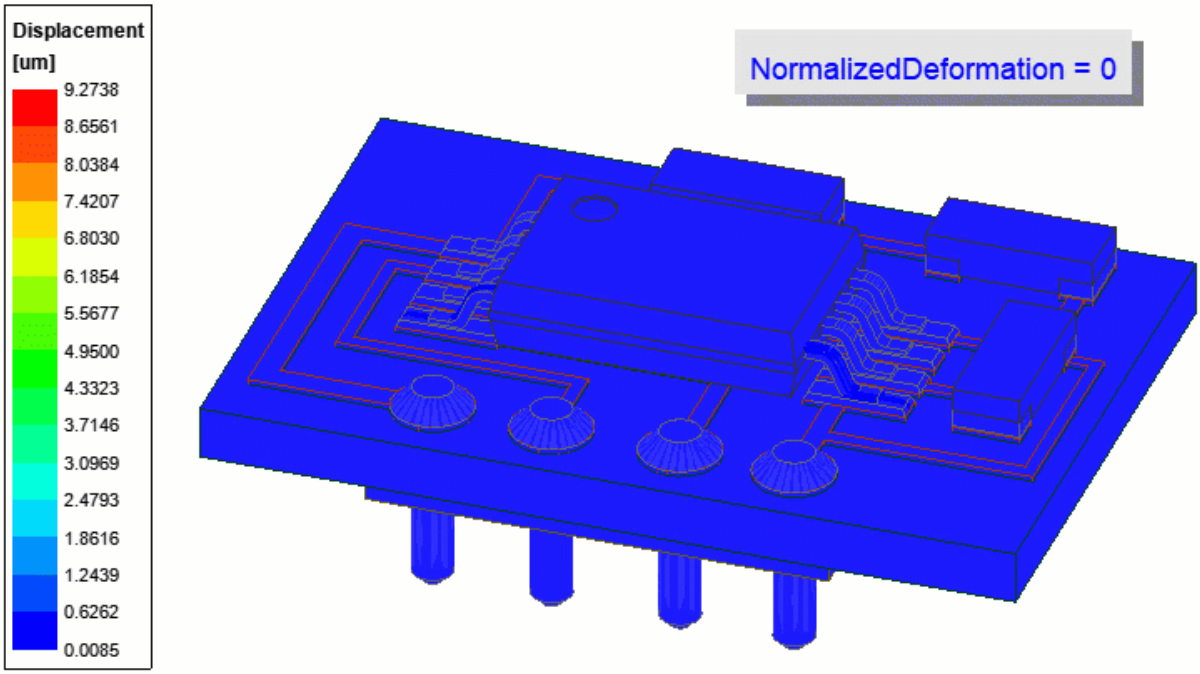
e. Click **OK**.

   The *Animation* dialog box appears, and the overlaid results are animated after the frames are computed.

f. Use the animation controls to stop, restart, reverse, increase/decrease the speed, or to go to a particular animation frame.

   An animation of the Mag_Displacement results, based on the preceding *NormalizedDeformation* setup example, is shown below. The displacement overlay attributes have been changed as follows: The deformation scale has been set to 150 times the actual deformed shape, and micrometers (um) has been selected as the legend units.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Fields Summary

A fields summary computes basic or derived field quantities and displays the data in a table in the **Fields Summary** dialog box. Currently, all results are reported in SI units (m, Pa, N, °C).

The displacement, stress, strain, temperature, and various mapped material properties that are available for plotting in reports and fields overlays are also available in the *Fields Summary* dialog box. Additionally, the following derived quantity is only available in the *Fields Summary* dialog box:

- **Reaction Force:** This derived quantity is applicable to *Support* boundaries (*Fixed, Frictionless,* or *Cylindrical*). It lists the three components of the *Total* reaction force at the boundary, corresponding to the global coordinate system axis directions. Individual nodal results are summed to determine the total force components.

For most of the available summary quantities, the minimum (*Min*), maximum (*Max*), *Mean*, and standard deviation (*StDev*) values are reported along with the Area/Volume (of specified boundaries/faces or solid objects, respectively). Additionally, *Normal* unit-vector components are reported for flat boundaries/faces. When multiple faces are assigned a single boundary or belong to a single face list, they must all be parallel and have the same vector sign for the normal direction to be reported. *Min, Max, Mean,* and *StDev* are not applicable to reaction forces. *Total* values are not applicable to other quantities.

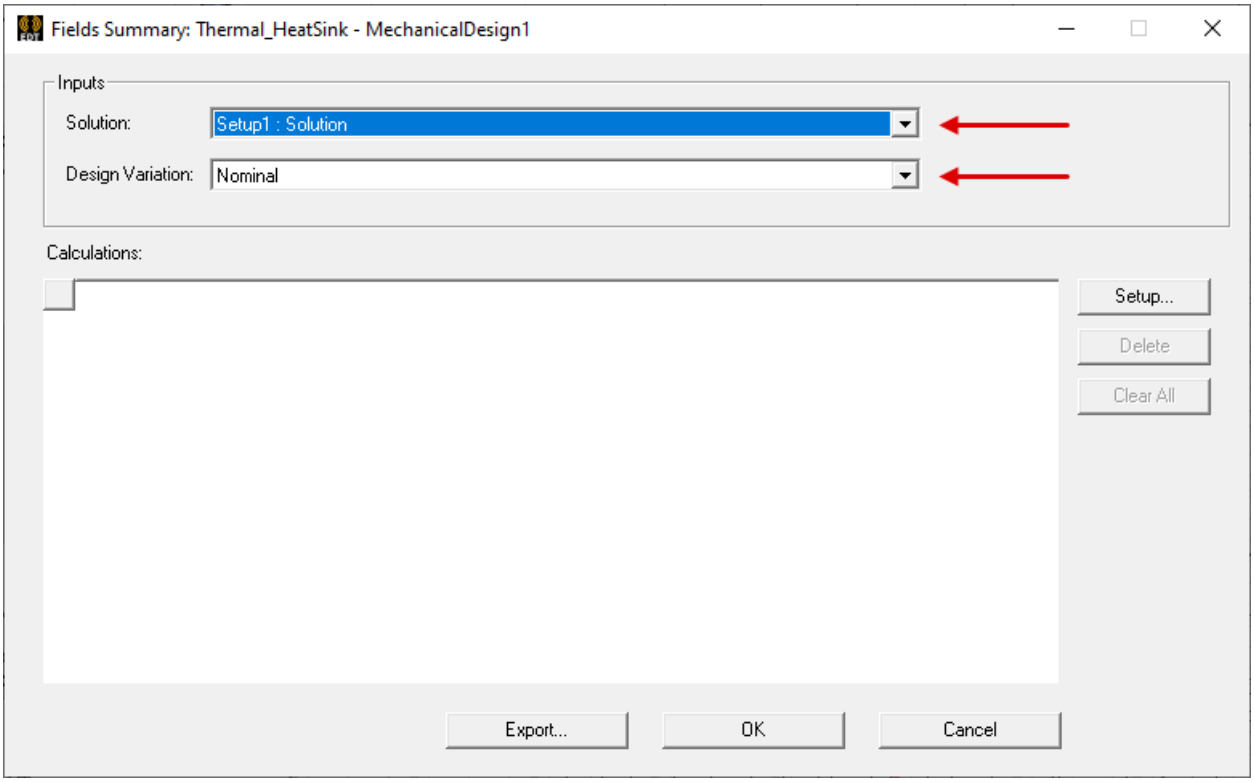
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fields Summary Procedure

1. Use one of the following two methods to begin creating a fields summary:
   - From the **Mechanical** menu, select **Fields** > **Create Fields Summary**.
   - In the Project Manager, right-click **Field Overlays** and select **Create Fields Summary** from the shortcut menu.

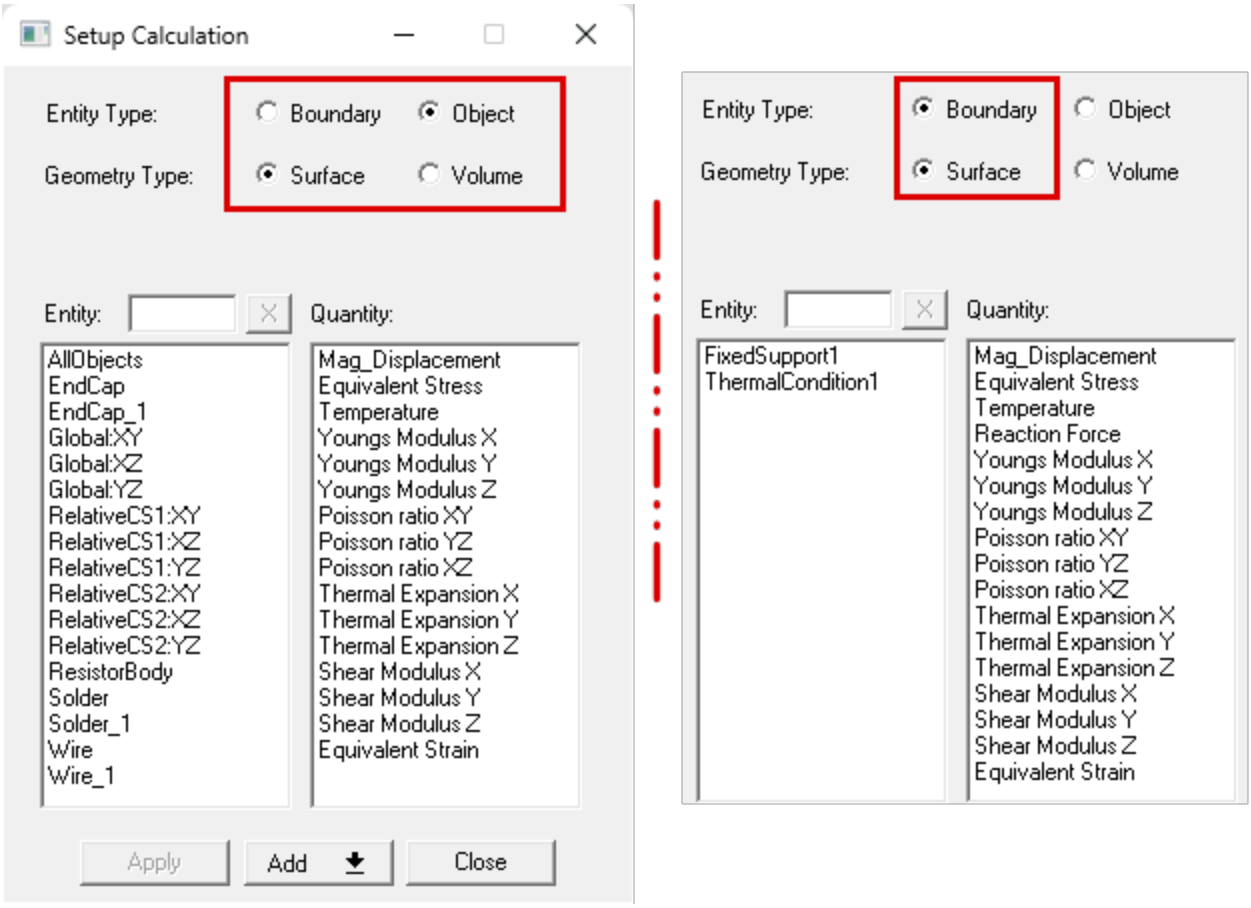
   The *Fields Summary* and *Setup Calculation* dialog boxes are displayed.

2. Under *Inputs* in the *Fields Summary* dialog box, select the **Solution** and **Design Variation** for which you want to create a fields summary.



3. In the *Setup Calculation* dialog box, select the **Entity Type** and **Geometry Type**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Entity Type:** Select **Boundary** (structural boundary or excitation) or **Object** (geometry)
- **Geometry Type:** Select **Volume** or **Surface**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When **Boundary** and **Volume** are selected:

- Only structural excitations that are assigned to a solid object are available under **Entities** (Thermal Condition, Non-Uniform Force).

When **Boundary** and **Surface** are selected:

- Only structural boundaries and excitations that are assigned to faces are available under **Entities** (all supports, Displacement, Uniform Force, Pressure).

When **Object** and **Surface** are selected, the following **Entities** are available:

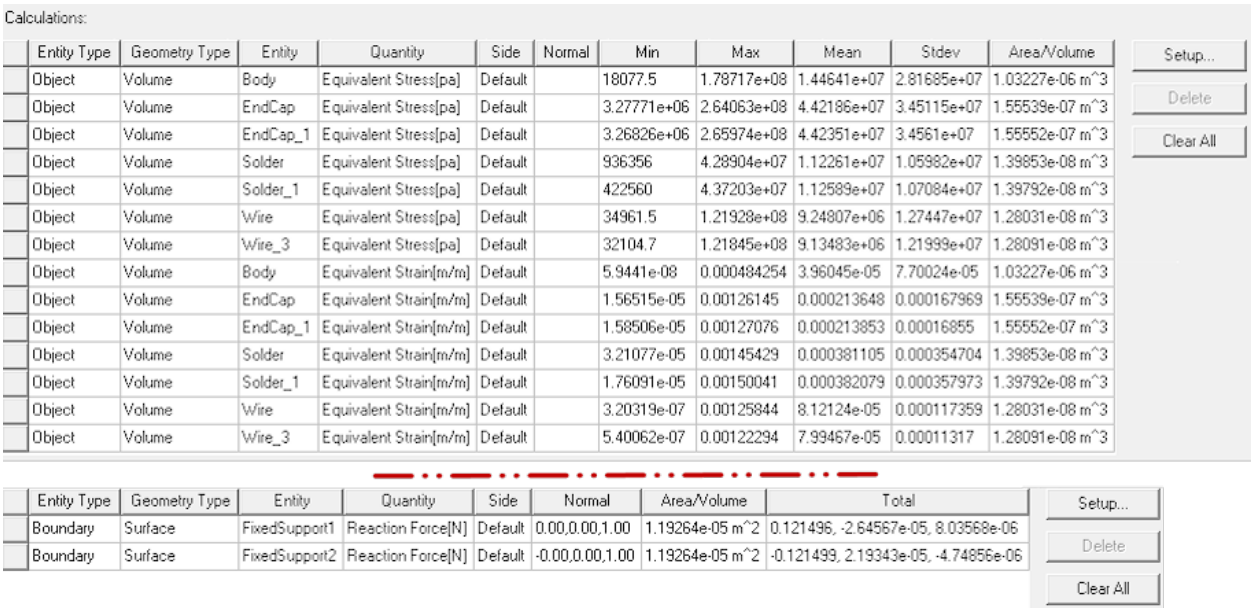
- All solid objects (selecting a solid object is equivalent to selecting all of its faces)
- Global and relative coordinate system planes (XY, YZ, and XZ)
- Face lists

4. Select the **Entities** and **Quantities** you want to display in the fields summary, as follows:
   a. Depending on your choices in step 3, complete one of the following three actions to define the **Entity**:
      - **Boundary:** Select one or more boundaries or excitations.
      - **Object** and **Volume:** Select one or more solid geometry objects.
      - **Object** and **Surface:** Select one or more solid or shell objects, planes, or face lists.
   b. Then, choose the **Quantity** to summarize (*Mag_Displacement, Equivalent Stress, Temperature*, and/or *Reaction Force*).

      Available quantities depend on your selections in step 3.

5. From the **Add** drop-down menu, select **Add As Single Calculation** or **Add As Multiple Calculations** to add rows for data.
   - **Add As Single Calculation** adds one row displaying the quantity for all selected entities collectively.
   - **Add As Multiple Calculations** adds a separate row of the selected quantity for each selected entity.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Calculations:

Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev	Area/Volume
Object	Volume	Body	Equivalent Stress[pa]	Default		18077.5	1.78717e+08	1.44641e+07	2.81685e+07	1.03227e-06 m^3
Object	Volume	EndCap	Equivalent Stress[pa]	Default		3.27771e+06	2.64063e+08	4.42186e+07	3.45115e+07	1.55539e-07 m^3
Object	Volume	EndCap_1	Equivalent Stress[pa]	Default		3.26826e+06	2.65974e+08	4.42351e+07	3.4561e+07	1.55552e-07 m^3
Object	Volume	Solder	Equivalent Stress[pa]	Default		936356	4.28904e+07	1.12261e+07	1.05982e+07	1.39853e-08 m^3
Object	Volume	Solder_1	Equivalent Stress[pa]	Default		422560	4.37203e+07	1.12589e+07	1.07084e+07	1.39792e-08 m^3
Object	Volume	Wire	Equivalent Stress[pa]	Default		34961.5	1.21928e+08	9.24807e+06	1.27447e+07	1.28031e-08 m^3
Object	Volume	Wire_3	Equivalent Stress[pa]	Default		32104.7	1.21845e+08	9.13483e+06	1.21999e+07	1.28091e-08 m^3
Object	Volume	Body	Equivalent Strain[m/m]	Default		5.9441e-08	0.000484254	3.96045e-05	7.70024e-05	1.03227e-06 m^3
Object	Volume	EndCap	Equivalent Strain[m/m]	Default		1.56515e-05	0.00126145	0.000213648	0.000167969	1.55539e-07 m^3
Object	Volume	EndCap_1	Equivalent Strain[m/m]	Default		1.58506e-05	0.00127076	0.000213853	0.00016855	1.55552e-07 m^3
Object	Volume	Solder	Equivalent Strain[m/m]	Default		3.21077e-05	0.00145429	0.000381105	0.000354704	1.39853e-08 m^3
Object	Volume	Solder_1	Equivalent Strain[m/m]	Default		1.76091e-05	0.00150041	0.000382079	0.000357973	1.39792e-08 m^3
Object	Volume	Wire	Equivalent Strain[m/m]	Default		3.20319e-07	0.00125844	8.12124e-05	0.000117359	1.28031e-08 m^3
Object	Volume	Wire_3	Equivalent Strain[m/m]	Default		5.40062e-07	0.00122294	7.99467e-05	0.00011317	1.28091e-08 m^3

Setup... Delete Clear All

Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Area/Volume	Total
Boundary	Surface	FixedSupport1	Reaction Force[N]	Default	0.00,0.00,1.00	1.19264e-05 m^2	0.121496, -2.64567e-05, 8.03568e-06
Boundary	Surface	FixedSupport2	Reaction Force[N]	Default	-0.00,0.00,1.00	1.19264e-05 m^2	-0.121499, 2.19343e-05, -4.74856e-06

Setup... Delete Clear All

> **Note:**
>
> You can double-click a row in the fields report to open the **Setup Calculation** dialog box. You can then edit the row's definition and click **Apply** to change it.

6. Optionally, click **Export** (or **Apply and Export**, if the settings haven't been saved yet) to open the **Save As** dialog box. Then, save a comma-separated values (.csv) file of the fields summary data to the desired folder on your computer or network.

   The .csv file looks like the following image when opened in a spreadsheet program, such as Excel (with column widths and text alignment adjusted):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B	C	D	E	F	G	H	I	J	K
1	SolutionName	Setup1 : Solution									
2	DesignVariation	Nominal									
3											
4											
5	Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev	Area/Volume
6	Object	Volume	Body	Equivalent Stress[pa]	Default		18077.5	1.79E+08	1.45E+07	2.82E+07	1.03227e-06 m^3
7	Object	Volume	EndCap	Equivalent Stress[pa]	Default		3.28E+06	2.64E+08	4.42E+07	3.45E+07	1.55539e-07 m^3
8	Object	Volume	EndCap_1	Equivalent Stress[pa]	Default		3.27E+06	2.66E+08	4.42E+07	3.46E+07	1.55552e-07 m^3
9	Object	Volume	Solder	Equivalent Stress[pa]	Default		936356	4.29E+07	1.12E+07	1.06E+07	1.39853e-08 m^3
10	Object	Volume	Solder_1	Equivalent Stress[pa]	Default		422560	4.37E+07	1.13E+07	1.07E+07	1.39792e-08 m^3
11	Object	Volume	Wire	Equivalent Stress[pa]	Default		34961.5	1.22E+08	9.25E+06	1.27E+07	1.28031e-08 m^3
12	Object	Volume	Wire_3	Equivalent Stress[pa]	Default		32104.7	1.22E+08	9.13E+06	1.22E+07	1.28091e-08 m^3
13	Object	Volume	Body	Equivalent Strain[m/m]	Default		5.94E-08	0.00048425	3.96E-05	7.70E-05	1.03227e-06 m^3
14	Object	Volume	EndCap	Equivalent Strain[m/m]	Default		1.57E-05	0.00126145	0.0002136	0.000168	1.55539e-07 m^3
15	Object	Volume	EndCap_1	Equivalent Strain[m/m]	Default		1.59E-05	0.00127076	0.0002139	0.0001686	1.55552e-07 m^3
16	Object	Volume	Solder	Equivalent Strain[m/m]	Default		3.21E-05	0.00145429	0.0003811	0.0003547	1.39853e-08 m^3
17	Object	Volume	Solder_1	Equivalent Strain[m/m]	Default		1.76E-05	0.00150041	0.0003821	0.000358	1.39792e-08 m^3
18	Object	Volume	Wire	Equivalent Strain[m/m]	Default		3.20E-07	0.00125844	8.12E-05	0.0001174	1.28031e-08 m^3
19	Object	Volume	Wire_3	Equivalent Strain[m/m]	Default		5.40E-07	0.00122294	7.99E-05	0.0001132	1.28091e-08 m^3

5	Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Area/Volume	Total
6	Boundary	Surface	FixedSupport1	Reaction Force[N]	Default	0.00 0.00 1.00	1.19264e-05 m^2	0.121496 -2.64555e-05 8.09346e-06
7	Boundary	Surface	FixedSupport2	Reaction Force[N]	Default	-0.00 0.00 1.00	1.19264e-05 m^2	-0.121499 2.19402e-05 -4.74797e-06

7. Choose one of the following two actions:
   - Click **OK** to save the settings and close the *Fields Summary* dialog box. When you open the dialog box again, the data for the saved settings is displayed.
   - Click **Cancel** to close the **Fields Summary** dialog box without saving the settings or summary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - Mechanical Boundaries and Excitations

This section includes information about the following items:

- Available Mechanical Boundaries and how each is assigned
- Available Mechanical Excitations and how each is assigned
- Editing Boundaries, Excitations, and Other Assignments after they've been created
- Assigning a Boundary or Excitation to a Portion of a Face

For Mechanical designs, the available boundaries and excitations depend on the *Solution Type* (Model, Thermal, or Structural). Details are provided in the subtopics that follow.

Boundaries

For Mechanical designs, the available boundaries depend on the *Solution Type* (Model, Thermal, or Structural). The available boundaries for each solution type are as listed below:

- **Modal Solutions:**
  - Fixed Support
  - Frictionless Support
  - Cylindrical Support
- **Thermal Solutions:**
  - Convection
  - Temperature
  - Rotating Fluid
- **Structural Solutions:**
  - Fixed Support
  - Frictionless Support
  - Cylindrical Support

Convection

Use a Convection boundary to simulate the transfer of heat between object faces and the surrounding environment.

You can assign convection boundaries to the following entities:

- **Faces** of solid or shell objects
- **Objects** (solid or shell): Assigning convection to an object is equivalent to assigning convection to all faces of the object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The convective heat transfer is calculated based on the difference between the surface temperature at the convection faces and the specified ambient temperature, and on the convection *film coefficient* (also referred to as the *heat transfer coefficient*). The following film coefficient options are available:

- A uniform, time-dependent, or temperature-dependent film coefficient that you specify.
  - For both steady-state and transient thermal solutions: You can specify a temperature-dependent film coefficient. To do so, create a piecewise linear (pwl) dataset to define the coefficient versus temperature values.
  - For transient solutions: You can specify a time-dependent uniform film coefficient. Do so by creating a piecewise linear (pwl or pwl_periodic) dataset or a piecewise constant (pwc) dataset to define coefficient versus time values.
- Non-uniform film coefficients that are imported from a linked Icepak design. The Icepak solution takes into account the gas or liquid surrounding the convection faces, the orientation of the faces, and the natural or forced convective flow of the fluid.

The ambient temperature can be defined globally (as a design variable) or specified individually for each convection boundary you assign. Additionally, for transient solutions, you can vary the ambient temperature with time by creating a piecewise linear (pwl or pwl_periodic) dataset or a piecewise constant (pwc) dataset to define the time-dependent temperature values. Please see the Ambient Temperature Guidelines topic for important considerations in defining this value. Note that heat can flow in either direction, from the object to the ambient environment or from the ambient environment to the object, depending on which is hotter.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Assign a *Convection* Boundary:

1. Select one or more objects or faces.

Important:

- It is often inappropriate to assign convection to an entire object. The reason is that internal faces and those in contact with adjacent objects should not transfer heat via convection; they are not exposed to the ambient environment. Similarly, faces to which other thermal boundaries or excitations are applied (such as Heat Flux) are often not exposed to the ambient environment. The thermal boundary or excitation may represent an adjacent object that is not actually included in the model geometry. In such cases, assign convection to the appropriate individual faces rather than the object.
- You cannot assign a convection boundary to the same face to which thermal contact has been assigned nor to a face that fully or partially overlaps an adjacent thermal contact face. An error message will be produced if you attempt to do so. When a face that is exposed to the ambient environment is in contact with an adjacent part, and contact is assigned where they meet to account for thermal resistance, imprint the smaller part onto the larger one. This operation splits the portion of the face where the parts overlap from the rest of the face. An example is the semiconductor mounting face of a heat sink. Imprint the semiconductor onto the heat sink, which creates two faces from the original one. Then, assign convection only to the exposed portion of the mounting face. (Also, see Assigning a Boundary or Excitation to a Portion of a Face.)
- In certain cases, it is acceptable to assign convection to a body or face that contacts one or more adjacent parts. Specifically, if all the involved objects are in default bonded contact (without thermal resistance), the portions of faces where the objects overlap each other will automatically be excluded from any convection boundaries. In such cases, you could assign convection to an object (such as a circuit board) and the areas where all of the adjacent components (such as capacitors, resistors, and semiconductors) that are bonded to the board will automatically be filtered out of the convection boundary. However, be aware that this automatic filtering of face-overlap areas does not occur for user-defined thermal contact assignments (see previous bullet). It only works for touching objects for which you have **not** assigned contact to account for thermal resistance.
- For single-face shell objects, assigning convection to the object is equivalent to assigning convection to the face.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

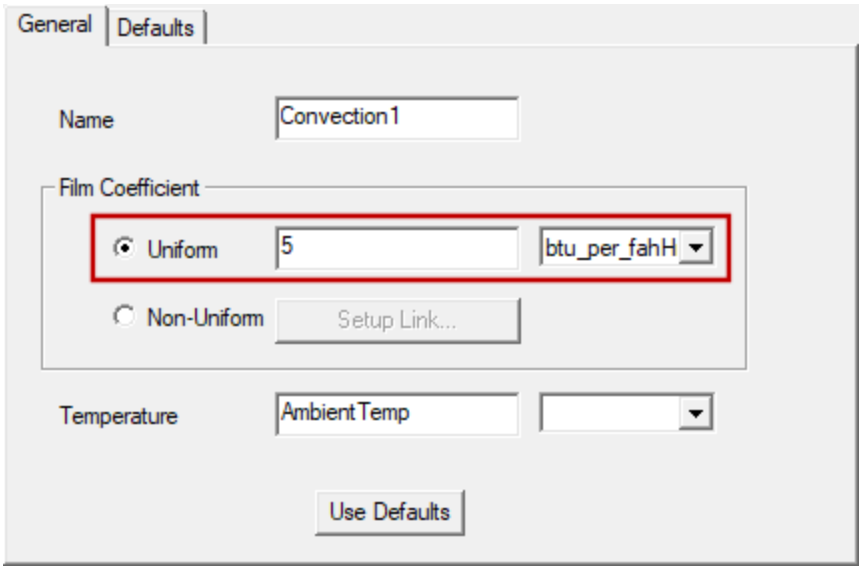
2. Access the *Convection* dialog box using one of the following three methods:
   - Using the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Convection**.
   - Right-click **Boundary** in the Project Manager and choose **Assign** > **Convection** from the shortcut menu.
   - Right-click in the Modeler window, or right-click a select object in the History Tree, and choose **Assign Boundary** > **Convection**.
3. Select one of the following *Film Coefficient* types:
   - **Uniform**: A consistent film coefficient definition will be used for all selected faces. The value can be numeric, a variable, or an expression (including constants or variables and operators), or it can be temperature-dependent (using a piecewise linear dataset reference). Additionally, for transient thermal solutions, the value can be time-dependent, using a piecewise linear (pwl) or piecewise constant (pwc) dataset reference.

     Specify a suitable **Uniform** *Film Coefficient* value and choose the desired *Units* from the adjacent drop-down menu.

     Alternatively:

     - For a temperature-dependent film coefficient, reference a dataset (for example, **pwl(HTC1,Temperature)**.
     - For a time-dependent film coefficient, reference a dataset (for example, **pwc (HTC1,Time)**.

       See the topics, Using Datasets to Define Transient Parameters and Piecewise Constant (pwc) Datasets for more information.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Film coefficients vary widely depending on the ambient environment material, whether that material is gaseous or liquid, the orientation of the faces (for natural convection), and the velocity of the gas or fluid along the solid object faces. For forced convection (using a fan or pump), fluid velocity, and therefore the film coefficients, are significantly higher than for natural convection (driven solely by the buoyancy of the ambient medium). The orientation of an object effects natural convection. Vertical faces tend to convect more heat than horizontal ones because horizontal faces obstruct natural convective flow, and the fluid velocities are typically higher along vertical faces.

Anodization of a material (common for aluminum heat sinks) significantly enhances radiative heat transfer, but it does not affect the convective film coefficient.

Tip:

For shell objects, the actual area of the convection assignment face is used in the heat transfer calculations. The resultant convective heat flow is correct for shells overlaid on solid objects, for which convective heat flow only occurs on the exposed side of the shell object. There is no option to automatically consider convective heat flow on both sides of a free-standing portion of a shell object (that is, where a shell has no solid object adjacent to it on either side). There is an easy workaround to achieve the desired results for free-standing shells. Simply double the convection film coefficient that you specify, which is equivalent to the actual film coefficient being applied to twice the surface area. You can also account for differing film coefficients on the two sides. For example, suppose the coefficient is 8 W/(m^2·°C) on one side and 12 on the other. Add the two coefficients together and specify 20 W/(m^2·°C) in the *Convection* dialog box. The resultant heat flow rate and temperatures will correctly reflect the combined effect of convection from both sides of the shell.

- **Non-Uniform:** Film coefficient values will be imported from an Icepak solution. In this case, values are assigned on an individual element face basis and can vary along each of the selected object faces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Imported film coefficients can be positive or negative. For convective cooling, positive values represent heat flowing from the warmer object faces into the cooler surrounding environment (typically air). When radiation effects are included in the Icepak solution, faces that are located near a heat source may receive radiative heat flow that is greater than the heat flow dissipated into the surrounding environment by convection. This phenomenon results in negative film coefficients locally. Even though mechanical-thermal solutions don't directly handle radiation currently, the imported coefficients from an Icepak solution, with radiation enabled, indirectly account for the radiation effects in the mechanical-thermal solution.

Limitation:

Currently, when you import film coefficients from Icepak, they do not represent both sides of free-standing shell objects (that is, shells that do not overlay a solid, non-fluid object on either side). In the Icepak solution, free-standing shells experiences convective and radiative heat flow to/from both sides of the shell. However, the Mechanical solution only imports coefficients representing the heat flow from one side of the shell object. This limitation does not affect shells overlaid on solid objects, which have only one side exposed for convective and radiative heat transfer.

Follow the procedure, How to Setup the Icepak Link, in the *Setup Link* subtopic before continuing to step 4 of this procedure.

4. Optionally, specify an alternative value for the ambient **Temperature** and choose the desired temperature **Units** from the adjacent drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- The default temperature is the variable, *AmbientTemp*, which is a global design setting with a default value of 20°C. The procedure for defining the global ambient temperature is given in step 7 below. You cannot alter design settings while the *Convection* dialog box is still open.

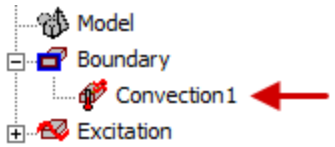
  You can specify a numeric **Temperature** value (along with its units), a user-defined variable, or an expression (consisting of constants, variables, and mathematical operators). For transient thermal solutions, you can specify a dataset reference to vary the temperature with time. Since any user-defined value or dataset overrides the global AmbientTemp value, you can specify a unique temperature for each assigned convection boundary, as needed.

  Refer to the topics, Using Datasets to Define Transient Parameters and Piecewise Constant (pwc) Datasets for more information on using datasets.

- Refer to the Ambient Temperature Guidelines topic for important information concerning ambient temperature values. When importing convection film coefficients from Icepak, it is *especially important* that you specify the ambient temperature appropriately.

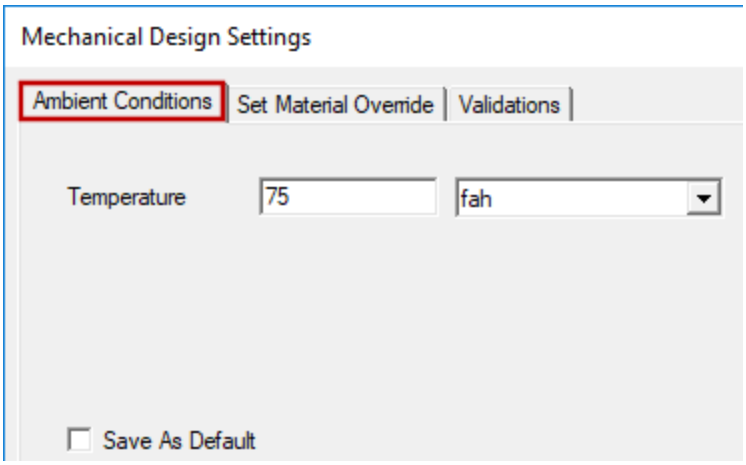
5. Optionally, change the **Name** of the boundary. The default name is Convection*x*, where *x* is a number that increments for each convection boundary that you assign.
6. Click **OK** to assign the boundary.

   The assigned convection appears under *Boundary* in the Project Manager:



7. If you are using the *AmbientTemp* variable for applied convection boundaries and wish to verify or change the value, complete the following procedure:
   a. Access the *Mechanical Design Settings* dialog box using *one* of the following two methods:
      - Using the menu bar, click **Mechanical** > **Design Settings**.
      - Right-click the thermal design heading in the Project Manager [**MechanicalDesign*x* (Thermal)**, by default]. Then, choose **Design Settings** from the shortcut menu.
   b. In the **Ambient Conditions** tab, specify the desired ambient **Temperature** value and use the adjacent drop-down menu to choose the **Units**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



c. Click **OK**.

Setup Link (Convection)

When importing *Non-Uniform* film coefficient results from an Icepak design to a *Convection* boundary in a Mechanical – Thermal design, use the *Setup Link* dialog box to specify the source project, source design, and other linking options.

Imported film coefficients (also called HTCs, for Heat Transfer Coefficients) improve the accuracy of Mechanical – Thermal solutions, as compared to specifying a fixed film coefficient for each applied convection boundary. In Icepak, a fluid region surrounds the solid objects, and you specify various flow parameters, such as:

- Openings (flow inlet and outlet faces)
- Grills (openings with screens or perforations that partially inhibit the flow)
- Velocity of the fluid (for forced convection)
- Ambient pressure of the fluid

The remaining Icepak model setup involves specifying heat sources, selecting materials, and so on. The source Icepak model setup should duplicate the conditions, materials, and geometry in the target Mechanical design, except that a fluid region surrounds the solid parts in the Icepak design. A fluid-thermal multiphysics analysis is performed. Fluid motion can be driven by natural convection (buoyancy) alone, specified fluid velocities, or a combination of both. Fluid velocities are calculated along with the resultant HTCs at each element face. A face's HTC increases as the fluid velocity along the face increases.

Refer to the Icepak help for more information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- The Mechanical design objects that represent heat sources, and those with convection boundaries applied, should have geometrically identical counterparts in the Icepak design. Coefficients are mapped according to the locations of the element faces. Identical geometry ensures that mapping of film coefficients from source element faces to target element faces is performed successfully and accurately. Additionally, the surface mesh size should be similar between the two designs. Though Icepak element faces are quadrilateral, and Mechanical faces triangular, the mapping results will be good if the element lengths are comparable.
- The fluid region in the Icepak design must be omitted from the Mechanical - Thermal design. The convection boundary alone produces the heat exchange between the solid objects and the ambient environment. A fluid region around the parts would represent an additional path for the conduction of heat and would skew the Mechanical – Thermal results.
- In the Icepak solution setup, and when assigning a *Thermal Source* in Icepak (analogous to a *Heat Generation* or *Heat Flux* excitation in Mechanical – Thermal), you have the option of enabling **Radiation** effects. If you do so, the HTCs imported from Icepak will include the effects of both convection and radiation, not just convective heat transfer alone.
- Radiation is very nonlinear, varying according to the difference between the fourth power of the absolute temperature at each surface (T_1^4 - T_2^4). It therefore becomes increasingly significant as the surface temperatures increase. At relatively low surface temperatures, radiation effects are not very significant, as compared to *forced* convection. However, for *natural* convection, the contribution of radiation is usually significant.
- The link between Icepak and Mechanical is a one-way coupling operation. Changes you make in the target Mechanical design have no effect on the source Icepak design's results.
- Mechanical design changes involving the specified ambient temperature, applied power (heat generation, heat flux, or EM loss), applied temperatures, or anything that changes the temperature results (such as material properties) may significantly affect the validity of the imported HTC values from Icepak. This relationship is true under either of the following two conditions:
    - Fluid motion is driven by buoyancy alone (that is, by natural convection), or
    - You are including radiation effects in Icepak (whether the fluid flow is by natural or forced convection)

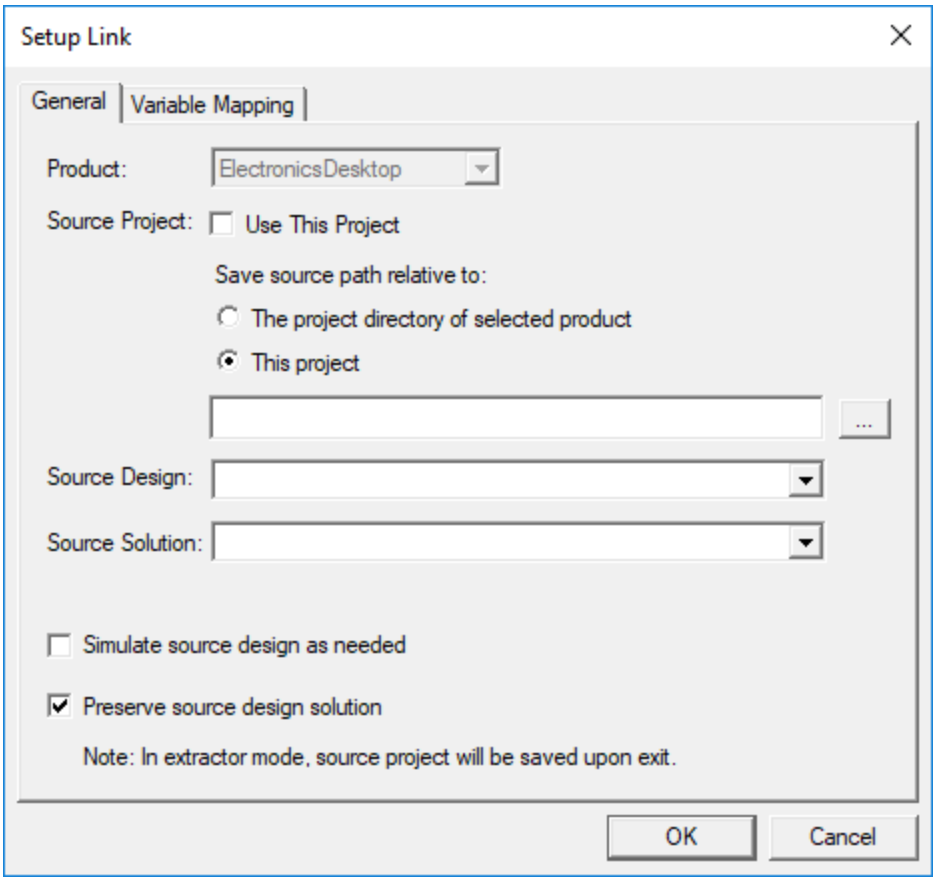
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In such cases, you should consider revising and rerunning the Icepak analysis to keep the two models consistent and to update the HTC values. Then, rerun the Mechanical – Thermal simulation to see the effects of the revised HTC values.

However, when *forced convection* is employed **and** radiation is *not* included, the HTCs do not change significantly with variations in temperature. In this case, you can safely run multiple Mechanical – Thermal analyses based on imported HTCs from a single Icepak solution.

How to Set Up the Icepak Link

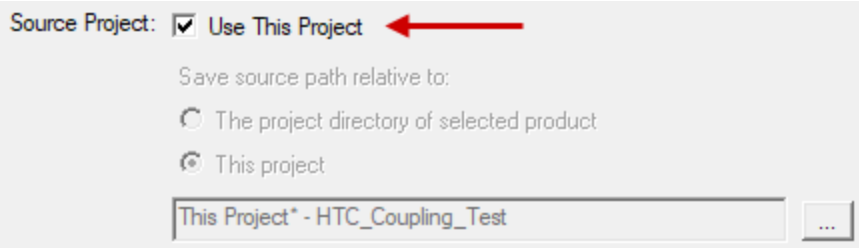
1. Follow steps 1 through 3 of the How to Assign a Convection Boundary procedure, selecting the **Non-Uniform** option in the third step. The *Setup Link* dialog box appears:



2. If the data is to be imported from an Icepak design in the same project, complete the following substeps. Otherwise go to **step 3**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

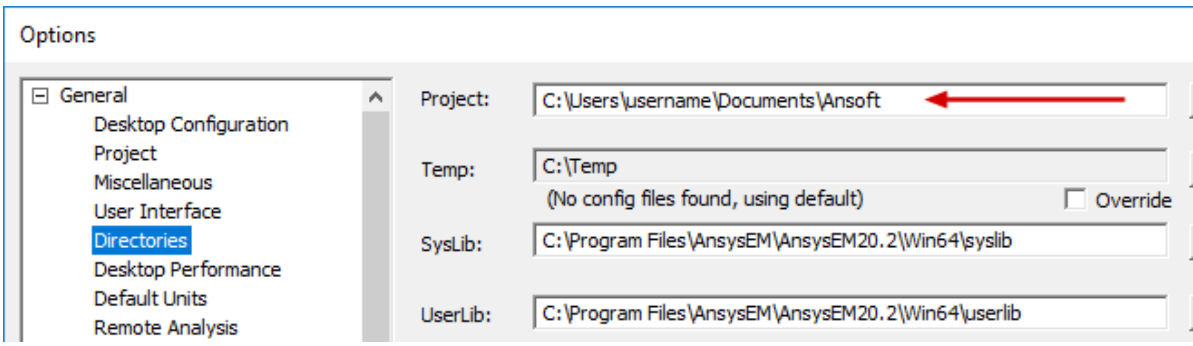
a. Select **Use This Project**.



> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

b. Skip to **step 4**.

3. Alternatively, if the data is to be imported from an Icepak design in a different project, clear the **Use This Project** option and complete the following substeps:
   a. The Mechanical design saves the relative path to the source project in one of two ways. Choose one of the following two options under *Save source path relative to*:
      - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):

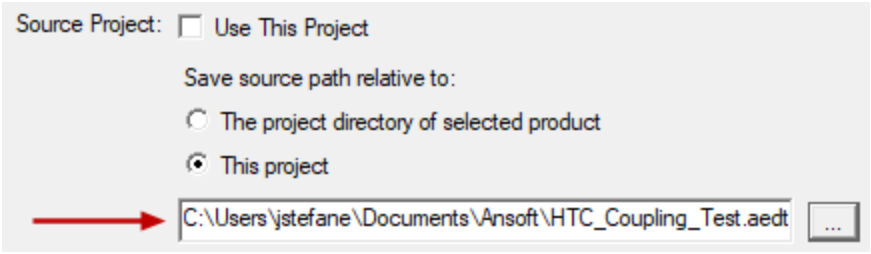


      - **This project:** Save source project path relative to the target project location.
   b. Click the ellipsis button (**...**) to the right of the *Source Project* text box.

      The *Open* dialog box appears.
   c. Locate the source project file and select it.
   d. Click **Open**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected **Source Project** is listed in the *Setup Link* dialog box:



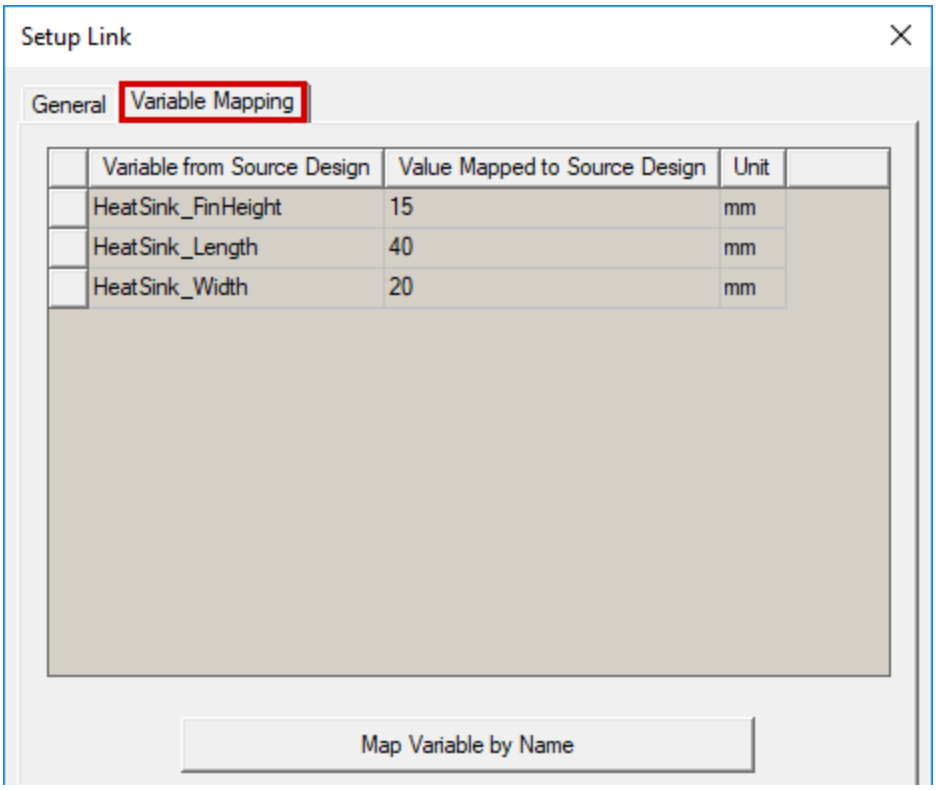
4. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:

> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

5. Optionally, select one or both of the following two options:
   - **Simulate source design as needed:** Select this option if you want an updated solution to be linked to the *target* design when the *source* design has been modified.
   - **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option has no effect if the source design is in the same project as the *target* design.
6. Click the **Variable Mapping** tab.

   If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:

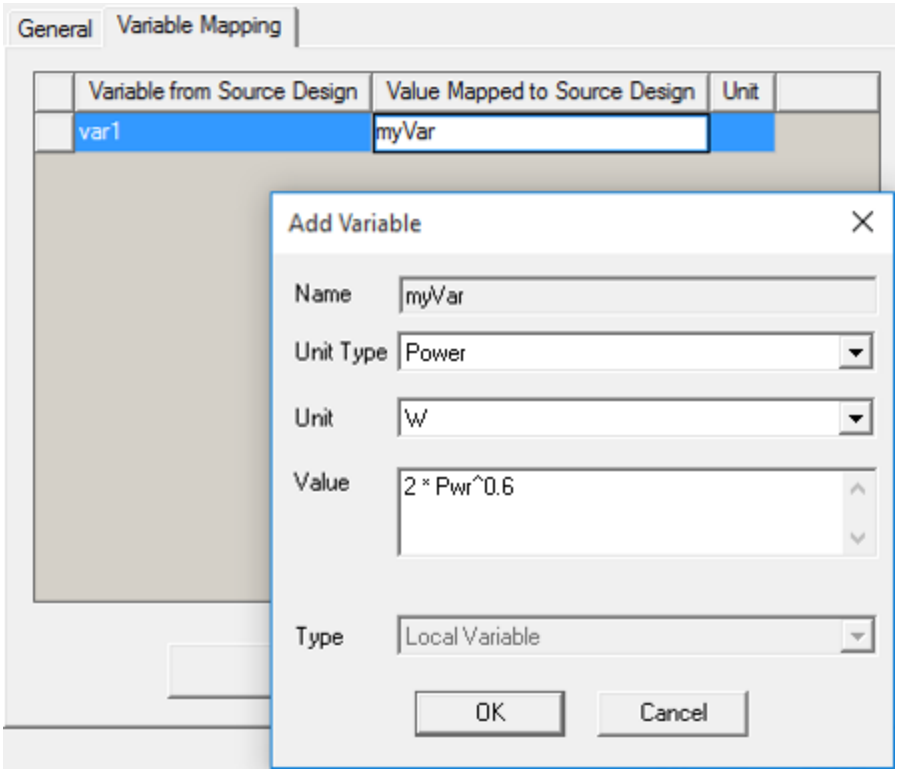
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



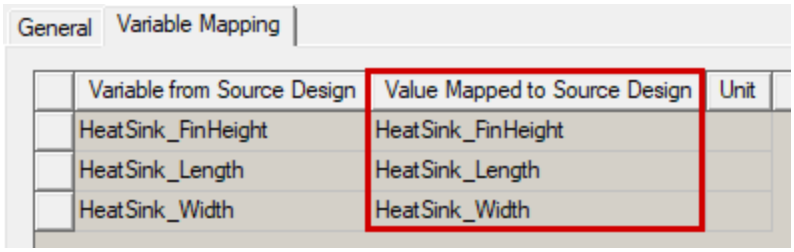
The values passed back to the source design from the mechanical design (second column) can either be constants (numeric values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

> **Note:**
>
> Automatic variable creation is supported in the *Variable Mapping* tab . You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the linked analyses.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- Variables can only be mapped by name if they already exist in the Mechanical design. When you copy and paste geometry from the source, or import it from the clipboard, variables used in the definition of the geometry automatically come into the Mechanical design. However, for non-geometry-related variables (such as a wattage applied to a thermal source excitation in the Icepak design), you would have to manually define the variable within the Mechanical design.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

Mapping variables to the source design by name is useful for Optimetrics analyses (for cases where changing the value of a variable affects both the target design and the film coefficients from the source design). You can create an Optimetrics setup in the target design that sweeps both the target and source designs.

8. Click **OK** to accept the link settings and close the *Setup Link* dialog box.
9. Continue the convection boundary setup at **step 4** of the *Convection* topic.

Note:

If you later revisit the *Convection* dialog box to verify or modify its properties, click the **Setup Link** button under the *General* tab to return to the *Setup Link* dialog box. This dialog box will reopen in **View Only** mode. To modify the properties, first select the **Edit Link** radio button.

Ambient Temperature Guidelines

For *Convection* boundaries, the amount of heat that will flow out of or into the face of the solid object is a function of the following three quantities:

- Thermal power being dissipated or absorbed (heat flux expressed as power per unit area)
- Convection film coefficient
- Temperature difference between the object face and the surrounding ambient air or fluid

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Uniform Film Coefficients:

When you specify an assumed *uniform* film coefficient, the ambient temperature can be any value that makes sense for the application (such as the outdoor air temperature, the interior temperature of a room, or the inlet temperature of air or fluid being forced through an enclosure by a fan). In most electronics enclosures or rotating machines (motors, alternators, or generators) it is appropriate to specify the temperature of the incoming air at the enclosure inlets.

For Non-Uniform (Imported) Film Coefficients:

When importing heat transfer coefficients (HTCs) from Icepak, it is *imperative* that the ambient temperature specified for the *Convection* boundary in the Mechanical design be consistent with the Icepak fluid region's inlet temperature. The reasons are as follows:

- Icepak solves coupled fluid flow / heat transfer multiphysics analyses. Heat transfer from solid objects to the fluid region is by conduction that takes into account the motion of the fluid to move the heat more rapidly than conduction through a static fluid would. No convective film coefficient is assumed. The heat flux is directly computed based on available thermal power or assigned temperatures, material conductivities, and fluid motion.
- Mechanical convection boundaries approximate heat flow in or out of a fluid region without modeling the fluid or the flow pattern. Instead, assumed ambient temperatures and film coefficients (or HTCs) are used to approximate the heat transfer.
- HTCs are not directly imported into Mechanical because they are not a direct product of the Icepak solution. It is the **Heat Flux** and **Temperature** results at each element face that are actually imported from Icepak into Mechanical. The HTCs are then calculated from these Icepak results based on the ambient **Temperature** value you specify for the Mechanical *Convection* boundary. The following equation is used:

$$HTC = \frac{q}{(T_F - T_a)}$$

where:

  - q is the heat flux through the solid element face in units of power per unit area (from results of the Icepak solution)
  - T_F is the solid element face temperature (from the results of the Icepak solution)
  - T_a is the ambient temperature you specify in the Mechanical design's Convection dialog box

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- In order for the calculated HTCs to be valid, T_a must be consistent with the **inlet fluid temperature that was specified in Icepak** and that resulted in the imported heat flux values. If the ambient temperature in Mechanical is not consistent with the inlet fluid temperature in Icepak, the calculated HTCs will not be accurate.
- For complex scenarios, there may be multiple openings in the Icepak air region, and different openings can have differing inlet fluid temperatures. In such cases, the following recommendations apply:
  - If the air is well mixed before reaching the convection boundaries of interest, set the ambient temperature of the Convection boundary in Mechanical to the average of the inlet temperatures in Icepak. If the inlet velocities and areas differ, calculate a weighted average based on the relative volumetric flow rates of each inlet.
  - If certain model faces are positioned to be predominately influenced by the flow from a particular inlet, apply multiple convection boundaries to selective model faces. Do not use the global *AmbientTemp* value for the convection boundaries, Instead, for each separate boundary assignment, numerically specify the ambient temperature equal to the Icepak inlet temperature that is most significant for the selected faces.
  - If the correlation between inlet temperatures and convection faces is not readily apparent, use Icepak, instead of Mechanical, to determine temperature results for all model variants. Icepak solutions require more time and computer resources, but they eliminate concerns about the ambient temperature uncertainty.

Cylindrical Support

A *Cylindrical Support* prevents the faces to which it is applied from translating in any combination of the following three directions:

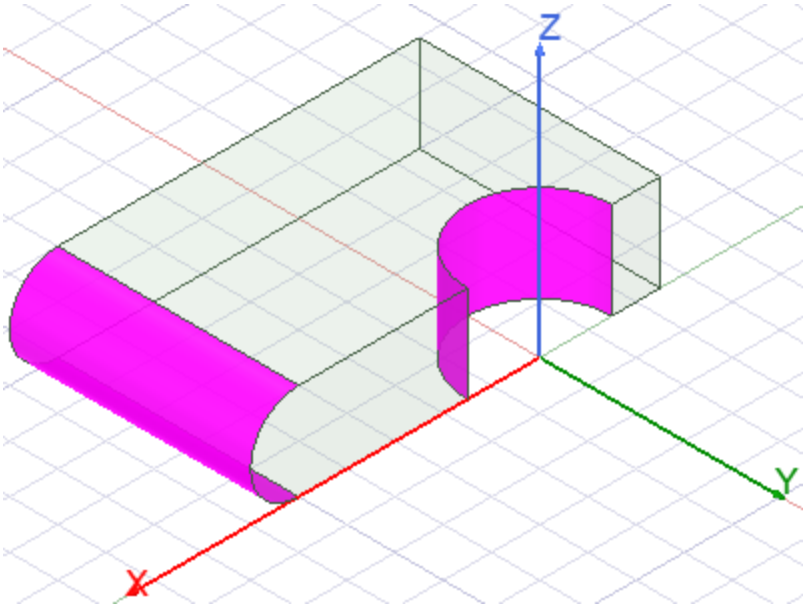
- Radial
- Axial
- Tangential

You can assign a *Cylindrical Support* to only the following entity type:

- **Cylindrical Faces** of solid objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The faces can be full or partial cylinders and can be the outside diameter face of a cylinder or the inside diameter face of a hole or hollow cylinder. Valid faces for a cylindrical support boundary are pictured below (highlighted in magenta):



The radial and tangential directions are evaluated separately at each node along the faces where the support is applied. For each separate cylindrical face, the axial (or thrust) direction is constant because it corresponds to the cylinder's axis at every point on the face.

> **Note:**
>
> The radial constraint direction of the cylindrical face is estimated by averaging the normal vectors of the element faces that meet at each node. The tangential direction is at a right-angle to the radial and axial directions. Therefore, the radial and tangential constraint directions are mesh-sensitive. For the best results, apply local mesh refinement to faces receiving a *Cylindrical Support* boundary. The finer the elements, the smaller the angular differences between adjacent element faces and the greater the constraint direction accuracy.

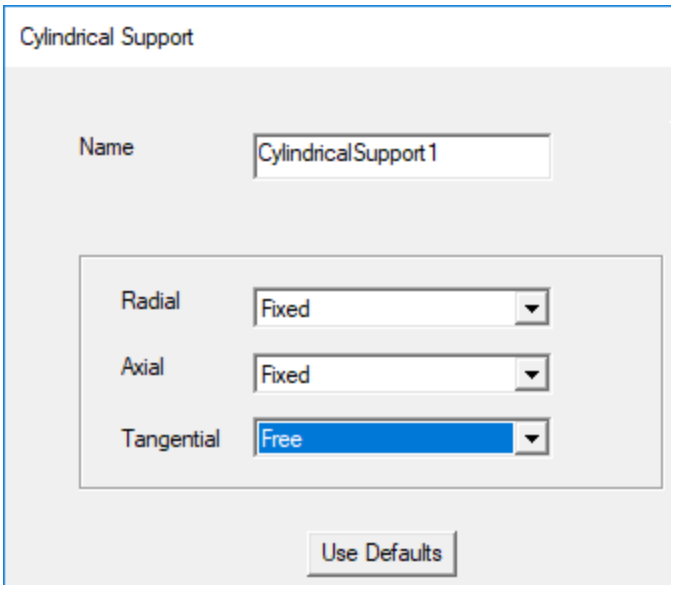
By default, all three directions are initially *Fixed* when you start to apply a *Cylindrical Support* boundary. Therefore, the default boundary definition is equivalent to a *Fixed Support* boundary. To distinguish the behavior of the boundary from a fixed support, and to represent typical cylindrical support behavior, change the setting for one or two directions from *Fixed* to *Free*. For example, to represent a bearing support that prevents radial and axial translation of a shaft but permits tangential motion (that is, shaft rotation), change the **Tangential** setting to **Free**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can change the boundary defaults to suit your most commonly used setup.

How to Assign a Cylindrical Support Boundary:

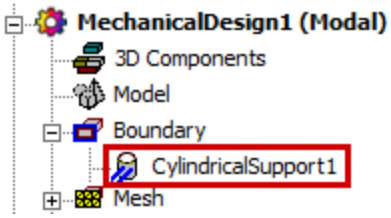
1. Using the *Face selection* mode, select one or more cylindrical faces of solid objects.
2. Use *one* of the following three methods of accessing the *Cylindrical Support* dialog box:
   - Right-click in the Modeler window and choose **Assign Boundary** > **Cylindrical Support** from the shortcut menu.
   - Right-click **Boundary** in the Project Manager and choose **Assign** > **Cylindrical Support**.
   - From the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Cylindrical Support**.
3. In the Cylindrical Support dialog box that appears, make the following changes:
   a. Optionally, change the **Name** of the boundary. (The default name is CylindricalSupport*x*, where *x* is a number incremented for each fixed support you add.)
   b. Using the drop-down menus, selected the desired state (**Fixed** or **Free**) for each of the following directions:
      - **Radial**
      - **Axial**
      - **Tangential**



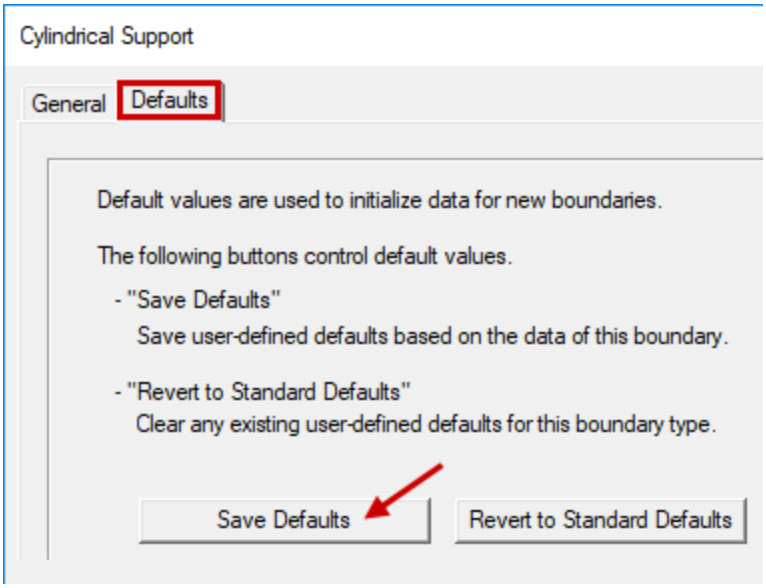
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

   The cylindrical support appears under *Boundary* in the Project Manager:



5. Optionally, if you want to save your cylindrical support settings as the default for future cylindrical supports, do the following:
   a. Under *Boundary* in the Project Manager, right-click **CylindricalSupport*x*** and choose **Properties** from the shortcut menu.
   b. Select the **Defaults** tab.
   c. Click **Save Defaults**:



   Alternatively, you can click **Revert to Standard Defaults** to clear any user-defined preferences and restore the original defaults.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Normally, the *Defaults* tab will not appear when you first assign a cylindrical support boundary. By default, it only appears when you revisit the properties of a previously defined boundary. This behavior is controlled by the option, **Use Wizards for data input when creating new boundaries**, found under the *Mechanical* > *Boundary Assignment* group of settings in the *Options* dialog box. This option is selected by default. When cleared, all tabs of multi-tab boundary dialog boxes appear immediately when you create a new boundary.

To access the *Options* dialog box, do one of the following:

- On the **Desktop** ribbon tab, click **General Options**.
- From the menu bar, click **Tools** > **Options** > **General Options**.

Fixed Support

Fixed supports are applicable to Modal and Structural solutions. A fixed support constrains the selected entities from translating in any of the three global directions (X, Y, and Z).

You can assign *Fixed Support* boundaries to only the following entity types:

- **Faces** of solid objects (Modal or Structural solutions)
- **Edges** of solid objects (Structural solutions only)

The faces or edges to which you assign a *Fixed Support* may be of any shape.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

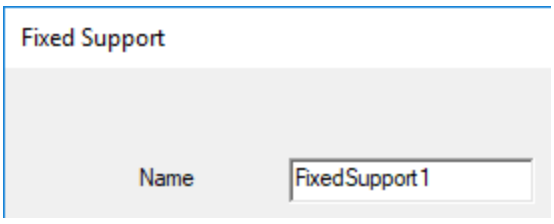
A *Fixed Support* boundary assigned to only a single straight edge is *not* sufficient to make an object statically stable. Since solid elements do not support rotational degrees of freedom, an edge support cannot prevent an object from rotating about the fixed edge; it can only prevent translations. That is, a straight edge with a fixed support assigned to it will act like a hinge.

Any object, or group of connected objects, with a *Fixed Support* boundary assigned to at least one face *is* statically stable. That is, there can be no rigid-body translation or rotation of the objects in any direction because a face cannot rotate if it is not free to translate.

For models that have objects or subassemblies that are not physically bonded to each other, each separate body or set of bodies must be statically stable. That is, fixed, cylindrical, or frictionless supports sufficient to prevent rigid body translations and rotations are required for each separate object or subassembly of objects.

How to Assign a Fixed Support Boundary:

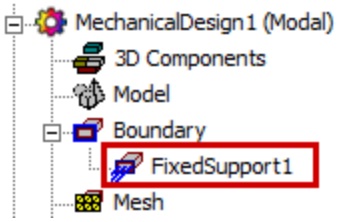
1. Using the *Face selection* mode, or optionally (for structural solutions) the *Edge selection* mode, select one or more solid object faces or edges.
2. Use *one* of the following three methods of accessing the *Fixed Support* dialog box:
    - Right-click in the Modeler window and choose **Assign Boundary** > **Fixed Support** from the shortcut menu.
    - Right-click **Boundary** in the Project Manager and choose **Assign** > **Fixed Support**.
    - From the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Fixed Support**.
3. Optionally, change the **Name** of the boundary. (The default name is FixedSupport*x*, where *x* is a number incremented for each fixed support you add.)



4. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fixed support appears under *Boundary* in the Project Manager:



Frictionless Support

A frictionless support constrains faces from translating in the direction normal to the selected face or faces (both the positive and negative normal vector directions are constrained). You can use it, for example, to model an area of sliding contact between an object of interest and the floor, table, or machine that it rests on.

You can assign *Frictionless Support* boundaries to only the following entity type:

- **Faces** of solid objects

The faces to which you apply a *Frictionless Support* may be of any shape (flat, cylindrical, spherical, conical, or having irregular curvature). The normal direction, and thus the constraint direction, is evaluated separately at each node along the faces where the support is applied. Therefore, the direction of constraint will only be constant for a flat face or multiple parallel flat faces.

> **Note:**
>
> For curved faces, the constraint direction is estimated by averaging the normal vectors of the element faces that meet at each node. Therefore, the constraint direction is mesh-sensitive. For the best results, apply local mesh refinement to *curved* faces receiving a *Frictionless Support* boundary. The finer the elements, the smaller the angular differences between adjacent element faces and the greater the constraint direction accuracy.

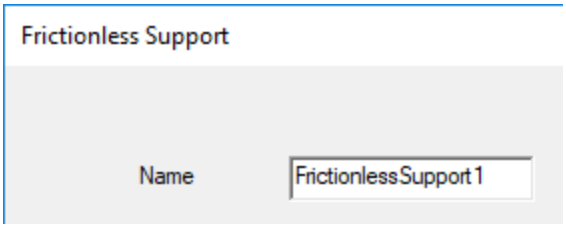
How to Assign a Frictionless Support Boundary:

1. Using the *Face selection* mode, select one or more solid object faces.
2. Use *one* of the following three methods of accessing the *Frictionless Support* dialog box:
   - Right-click in the Modeler window and choose **Assign Boundary** > **Frictionless Support** from the shortcut menu.
   - Right-click **Boundary** in the Project Manager and choose **Assign** > **Frictionless Support**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

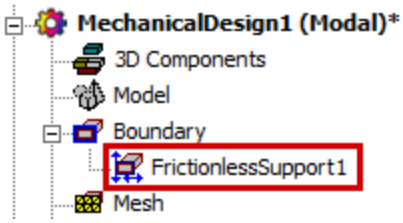
- From the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Frictionless Support**.

3. Optionally, change the **Name** of the boundary. (The default name is FrictionlessSupport*x*, where *x* is a number incremented for each fixed support you add.)



4. Click **OK**.

   The frictionless support appears under *Boundary* in the Project Manager:



Rotating Fluid

Use a *Rotating Fluid* boundary to model the heat transfer in the gap of a rotating machine (such as a motor, generator, or alternator). The *Gap* is the area of clearance between the adjacent cylindrical faces of the rotating and stationary parts (such as the clearance between the rotor and stator of an induction motor). This gap is typically very narrow to maximize the electromagnetic efficiency of the machine.

Background

A thermal analysis of a rotating machine must quantify the flow of heat between the stator and rotor (across the gap that's between them). The fluid is typically air, though any fluid is supported. This heat transfer involves the following considerations:

- Conduction
- Convection
- Radiation

You can ignore radiation because the temperature in rotating machines is not high, convective flow is forced (not due only to buoyancy), and the temperature difference between the rotor and stator surfaces is small. Radiation is insignificant under these conditions as compared to conductive and convective heat transfer.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To model convective heat transfer in the motor, the motion of the fluid must be considered. The conventional way to do this is by using CFD (computational fluid dynamics) analyses. However, this type of computationally expensive study is likely unnecessary in the early stages of motor design. A simpler and faster way to determine the thermal characteristics of a motor uses a steady-state thermal analysis and an *equivalent thermal conductivity* to model the convection within the gap.

Equivalent Thermal Conductivity (k_{equiv})

An equivalent thermal conductivity, which represents the combination of convective and conductive effects, can be calculated and used to determine the flow of heat in the fluid. This calculated equivalent thermal conductivity overrides the thermal conductivity specified in the material's properties.

Definitions of Terms:

- k is the thermal conductivity of the fluid.
- T_1 and T_2 are the surface temperatures of the rotor and stator, respectively.
- δ is the gap thickness.
- α is the convective film coefficient defined by the flow pattern in the gap.

Ignoring the negligible radiation effects, the heat transfer in the air gap can be written as:

Conductive heat transfer (q_{cond}):

$$\dot{q}_{cond} = k\frac{T_2 - T_1}{\delta}$$

Convective heat transfer (q_{conv}):

$$\dot{q}_{conv} = \alpha(T_2 - T_1) = \alpha\delta\frac{(T_2 - T_1)}{\delta}$$

Total heat transfer (q):

$$\dot{q} = (k + \alpha\delta)\frac{(T_2 - T_1)}{\delta} = k_{equiv}\frac{\partial T}{\partial r}$$

Equivalent thermal conductivity (k_{equiv}):

$$k_{equiv} = \alpha\delta + k$$

The following parameters affect the equivalent thermal conductivity:

- Gap radius
- Gap thickness

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Speed of rotation
- Material properties of the working media in the gap (conductivity, mass density, and dynamic viscosity)

Prerequisites for Assigning a Rotating Fluid Boundary:

The following modeling requirements must be satisfied before assigning a *Rotating Fluid* boundary (refer to the images below the list):

- **Rotor:** A cylindrical part, or assembly of parts, forming the rotating portion of the machine. The rotor can be the inner or outer part of the rotating machine (depending on the configuration).
- **Stator:** A cylindrical part, or assembly of parts, forming the stationary portion of the machine. The stator can be the outer or inner part of the rotating machine (depending on the configuration).
- **Band:** A cylindrical region, concentric with the rotor and stator, containing the *Rotor* and a portion of the *Gap* (typically half of the gap thickness). The material for this region is the working media in the gap, which must be a fluid (typically air).
    - For *Configuration 1: Inner Rotor* (left image below), the *Band* is a solid cylinder with an outside radius equal to the radius of the *Rotor* plus half of the *Gap* thickness.
    - For *Configuration 2: Outer Rotor* (right image below), the *Band* is a hollow cylinder with an inside radius equal to the *Stator* radius plus half of the *Gap* thickness. The outer radius of the *Band* must be equal to or greater than the outside radius of the *Rotor*.

  The face at the mid-thickness of the gap is the *Band Surface* of the rotating fluid boundary condition.

  > **Note:**
  >
  > - For either configuration, the *Band* must not touch any stationary object.
  > - The program will perform implicit subtraction of the rotating parts from the *Band*.

- ***Static Region*:** A concentric cylindrical region containing the stationary parts of the rotating machine.

  Similar to the *Band* object, the material for this region is the working media in the gap, which must be a fluid (typically air).

    - For *Configuration 1: Inner Rotor* (left image below), the *Static Region* can be a hollow or solid cylinder. The outside radius must be equal to or greater than the outside radius of the *Stator*. You do not have to subtract the *Band* from the *Static Region*. The program will perform implicit subtraction automatically. If you prefer to

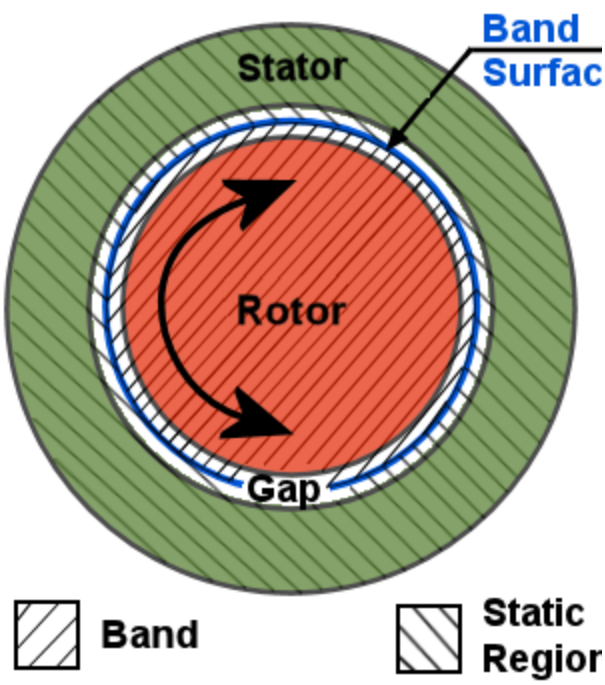
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model the *Static Region* as a hollow cylinder, set the inside radius equal to the *Band Surface* radius.

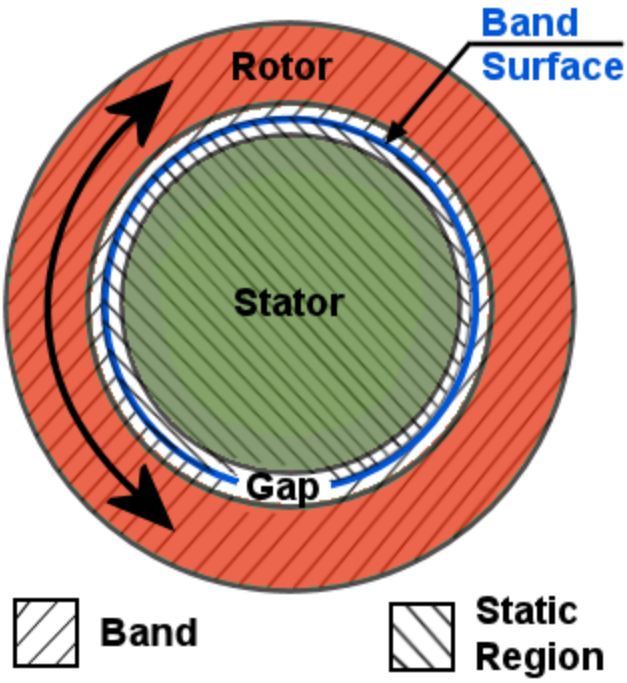
- For *Configuration 2: Outer Rotor* (right image below), the *Static Region* is a solid cylinder having an outside radius equal to the *Band Surface* radius. You can subtract this region from an initial solid *Band* cylinder to produce the required hollow *Band* cylinder.

> **Note:**
>
> Implicit subtraction automatically takes care of the overlap between the *Static Region* and the *Rotor, Stator,* or *Band*.

Configuration 1: Inner Rotating Part



Configuration 2: Outer Rotating Part



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- In both of the above images, the *Gap* is not a modeled object, it is the void between the rotor and stator. However, the gap is occupied by a portion of the Band and Static Region objects.
- Fluid material elements are only generated in the gap, within stator and rotor slots or holes (if any are included in the model), around coils, and in any padding volumes (where the fluid regions extend beyond the end faces of the rotor and stator or beyond the motor outside diameter). Put simply, fluid elements are generated anywhere within a fluid region that is not occupied by a solid part.
- The equivalent conductivity calculated for the motor gap is applied to all fluid objects that are selected when assigning the *Rotating Fluid* boundary. If all fluid regions are selected when assigning the boundary, then all fluid will conduct heat based of the estimated equivalent conductivity in the gap.
- Optionally, in addition to the minimum requirement of a static region and band region, you can include other fluid regions. For example, you might include separate inner and outer regions, a band region, shaft region, or regions encompassing any area where you want to control the thermal treatment (such as around the coils or within slots or holes). By doing so, you can exclude fluid regions from the *Rotating Fluid* boundary assignment so that the estimated equivalent conductivity in the gap is **not** applied to them. Excluded regions will utilize the conductivity defined in the material properties.
- The rotor and stator solid objects can each be comprised of a single cylindrical body or a combination of multiple objects arranged into a cylindrical package. The exception to this rule is when you wish to enable the *Clone Mesh* option, which is a feature of the *Cylindrical Gap Treatment* mesh operation. In this case, clone meshing will fail if either the rotor or stator consist of more than a single cylindrical body. (See the Mesh Considerations topic for more information.)

How to Assign a Rotating Fluid Boundary:

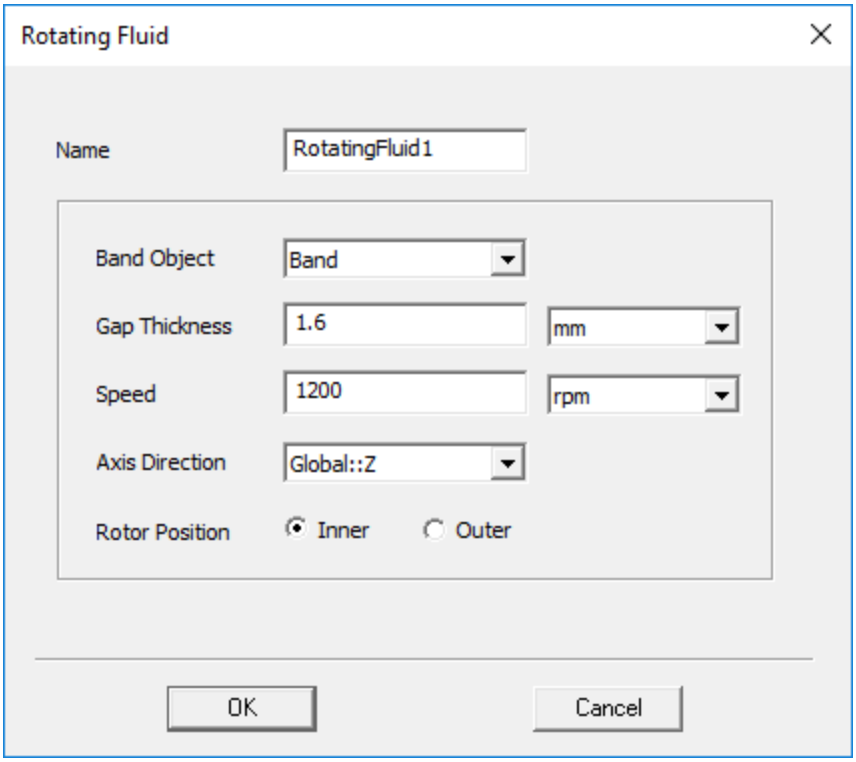
After preparing the model geometry, as described above, apply a *Rotating Fluid* boundary as follows:

1. Select the **Band**, **Static Region** and any other *fluid* objects that are included in the motor model for which you want the estimated equivalent conductivity of the gap applied.
2. Access the *Rotating Fluid* dialog box using one of the following three methods:
    - Using the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Rotating Fluid**.
    - Right-click **Boundary** in the Project Manager and choose **Assign** > **Rotating Fluid** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

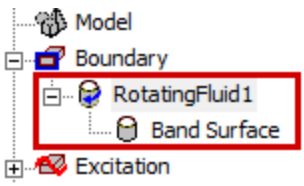
- Right-click in the *Modeler* window and choose **Assign Boundary** > **Rotating Fluid** from the shortcut menu.

3. Specify the following settings:
   a. **Name:** Optionally change the default name to one of your choosing.
   b. **Band Object:** Select the name of the *Band* object from this drop-down menu.
   c. **Gap Thickness:** Specify the gap thickness dimension and its length unit. For gaps with a nonuniform thickness, specify the average gap thickness.
   d. **Speed:** Specify the rotation rate and its units.
   e. **Axis Direction:** Select the axis of rotation from this drop-down menu.
   f. **Rotor Position:** Select either **Inner** or **Outer** to indicate the location of the rotating part of the machine.



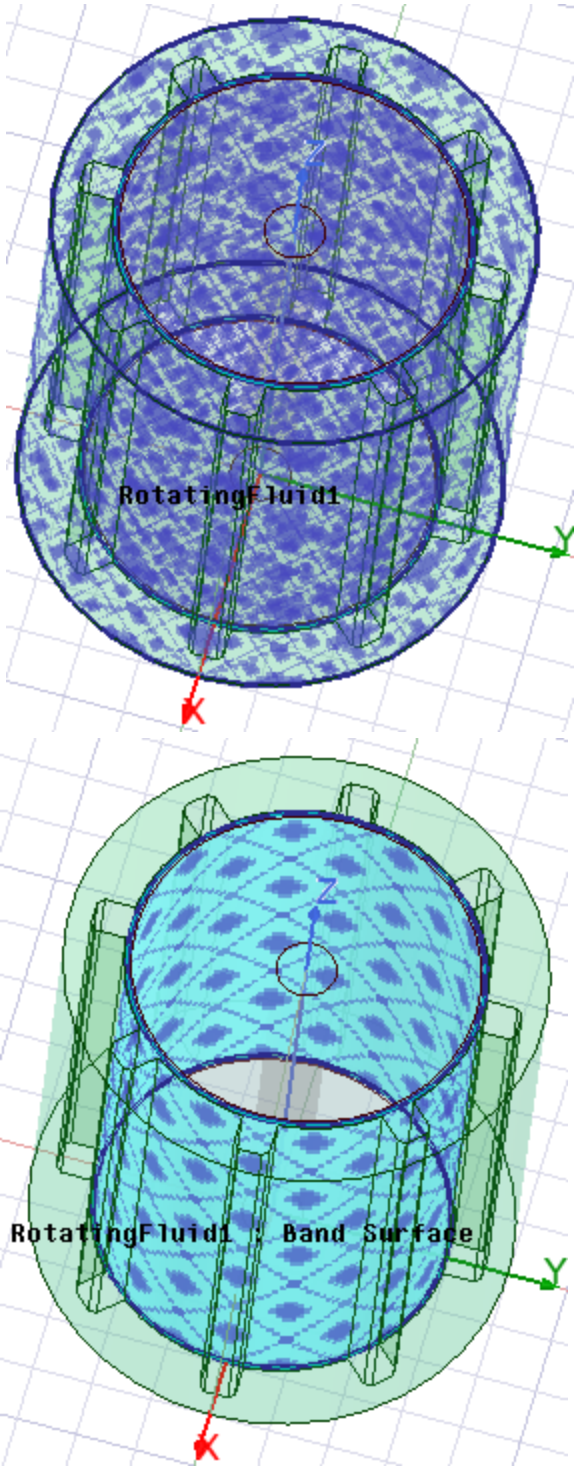
4. Click **OK**.

   The *Rotating Fluid* boundary and its associated *Band Surface* appear under *Boundaries* in the Project Manager:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the boundary or band surface is selected in the Project Manager, the associated visualization is displayed on the model:



5. Apply the **Cylindrical Gap Treatment** mesh operation to achieve the required thermal results accuracy in the gap area. Optionally, you can apply additional mesh operations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, apply the **Rotational Layer Based** mesh operation to improve the rotor and stator mesh in the area adjacent to the gap.

See the Mesh Considerations subtopic.

Note:

For a complete thermal solution setup, you will need to assign the following additional conditions:

- **Heat Generation excitation:** You can determine the amount of heat produced by the motor from it's rated power and efficiency. For example, for an 80% efficient 5 HP motor, the heat generation at full load will be (5 HP / 0.8) * (746 W/HP) * (1 - 0.8) = 932.5 W.

  Alternatively, you can assign an **EM Loss excitation** to import EM losses from a Maxwell 3D motor analysis as the source of heat for your Mechanical–Thermal solution.

- **Convection boundary:** Assign convection boundaries to the outermost cylindrical surface of the outer part (whether rotating or stationary) and optionally to the end faces of the rotating and/or stationary parts to remove heat from the model.

  Alternatively, assign convection boundaries to the outer faces of the fluid regions.

  The steady-state thermal solution provides the temperature and heat flux results when the system reaches a state of thermal equilibrium (that is, when heat in = heat out, and when temperature and heat flux distributions are stable).

Mesh Considerations

The gap in an electrical rotating machine (motor, generator, or alternator) is typically very small compared to the geometry of the overall machine. Local refinement of the element size is needed to provide good results in the gap region without the element count escalating to an impractical number. Relatively coarser elements are acceptable for the bulk of the volume comprising each of the rotating machine objects (that is, in the areas sufficiently removed from critical regions and small features).

Ansys Electronics Desktop offers a number of mesh controls for providing local refinement. Several of these controls are general in nature and can be applied to models and designs of many different types. In addition, there are mesh controls specifically intended for rotating machinery.

The following list contains a few of the available mesh operations. These are ones that may prove useful in a preliminary rotating machine thermal simulation.

General Mesh Operations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Length Based:** Element length constraint that may be applied to the surface mesh or inside selected objects

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **On Selection: Length Based** or **Assign** > **Inside Selection: Length Based**)

- **Skin Depth Based:** Surface triangle length constraint in which you also specify the *Skin Depth* and the *Number of Layers of Elements* to create within the specified skin depth

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **On Selection: Skin Depth Based**)

- **Surface Approximation:** Adjust the resolution of meshes along selected curved faces using a simple fine/coarse slider control or manually specified parameters.

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **Surface Approximation**)

- **Apply Curvilinear Meshing:** Create curvilinear elements along selected curved surfaces or along all curved surfaces of selected objects.

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **Apply Curvilinear Meshing**)

Mesh Operations Specifically for Rotating Machines:

- **Cylindrical Gap Treatment:** Applied to the *Band* object of a rotating machine model

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **Cylindrical Gap Treatment**)

  This mesh operation is essential for obtaining good thermal results from rotating machinery models. It identifies the two portions of the *Gap* that are separated by the *Band Surface*. The side adjacent to the rotor is called the *Moving Side*, and the side adjacent to the stator the *Static Side* (though, in reality, all of the fluid in the gap is treated as moving). It applies either default or user-specified mesh parameters to the elements within each side of the *Gap*.

  Default parameters are applied when the **Clone Mesh** option is cleared (that is, *not* selected). Specifically, the default parameters are a **Band Mapping Angle** of **3°** (this parameter is the angular location increment of the nodes around the circumference), and the **Number of Layers** = **1** each for the **Moving Side** and **Static Side**.

  Additionally, this operation aligns nodes in the axial direction along the *Band Surface* into discrete rows, with nodes in each row having the same axial coordinate. In the gap, relative motion causes the temperature to be constant in the angular direction. When a 360° ring of nodes is lined up at a constant axial position, they can also be assumed to be at the same temperature, though the temperature can vary from row to row in the axial direction.

  Apply the *Cylindrical Gap Treatment* mesh operation to all models with a *Rotating Fluid* boundary applied.

  - ***Clone Mesh* option:** When you select this option, the following additional user-defined parameters become available:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mapping Angle:** The angular increment around the circumference of the *Band* at which elements are created
- **Number of Layers – Moving Side:** The number of element layers generated on the moving side of the *Gap*
- **Number of Layers – Static Side:** The number of element layers generated on the static side of the *Gap*

> **Note:**
>
> Additional model requirements are imposed when you select the *Clone Mesh* option. (See Prerequisites for using the Clone Mesh Option below.)

- **Rotational Layer Based:** For selected solid objects, generate a **Number of Layers** (up to 3) along the faces nearest to the *Band Surface* and having a user-specified *Total Layer Thickness* (combined).

  (Right-click **Mesh** and choose **Assign Mesh Operation** > **Inside Selection: Rotational Layer Based**)

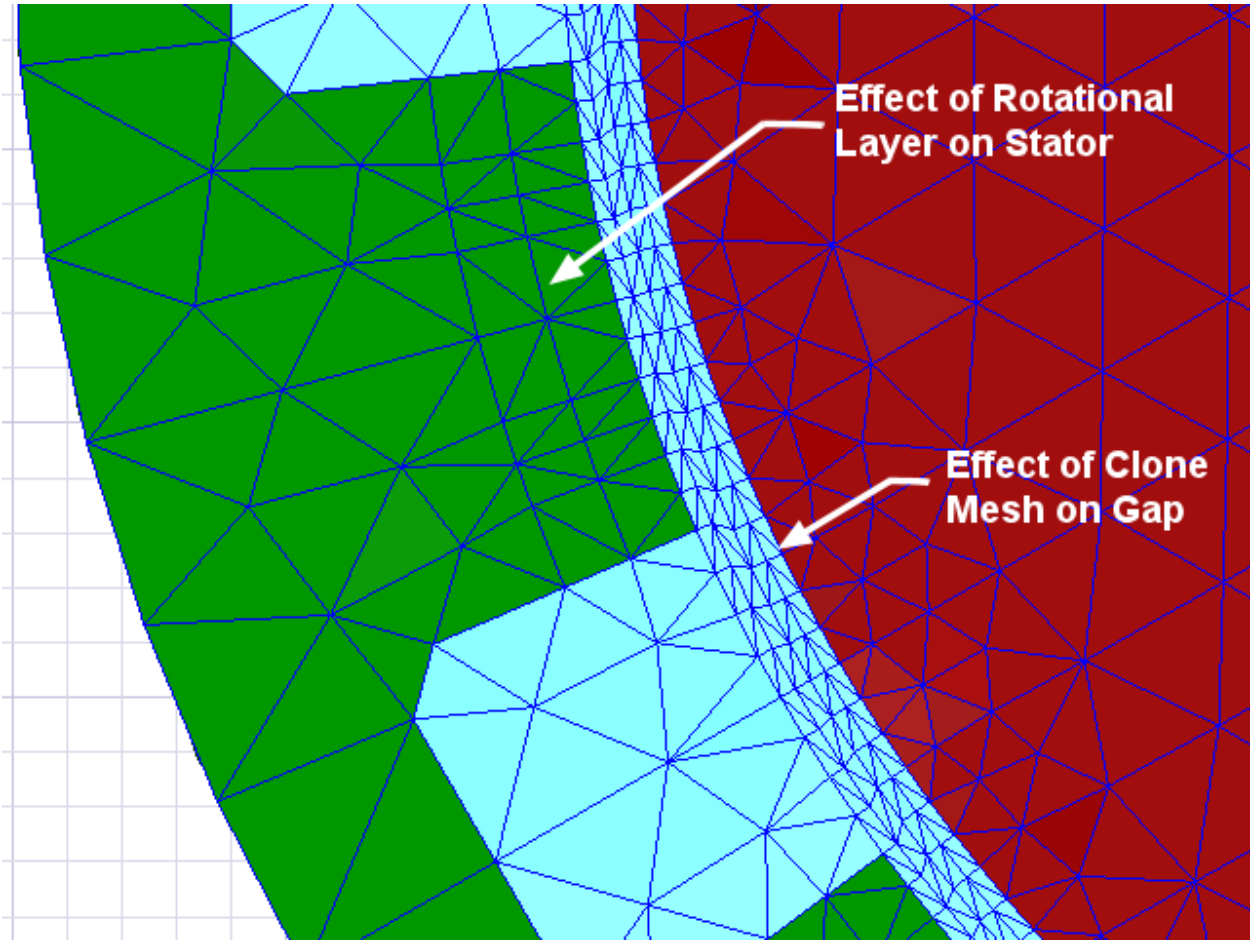
  This mesh operation is only valid if *Cylindrical Gap Treatment* has also been applied to the model.

  > **Note:**
  >
  > Additional model requirements are imposed when you apply this mesh operation. (See "Prerequisites for using the Rotational Layer Based Mesh Operation" on page 9-35 below.)

> **Note:**
>
> Each of the mesh operations discussed on this page, and others as well, are documented within the Maxwell help. Search for "*Defining Mesh Operations*" in the Maxwell help to quickly locate this information.

The following image shows the effects of *Cylindrical Gap Treatment* with the *Clone Mesh* option selected. The parameters were **Mapping Angle** = **1.5 deg**, *Number of Layers* = **2** each (**Moving Side** and **Static Side**). Also, the *Rotational Layer Based* mesh operation was applied to the stator (Green object). The parameters were **Number of Layers = 2** and **Total Layer Thickness = 0.28 inch**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Prerequisites for using the Clone Mesh Option

When you apply the *Cylindrical Gap Treatment* mesh operation and select the *Clone Mesh* option, the following additional modeling requirements must be satisfied. These requirements are in addition to those listed under Prerequisites for Applying a Rotating Fluid Boundary in the *Rotating Fluid* topic. The additional requirements are *not* imposed for *Cylindrical Gap Treatment* if the *Clone Mesh* option is cleared (*not* selected).

- The *Band* must enclose at least two rotating parts. In a simplified representation of a rotating machine, one easy way to satisfy this requirement is to add a shaft to the model. In a more complex model there may be coils, a squirrel cage, or other typical rotor components.
- Either the rotating part or the stationary part must include a repeating pattern of features around the circumference (such as coil slots). The mesh at one of the repeating features is then cloned in a circular pattern to provide the mesh for the other related features. The

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

clone mesh will fail if both the rotating part and the stationary part are simple solid or hollow cylinders.

When the *Clone Mesh* fails, the program automatically falls back to a conventional mesh.

Prerequisites for using the Rotational Layer Based Mesh Operation

When you apply the *Rotation Layer Based* mesh operation, the following additional modeling requirements must be satisfied. These requirements are in addition to those listed under Prerequisites for Applying a Rotating Fluid Boundary in the *Rotating Fluid* topic.

- Individual protrusions must exist around the circumference of the target object (such as poles separated by slots or the bars of a squirrel cage).
- The axial length of the protruding features in the target refinement area (adjacent to the *Band Surface*) must be less than the Band's axial length. That is, the *Band* must include padding beyond the limits of the rotating parts it encloses, which is a standard feature of Maxwell rotating machine designs, or beyond the length of stationary parts receiving this refinement.
- The specified **Total Layer Thickness** cannot exceed the *Gap* thickness times the **Number of Layers** requested. However, if you violate this rule, the mesh operation won't fail; the *Total Layer Thickness* will automatically be adjusted to an acceptable value. A message will appear in the *Message Manager* window indicating that this adjustment has been made.

When the *Rotational Layer Based* mesh operation fails, the conventional mesh is completed without the requested refinement.

Temperature

Use *Temperature* boundaries to dictate fixed temperatures at selected faces. Alternatively, for transient solutions, the temperature can vary with time. Heat flow can be in either direction at the temperature boundary, either into the object's face, out of the object's face, or a combination of the two directions (in at some nodes, out at others).

You can assign temperature boundaries to the following entities:

- **Faces** of solid or shell objects

How to Assign a *Temperature* Boundary:

1. Select one or more faces of solid or shell objects.
2. Access the *Temperature* dialog box using one of the following three methods:
    - Using the menu bar, click **Mechanical** > **Boundaries** > **Assign** > **Temperature**.
    - Right-click **Boundary** in the Project Manager and choose **Assign** > **Temperature**

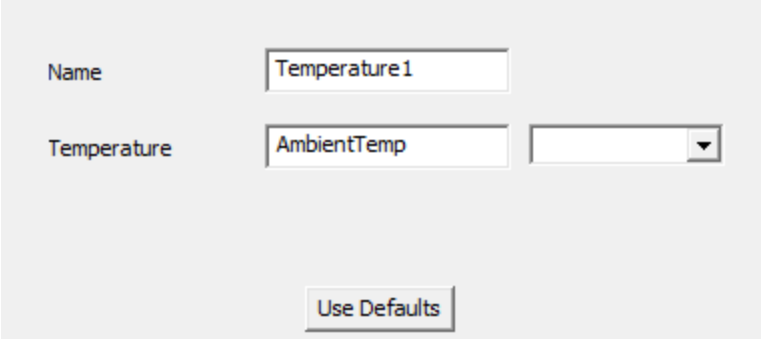
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the shortcut menu.

- Right-click in the Modeler window and choose **Assign Boundary** > **Temperature**.

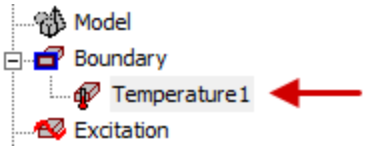
3. Optionally, change the **Name** of the boundary. The default name is Temperature*x*, where *x* is a number that increments for each temperature boundary that you apply.
4. Specify the **Temperature** to assign, which can be numeric, a variable, or an expression (consisting of numbers or variables and mathematical operators). Additionally, for transient thermal solutions, the value can be a time-dependent dataset reference.

> **Note:**
>
> The default temperature is the global variable *AmbientTemp*, for which you can be change its default value (20° C) in the *Mechanical Design Settings* dialog box (see step 7).



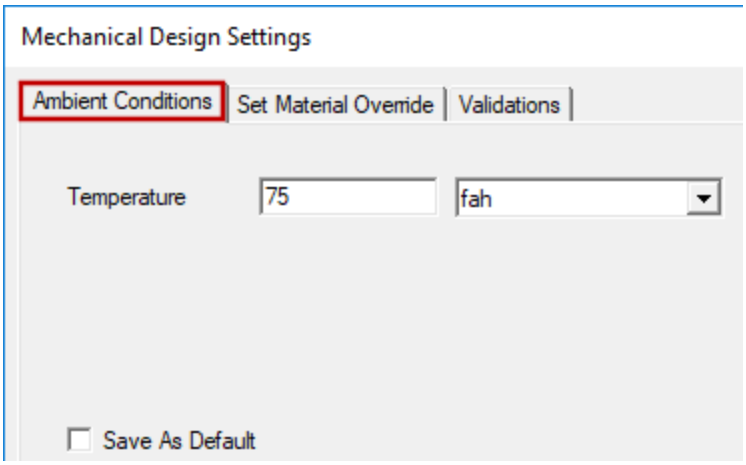
5. Choose the desired *Temperature* **units** from the drop-down menu.
6. Click **OK** to apply the boundary.

   The assigned temperature appears under *Boundary* in the Project Manager.



7. If you are using the *AmbientTemp* variable for applied temperature boundaries, access the *Mechanical Design Settings* dialog box using *one* of the following two methods:
   - Using the menu bar, click **Mechanical** > **Design Settings**.
   - Right-click the thermal design heading in the Project Manager [**MechanicalDesign*x* (Thermal)**, by default]. Then, choose **Design Settings** from the shortcut menu.
8. In the **Ambient Conditions** tab, specify the desired ambient **Temperature** value and units.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Click **OK**.

Excitations

For Mechanical designs, the available excitations depend on the *Solution Type* (Model, Thermal, or Structural). No excitations are available for Modal solutions. For Thermal or Structural solutions, the available excitations are as listed below:

- **Thermal Solutions:**
  - EM Loss
  - Heat Flux
  - Heat Generation
- **Structural Solutions:**
  - Force
  - Pressure
  - Displacement
  - Thermal Condition

Displacement

A *Displacement* excitation has two purposes:

- Displace a face or edge a specified distance in a specified direction.
- Prevent motion (constrain) a face or edge in one or more particular directions while allowing motion in the other directions.

Often, a displacement excitation is used to achieve a combination of these two purposes. For example, you might want to displace an edge or face a certain distance in the X direction (non-zero X Displacement), prevent motion in the Y direction (Y Displacement = 0), and allow the Z

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

displacement that will occur naturally due to motion or deformation of the structure (Z direction = "Free"). You can achieve all three of these goals by assigning a single displacement excitation.

You can assign *Displacement* excitations to the following entity types:

- **Faces** of solid objects
- **Edges** of solid objects

The faces or edges to which you assign a *Displacement* may be of any shape or orientation, flat/straight or curved.

How to Assign a Displacement Excitation:

The appearance of the *Displacement* dialog box varies depending on whether the selection set includes an edge and whether the *Normal To* or *Component* option is selected (when applicable). The following procedure covers all possible variants:

1. Using the *Face, Edge,* or *Multi* selection mode, select one or more solid object **faces** and/or **edges**.
2. Use *one* of the following three methods of accessing the *Displacement* dialog box:
    - Right-click in the Modeler window and choose **Assign Excitation** > **Displacement** from the shortcut menu.
    - Right-click **Excitations** in the Project Manager and choose **Assign** > **Displacement**.
    - From the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Displacement**.
3. Optionally, change the **Name** of the excitation. (The default name is Displacement*x*, where *x* is a number incremented for each displacement you add.)
4. If your selection set includes only faces, choose one of the following two options from the **Define By** drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

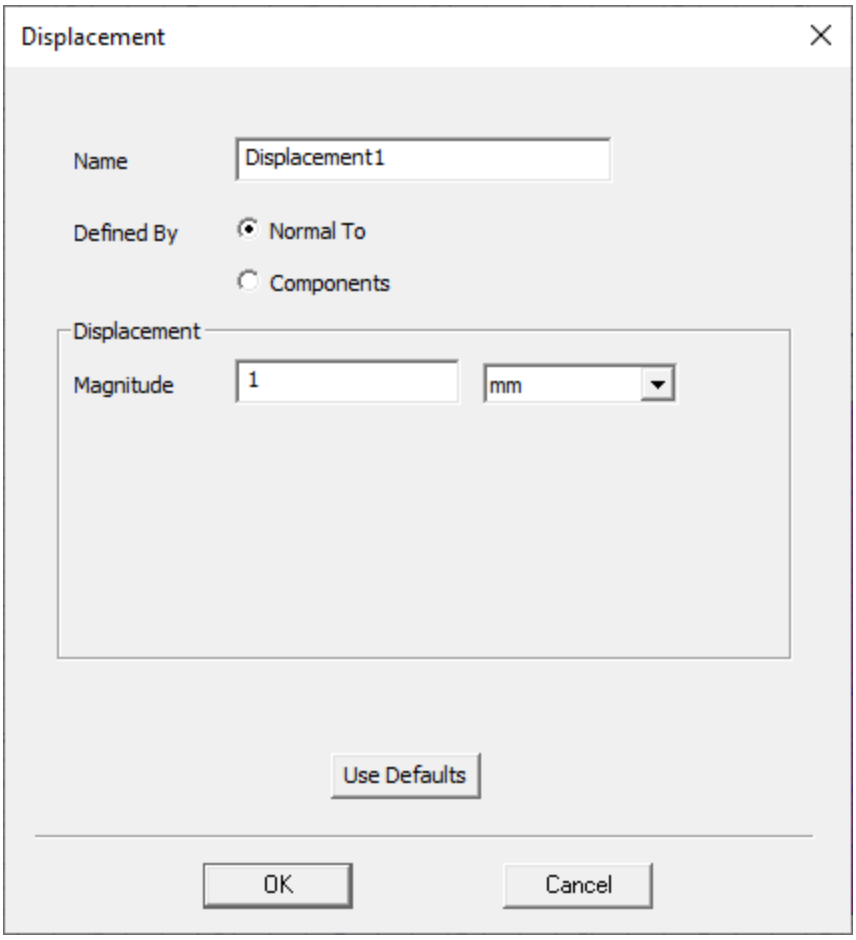
- **Normal To:** Displacement acts in the face-normal direction

  > **Note:**
  >
  > For this option:
  >
  > - A positive displacement pulls outward on the face, away from the solid body.
  > - A negative displacement pushes inward on the face, towards the solid body.
  >
  > For a curved face, the normal direction varies and is separately evaluated at each node along the face, since both displacement excitations and resultant displacements are computed at nodes.

- **Component:** Specify X, Y, and Z components of displacement (global and user-defined coordinate systems are supported)

5. Specify the displacement as follows, depending on the type of selected entities and your selection in the previous step:
   - If you selected the **Normal To** option (faces only), the dialog box appearance will be as shown in the following figure:

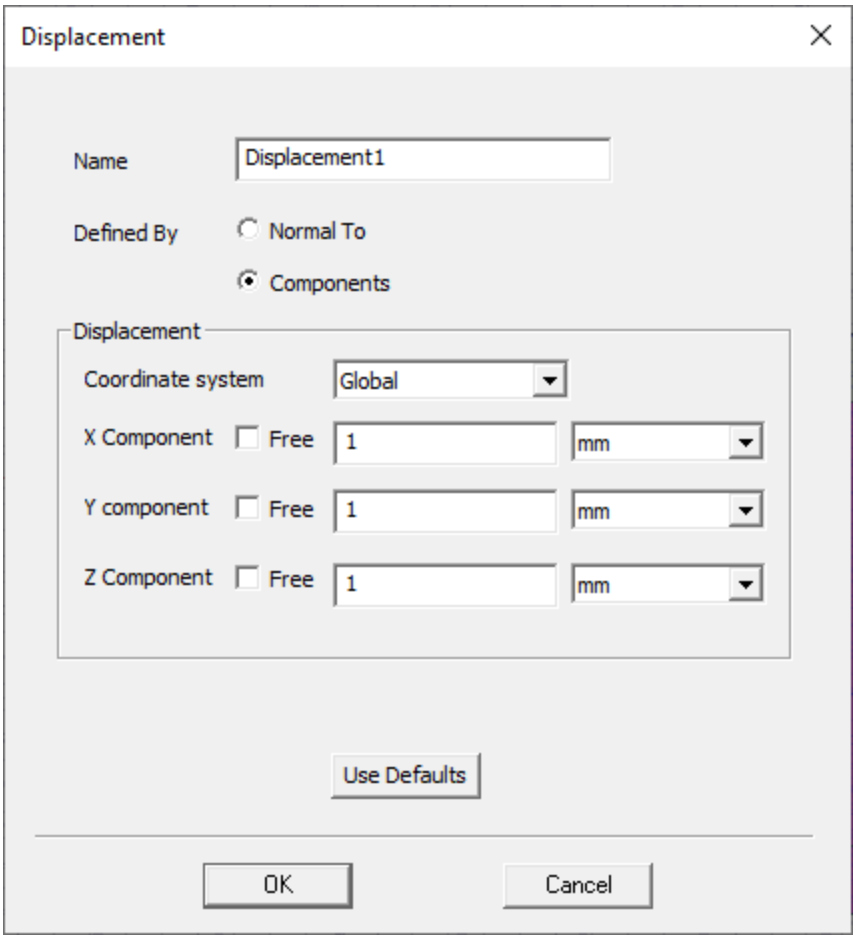
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Specify the desired displacement **Magniutde**.
b. Choose the length units from the adjacent drop-down menu.

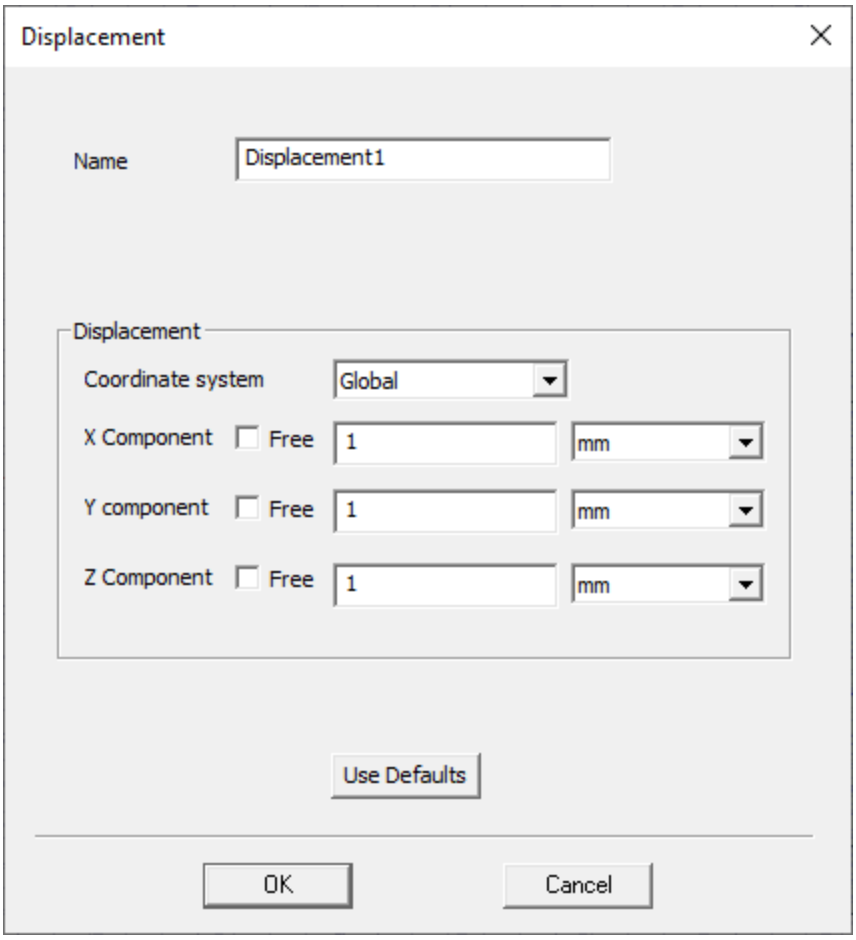
- If you selected the **Components** option (faces only), the dialog box appearance will be as shown in the following figure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Or, if your selection set included at least one edge, the dialog box appearance will be as shown in the following figure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In either case, do as follows:

a. From the **Coordinate System** drop-down menu, choose the CS on which the displacement components will be based.

   The menu will list the Global CS and any user-defined CSs that exist in the design. *Global* is selected by default.

b. Optionally, for any direction in which you do NOT want to impose a displacement but want to leave the selected entities free to displace on their own, select the **Free** checkbox.

   Select one, two, or none of the Free checkboxes. If you select all three, the displacement excitation will have no effect on the model.

c. For non-Free directions, type the displacement component magnitudes in the **X Component, Y Component,** and **Z Component** text boxes.

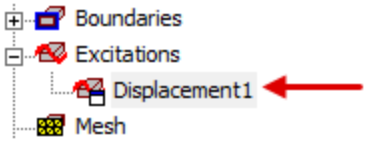
   The values may be zero (to prevent motion), positive, or negative.

d. Choose the length units for each applicable component from the adjacent drop-down menu. For zero-components, the units choice doesn't matter.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK**.

   The displacement appears under *Excitations* in the Project Manager:



EM Loss

You can import electromagnetic (EM) losses from other Ansys Electronics Desktop designs to use as thermal excitations in the target Mechanical – Thermal design. EM Losses may be volumetric or surface based depending on the associated source object. You can assign EM losses to only the following entity type:

- **Objects** (solid or shell)

You can import EM losses from another design in the same project file or from a different project. The program first looks for a source object with the same name as the target object. The program attempts to determine the type of loss (surface or volume) from the properties of source objects with the same name as the target objects. However, actual EM loss mapping is done based solely of the X, Y, and Z locations of the element centroids.

The source and target object names, geometry, and meshes do not have to match exactly. However, you will obtain the best results when the source and target geometry is the same, and the mesh quality and element densities (fineness or coarseness) are very similar. For this reason, you may want to take advantage of mesh linking. The exception, of course, is when you are importing EM losses from a Maxwell 2D model to your Mechanical–Thermal model, which is always 3D.

You can import EM loss data from the following types of source designs into a Mechanical – Thermal target design:

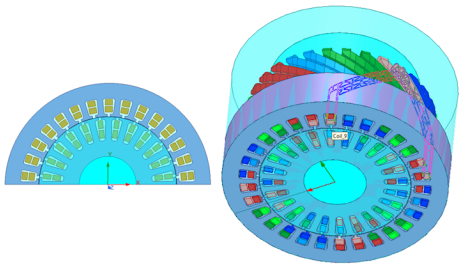
- **HFSS:** From Modal, Terminal, or Eigenmode solutions – import surface losses from non-solve-inside conducting objects and volume losses from solve-inside lossy objects.

  However, you cannot import EM Losses from HFSS source designs that include an IE or PO Region.

- **Maxwell 3D:** Surface losses from *Impedance* boundaries, and volume losses from solve-inside conducting objects
- **Maxwell 2D:** Losses can be mapped from a 2D model that lies on the global XY plane to a Mechanical–Thermal target model. The end face of the solid geometry must match the 2D model. The 3D model can be an extrusion of the end face geometry in either the +Z or the -Z direction, and the extrusion can include swept motion or rotation in the X and Y directions. To clarify these statements, considering the following image comparing a 2D model of an electric motor (with half-symmetry about the XZ plane) and a corresponding

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D model (with half-symmetry about the XY plane). Notice how the windings are extruded along a path with both axial and tangential progression:



The features of the 2D model are exactly represented on the same plane in the 3D model. The program mirrors the half-symmetry results and applies the loss to the full set of 3D objects extruded from the faces lying along the XY plane. Note that RMxprt provides a way to export solved models as either Maxwell 2D or Maxwell 3D designs. The geometry for a Mechanical design can be copied from a Maxwell 3D design's geometry. Either the corresponding Maxwell 2D or Maxwell 3D design can be used as the source model for EM Losses in the Mechanical–Thermal solution.

The 2D solution assumes that the geometry is extruded purely in the Z direction. Therefore, for models such as in this example, 2D-to-3D mapping of losses is less accurate than a 3D-to-3D coupling of identical geometry. However, the solution time is much shorter, and the accuracy sufficient for initial rough-design phases.

- **Q3D:** Surface losses from conducting objects for the AC field type, and volume losses from lossy objects for the DC field type

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

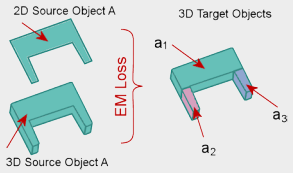
Note:

- Lossy objects are those with material definitions that include a *Dielectric Loss Tangent* greater than zero.
- You can apply multiple EM Loss excitations to the same object, each importing the EM Loss from a different design. Use this capability to superimpose EM Losses (for example, combining volume losses from Q3D with surface losses from HFSS), to see the thermal effects when two or more EM Losses are combined.
- You can scale each imported EM Loss excitation by a user-defined multiplier. For example, you could increase the imported loss by a factor of 1.25 to build a 25% safety margin into your thermal solution.
- You can offset volumetric and surface-based EM Loss excitations with Heat Generation or Heat Flux excitations applied to the same volumes or surfaces, respectively. The offset excitation can increase or decrease the EM Loss excitation.
- A summary of coupled EM Loss data can be viewed on the *Profile* tab of the *Solutions* dialog box during the solution process. In the Project Manager, expand **Analysis**, right-click the solution setup, and select **Profile**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

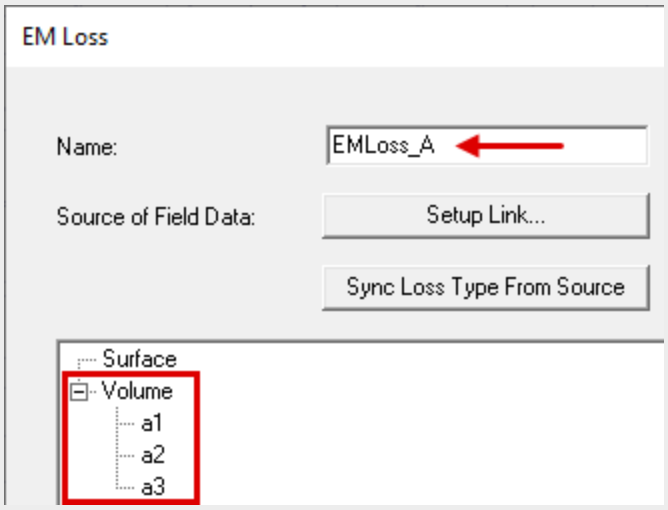
Warning:

Ideally, there should be a one-to-one correlation between source 2D or 3D objects and target 3D objects. However, it is possible for a source object to correspond to multiple smaller objects in the target design, with the combination of the target objects corresponding to the larger source object, as shown in the following example:



You must handle this situation in a particular way:

- Specify the EM Loss between the single source object (“A”) and the multiple corresponding target objects (“a1,” “a2,” and “a3”) in a **single excitation**. If you look at the properties of the EM Loss assignment, it must resemble the following image (with a single excitation (EMLoss_A) including all three volumes as targets:



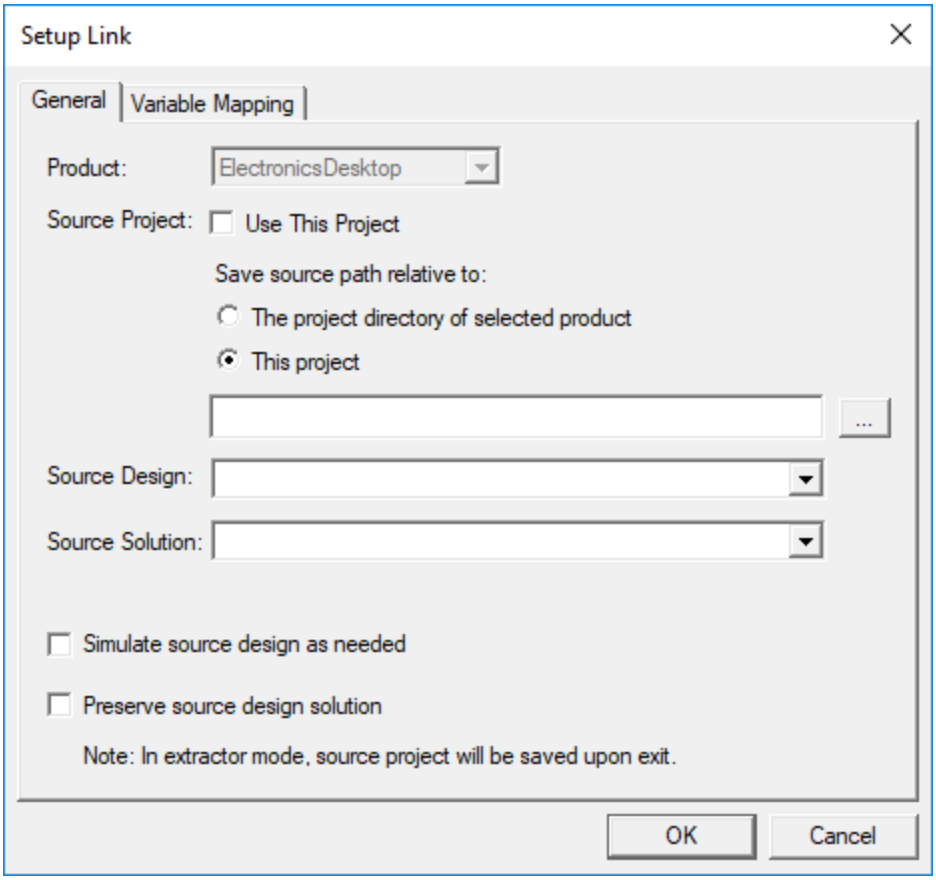
- Conservation of loss is enforced at the object level for each individual EM Loss excitation. That is, the total loss of the target objects always equals the total loss of the source objects for each EM Loss excitation. If you were to assign the EM loss of a large source object to multiple, smaller target objects in separate excitations (A-to-a1, A-to-a2, and A-to-a3), each of the smaller target objects would receive the full EM loss of the larger source object (“A”), thus exaggerating the total EM loss in the target design, and the solution results would be incorrect.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Assign an EM Loss Excitation:

1. Select one or more target objects to which you want to apply imported EM losses. Whether the linked EM losses will be surface-based or volumetric, you always assign them to *objects*.
2. Begin applying an *EM Loss* excitation using one of the following three methods:
   - Right-click in the Modeler window or on a selected object in the History Tree and choose **Assign Excitation** > **EM Loss** from the shortcut menu.
   - Right-click **Excitation** in the Project Manager and choose **Assign** > **EM Loss**.
   - Using the menu bar, click **Mechanical** > **Excitations** > **Assign** > **EM Loss**.

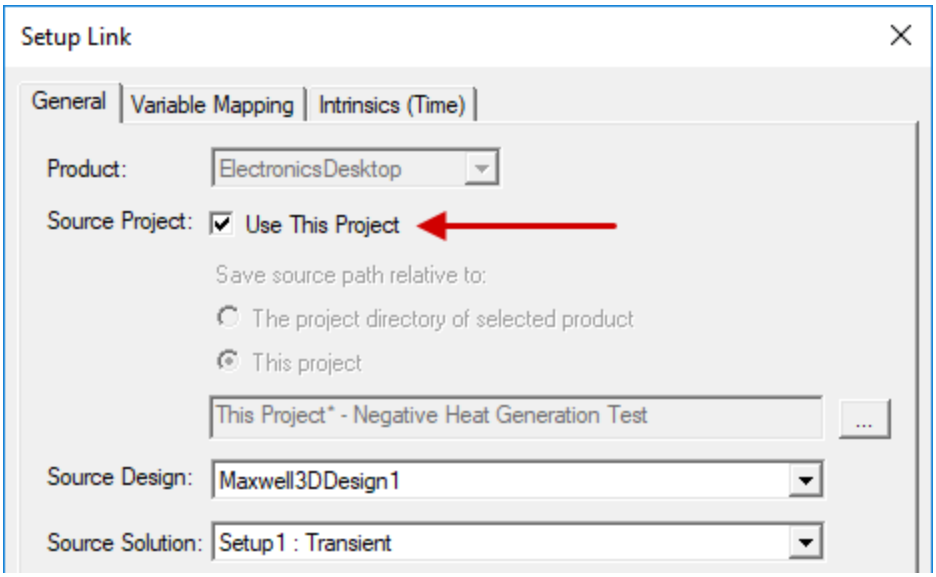
   The *Setup Link* dialog box appears:



3. If the EM loss data is to be imported from a design in the same project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Select **Use This Project**:



> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

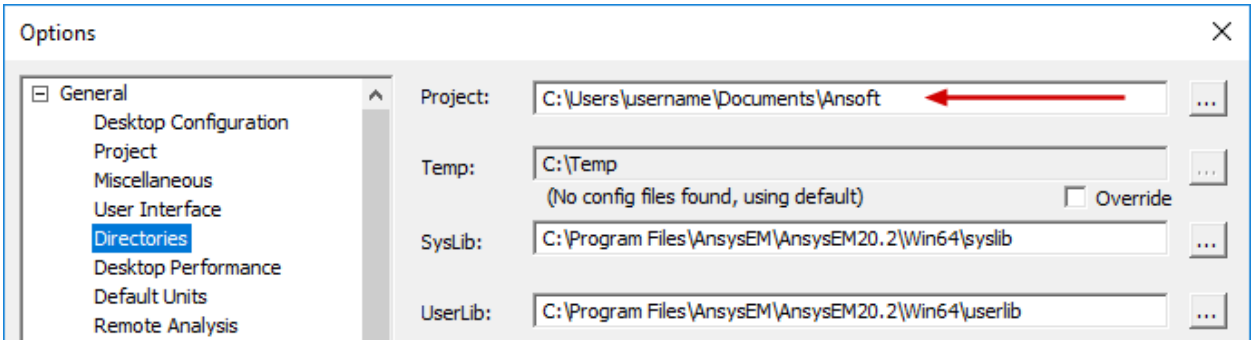
b. Skip to **step 6**.

4. Otherwise, if the EM loss data is to be imported from a different project, clear **Use This Project**.

   The Mechanical design saves the relative path to the source project in one of two ways. Choose one of the following two options under *Save source path relative to*:

   - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



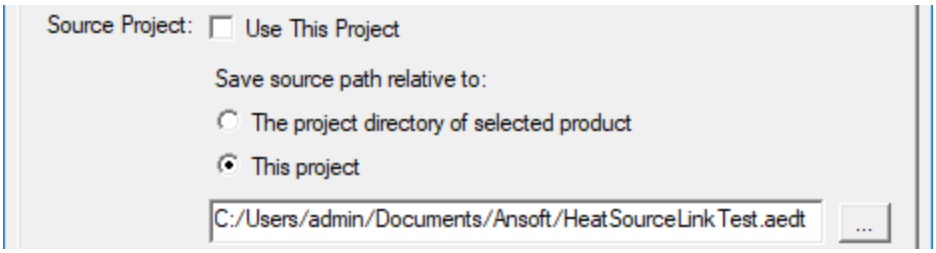
- **This project:** Save source project path relative to the target project location.

5. Click the ellipsis button (**...**).

   The *Open* dialog box appears. Then:

   a. Locate the source project file and select it.
   b. Click **Open**.

   The selected project is listed in the *Setup Link* dialog box:



6. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:

> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

7. Optionally, select one or both of the following two options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you will be performing a two-way coupled simulation, you must select both of these options.

- **Simulate source design as needed:** Select this option if you want an updated solution to be linked to the *target* design when the *source* design has been modified.

  > **Note:**
  >
  > Certain changes to the source design may not invalidate its solution. In these cases, a new source solution will only be run if the previous one did not converge. To achieve convergence, you may have to increase the specified maximum number of passes or relax the convergence criterion in the source solution setup.

  (Also, see Cached Link Data and Solution Invalidation below for additional considerations).

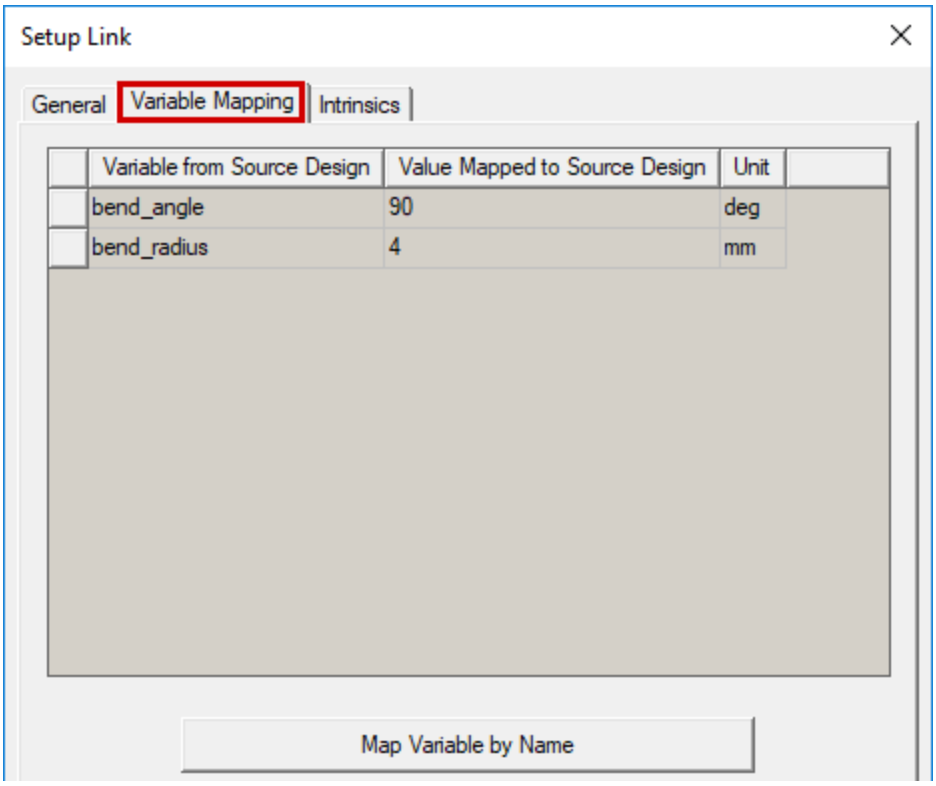
- **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option is not applicable if the source design is in the same project as the *target* design.

  > **Note:**
  >
  > In the *Extractor mode*, the source project will be saved upon exit. *Extractor mode* means that the source project is opened during the link solely for the purpose of solving.

8. Click the **Variable Mapping** tab.

   If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

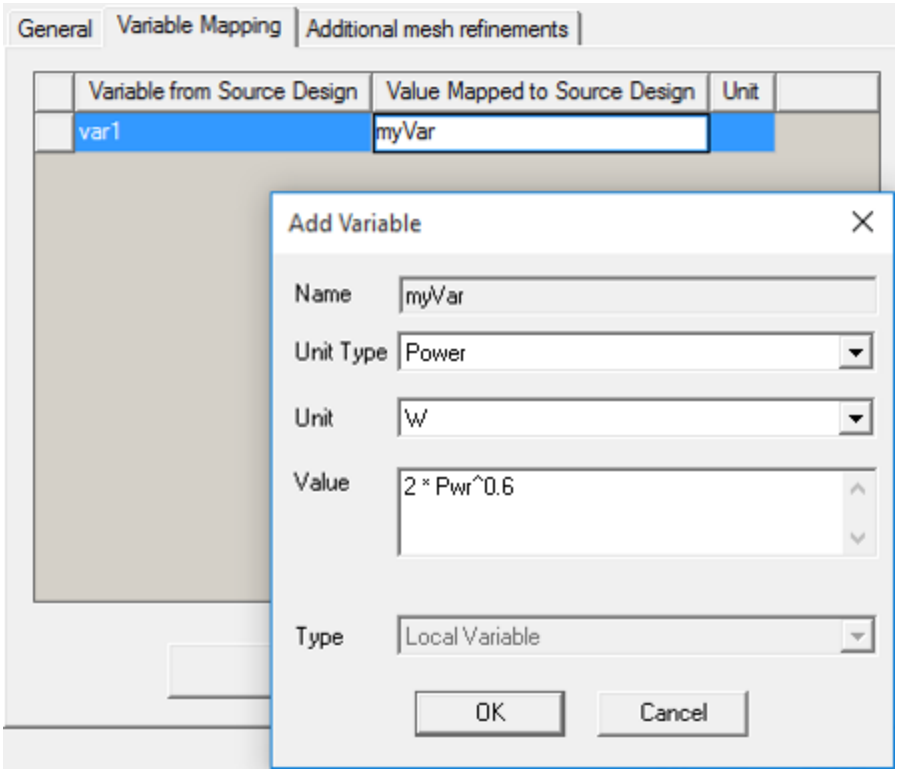


The values passed back to the source design from the mechanical design (second column) can either be constants (numeric values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

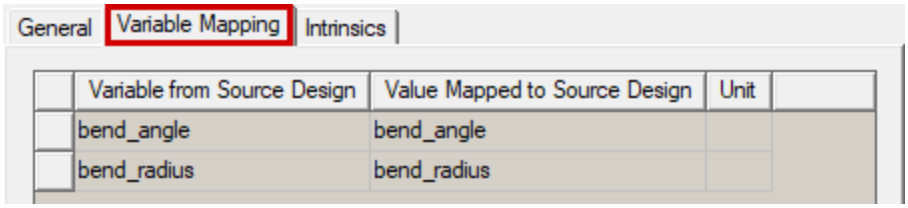
Note:

Automatic variable creation is supported in the *Variable Mapping* tab . You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the two coupled analyses.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

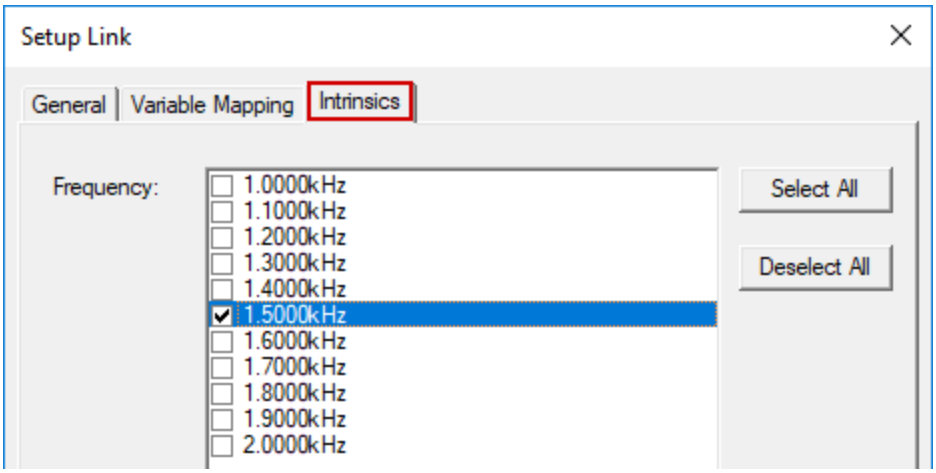
Important:

- Variables can only be mapped by name if they already exist in the Mechanical design. When you copy and paste geometry from the source, or import it from the clipboard, variables used in the definition of the geometry automatically come into the Mechanical design. However, for non-geometry-related variables (such as a variable voltage or wattage applied to an excitation in the HFSS *Edit post process sources* dialog box), you would have to manually define the variable within the Mechanical design.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

Mapping variables to the source design by name is useful for Optimetrics analyses (for cases where changing the value of a variable affects both the target design and the EM losses from the source design). You can create an Optimetrics setup in the target design that sweeps both the target and source designs.

10. Source designs may provide EM losses at multiple frequencies when they are set up appropriately. In such cases, click the **Intrinsics** tab and select one or more frequencies from the **Frequency** list. Your selection defines the solution frequency (or frequencies) in the source design from which you wish to import the EM losses as a source of heat in the Mechanical – Thermal design. (See *Important* note below.)

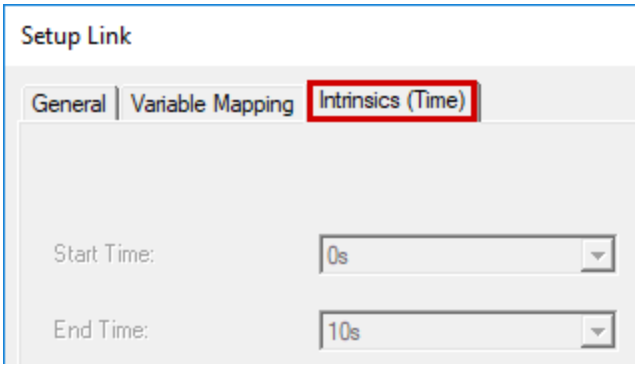
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Multiple frequencies are available from HFSS designs (**Modal** and **Terminal** solution types only) and Maxwell 3D designs (**Eddy Current** solutions), providing that you include a frequency sweep and choose to save the fields for all frequencies.

> **Important:**
>
> When you select multiple frequencies, the EM losses are *additive*. That is, the excitations for each frequency are considered to be occurring simultaneously. Therefore, the thermal solution and any field reports or overlays you create are based on the *total combined EM losses of all the selected frequencies*.
>
> To see the thermal effects of a single excitation frequency, solve the Mechanical – Thermal design with only one frequency at a time selected under the *Intrinsics* tab.

11. For Maxwell transient solutions (*Magnetic* or *Electric*), click the **Intrinsics (Time)** tab and select the desired **Start Time** and **End Time** from the drop-down menus.

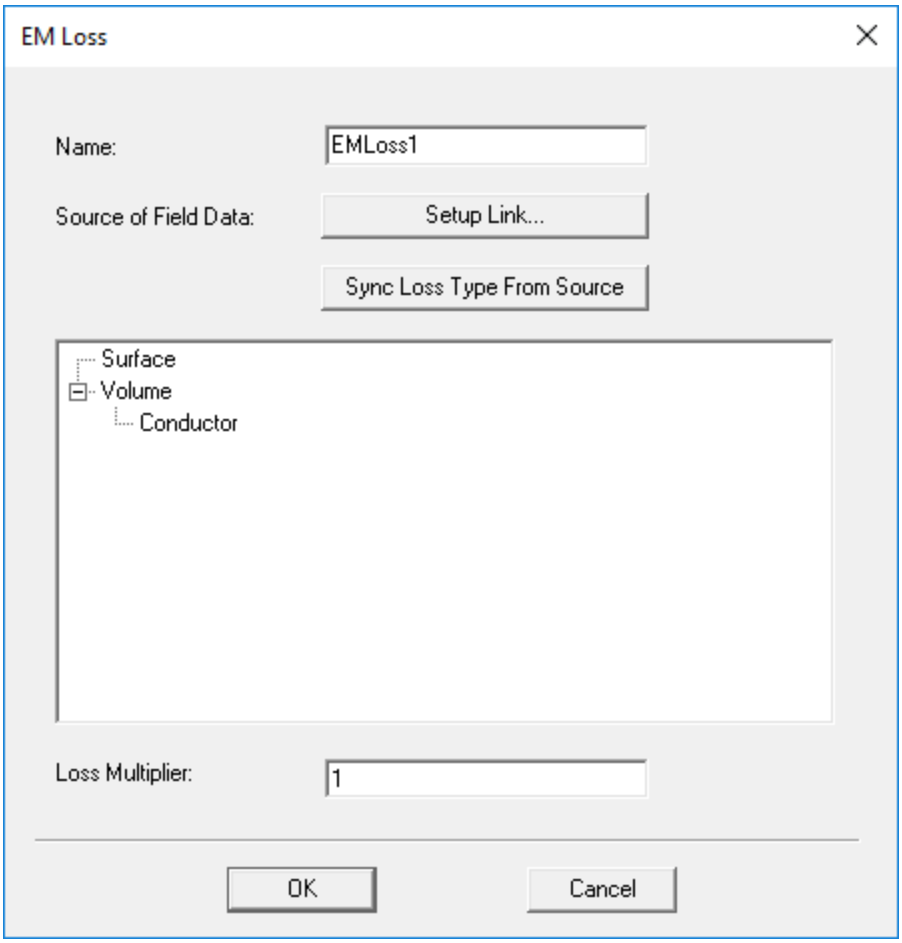


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The Mechanical solution is based on the total energy loss in the source design during the period you specify in the Intrinsics (Time) tab divided by the same specified time period. This quotient is the average power loss that will be applied to the target thermal solution as a constant EM Loss.

12. Click **OK** to close the *Setup Link* dialog box.

    The *EM Loss* dialog box appears:



13. Optionally, specify a customized **Name** for the EM loss excitation.
14. Optionally, specify a **Loss Multiplier** other than 1 (<1 to decrease the imported loss, >1 to increase it).
15. As needed, drag and drop the listed objects to place them under **Surface** or **Volume**, specifying the type of EM loss.

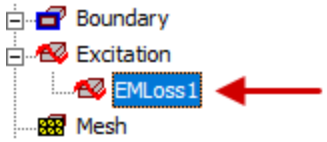
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The program will attempt to determine the appropriate loss type (*Surface* or *Volume*) based on the source design's object properties. Verify that the automatic assignments are correct and modify them as needed. You can click and drag objects from one loss type to the other.
- If you've changed the EM loss type of any object between *Volume* and *Surface* and wish to restore the automatic assignments, click **Sync Loss Type From Source**. This command is also useful for reevaluating the loss type after the source design has been modified.

16. Click **OK** to close the *EM Loss* dialog box.

    The EM loss assignments appear under *Excitation* in the Project Manager.



Note:

- If you double-click the *EM Loss* entry in the Project Manager (or right-click it and choose **Properties**), The *EM Loss* dialog box will reappear for editing the excitation.
- To edit the EM loss *link*, click **Setup Link**, which will open the *Setup Link* dialog box. When revisiting this dialog box, it will be in *View Only* mode. The settings will be displayed but unavailable for editing. Select the **Edit Link** option under the *General* tab to return the dialog box to the editing mode.
- If the properties of objects in the source design have been edited, you can click **Sync Loss Type From Source** to reevaluate the loss type categorization.

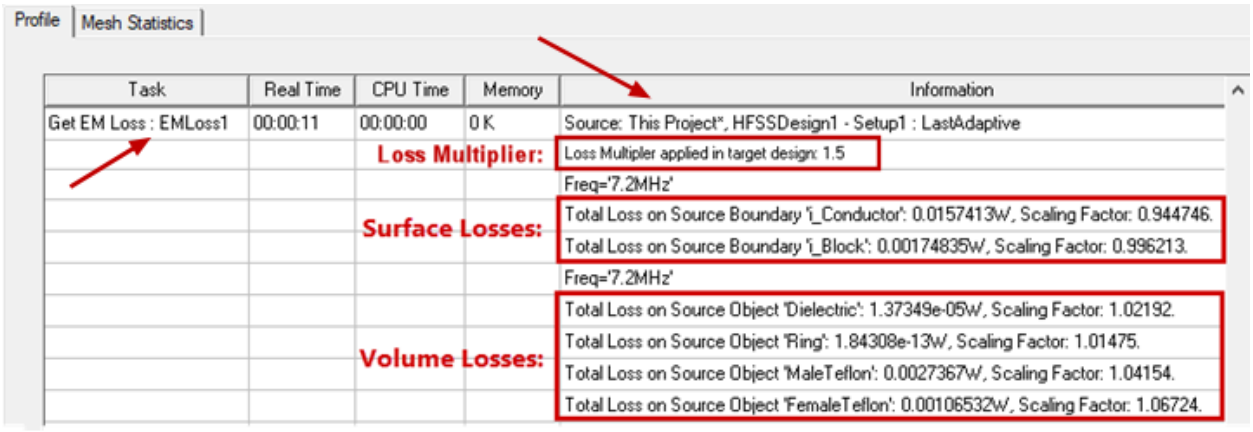
Verifying EM Loss Linking

You can verify that EM loss mapping has been performed after the source and target setups have been analyzed:

- Under *Analysis* in the Project Manager, right-click the target setup and choose **Profile** from the shortcut menu.

  The retrieval of external EM loss data is reported in the simulation profile. Surface and volume losses are grouped separately:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Best Practices:

The application does not enforce an identical geometry limitation between the source and target designs for EM loss linking. The EM loss mapping scheme provides some flexibility. However, your results may be unpredictable and incorrect if there is not a very close correlation between objects with EM losses in the source design and corresponding objects with imported EM losses in the target design. Follow these best practice guidelines for accurate and reliable results:

- Though not a requirement, it is best to keep the number of EM loss objects in the source and target designs the same. Doing so keeps the designs straightforward and helps to minimize the likelihood of errors.
- The geometry (volume, shape, orientation, and absolute location in global coordinates) of each corresponding pair of source and target objects should be consistent between the source and target designs.
- The mesh quality should be comparable between the source and target objects.

  The source and target meshes do not have to be identical. However, you can take advantage of an imported mesh to reduce solution time and to ensure identical geometry, which is enforced for mesh linking. (See the Mesh Linking topic for more information. For instructions on creating identical geometry in the target design, see the Mesh Linking Requirement section of the same topic.)

> **Tip:**
>
> EM loss mapping is performed on the basis of the locations of the element centroids in the source and target objects. Therefore, you do not have to specify the source object that is associated with each target object. For this reason, you can assign EM loss excitation to multiple target objects in a single operation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cached Link Data and Solution Invalidation

The EM losses from the source design are cached in the target design after its simulation. You can remove this cached data (instructions below).

If you've updated the source design and want the imported EM losses in the target design to reflect the changes in the source project, you need to clear the linked data. Doing so invalidates the previous solutions. Clear the linked data using one of the following two methods:

Method 1: (to clear linked data and/or solutions)

1. Using the menu bar, click **Mechanical** > **Results** > **Clean Up Solutions**.

   The *Clean Up Solutions* dialog box appears.

2. In the Solutions section of the dialog box, choose one of the following sets of options:
   - Select the **Linked Data Only** option.
   - Select the **Fields and Meshes** and **Include Linked Data** options.
   - Select the **All Solutions** and **Include Linked Data** options.
3. Click **Do Deletions**.

Method 2: (to clear linked data only)

- Under *Analysis* in the Project Manager, right-click a solution setup and choose **Clear Linked Data** from the shortcut menu.

> **Note:**
>
> - Current EM losses and previously solved results are invalidated when the cached data is cleared.
> - EM losses are cached for each solution setup with an EM loss excitation specified.
> - The cached data is automatically cleared when the target Mechanical design is edited in a way that invalidates its results (such as adding, removing, or modifying geometry or changing the solution setup to import a mesh).
> - Cached link data in the target (Mechanical) design is *not* automatically removed when the source design is edited.

Force

A *Force* excitation applies a resultant force in any desired vector direction. Force excitations can be uniform (user-specified magnitude) or non-uniform (force distribution imported from an electromagnetic simulation solution).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **uniform** forces: Specify X, Y, and Z components of force to produce the desired resultant force and direction. Global and user-defined coordinate systems are supported. The specified force components are applied to each face in the selection set. For example, if you assign a 10 newton force in the X-direction with three faces selected, the model receives a total force of 30 newton in the X-direction.
- For **non-uniform** forces: Set up a link to a source solution. Electromagnetic force results (specifically surface or volume force density) will be imported from the source design to the target Mechanical–Structural design.

You can assign *Force* excitations to the following entity types:

- **Faces** of solid objects
- Solid **objects** (only for the non-uniform option with a Maxwell 3D source design)

The faces to which you assign a *Force* may be of any shape or orientation, flat or curved.

The following criteria apply for EM force source designs:

- The geometry of EM force source objects in a source design must be the same as the target design objects that will receive the imported forces whether surface or volume-based. Currently, a partial model cannot be used as a source of forces for a full Structural target model.
- HFSS solutions only produce surface force density results. Therefore, when linking to an HFSS solution, the Force excitation must be assigned to **faces** of the target Structural model.
- Maxwell 3D solutions include both surface force density and volume force density results. Therefore, Structural Force excitations linked to Maxwell source designs can be assigned to either solid **objects** or **faces**.

> **Important:**
>
> Non-uniform (imported) Force excitations are not currently supported for Maxwell 3D source designs in which the solution type is **Magnetic: Eddy Current**. Mechanical–Structural solutions support non-uniform Forces from all other Maxwell 3D solution types.

How to Assign a Force Excitation:

1. Using the *Face, Object,* or *Multi* selection mode, select one or more solid **objects** and/or **faces**, as applicable.
2. Use *one* of the following three methods of accessing the *Force* dialog box:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click in the Modeler window and choose **Assign Excitation** > **Force** from the shortcut menu.
- Right-click **Excitations** in the Project Manager and choose **Assign** > **Force**.
- From the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Force**.

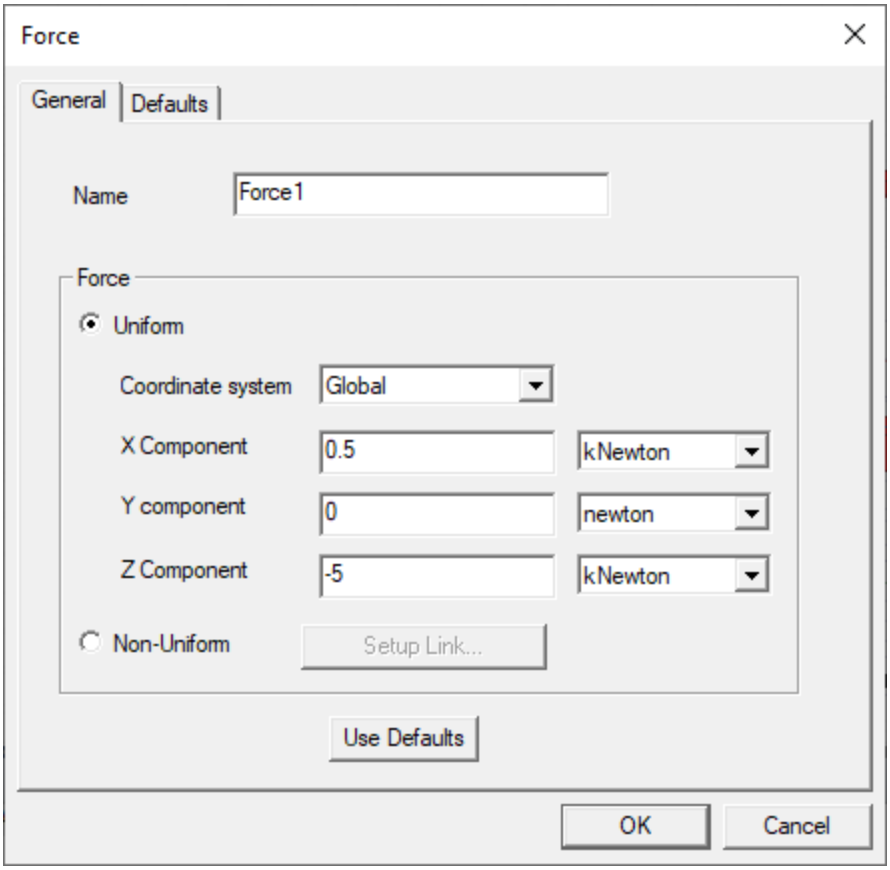
3. Select one of the following *Force* types:
   - **Uniform:** A constant, user-specified force will be applied at all selected faces.
     a. From the **Coordinate System** drop-down menu, choose the CS on which the force components will be based.

        The menu will list the Global CS and any user-defined CSs that exist in the design. *Global* is the default selection.

     b. Type the force component magnitudes in the **X Component, Y Component,** and **Z Component** text boxes.
     c. Choose the desired force units from the drop down menus to the right of each component text box. The following units are supported:

        fNewton, pNewton, nNewton, uNewton, mNewton, newton, kNewton, megNewton, gNewton, dyne, PoundsForce, and kp

        For zero-components, the units choice doesn't matter.



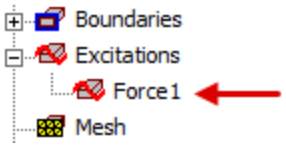
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Non-Uniform:** Surface force density results will be imported from an HFSS or Maxwell 3D design, or volume force density results will be imported from a Maxwell 3D design.

  Follow the procedure, How to Setup the Force Link in the *Setup Link* subtopic before continuing to step 4 of this procedure.

4. Optionally, change the **Name** of the excitation. (The default name is Force*x*, where *x* is a number incremented for each force you add.)
5. Click **OK**.

   The force appears under *Excitations* in the Project Manager:



Setup Link (Force)

When importing *Non-Uniform* force density results from an HFSS or Maxwell 3D design to a *Force* excitation in a Mechanical–Structural design, use the *Setup Link* dialog box to specify the source project, source design, and other linking options.

This feature makes it convenient to determine structural stresses and deformation due to electromagnetic forces, which can be determined using other simulation design types.

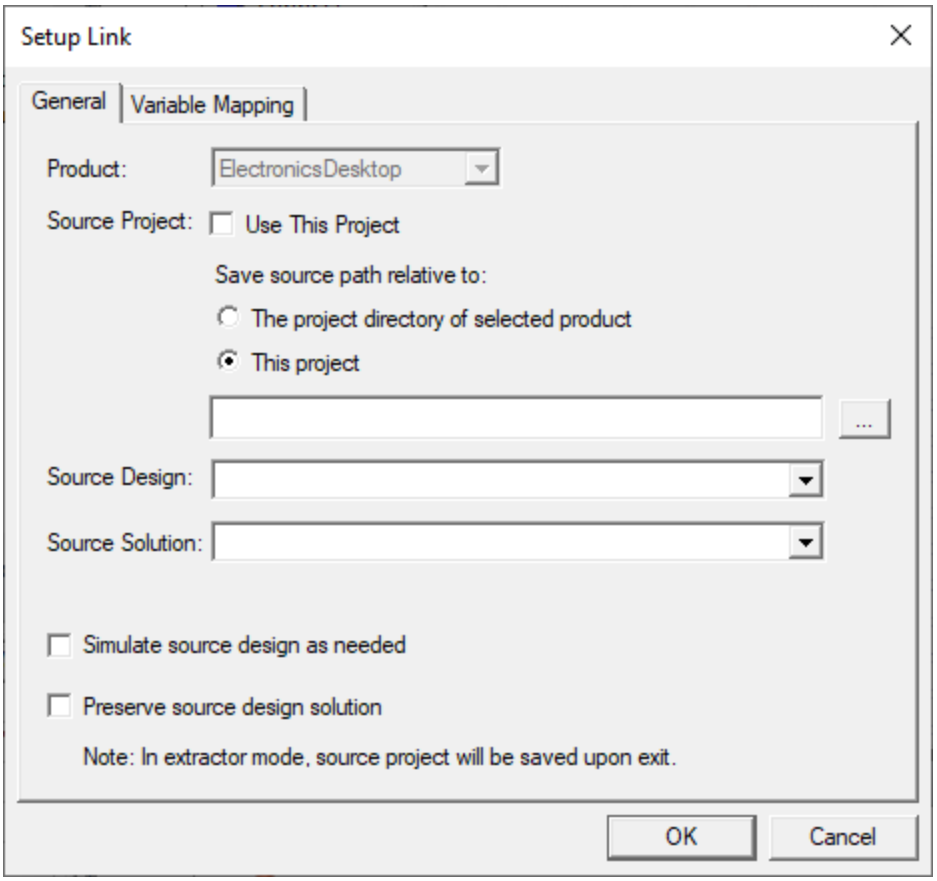
Important:

- The Mechanical design objects that are imported force targets should have geometrically identical counterparts in the HFSS or Maxwell 3D source design. Identical geometry ensures that mapping of force results from source to target entities is performed successfully and accurately. Additionally, the surface mesh size should be similar between the two designs. Optionally, you can take advantage of Mesh Linking between Mechanical and HFSS or Maxwell 3D designs, for identical meshes.
- Any air or vacuum regions in a source design must be omitted from the Mechanical–Structural design. Air and vacuum are not valid structural materials. Also, if you were to substitute a structural material with significant stiffness for air or vacuum, the region around the other solid parts would interfere with the deformation of the solid objects in a Mechanical–Structural solution.
- The link between source and product designs is a one-way coupling operation. Changes you make in the target Mechanical design and the results of the simulation have no effect on the source design.

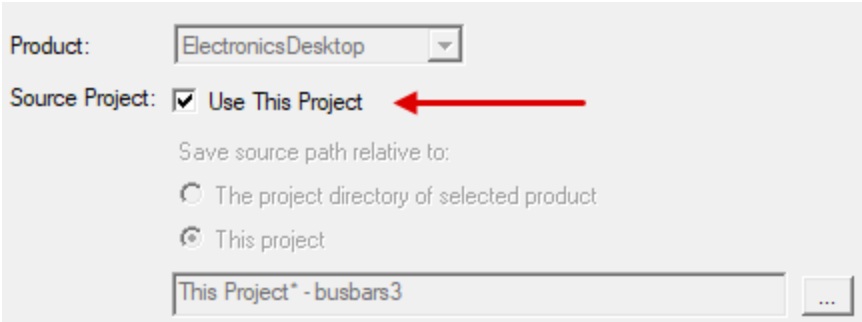
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How to Set Up the Force Link

1. Follow steps 1 through 3 of the How to Assign a Force Excitation procedure, selecting the **Non-Uniform** option in the fourth step. The *Setup Link* dialog box appears:



2. If the data is to be imported from a design in the same project, complete the following substeps. Otherwise go to **step 3**:
   a. Select **Use This Project**.

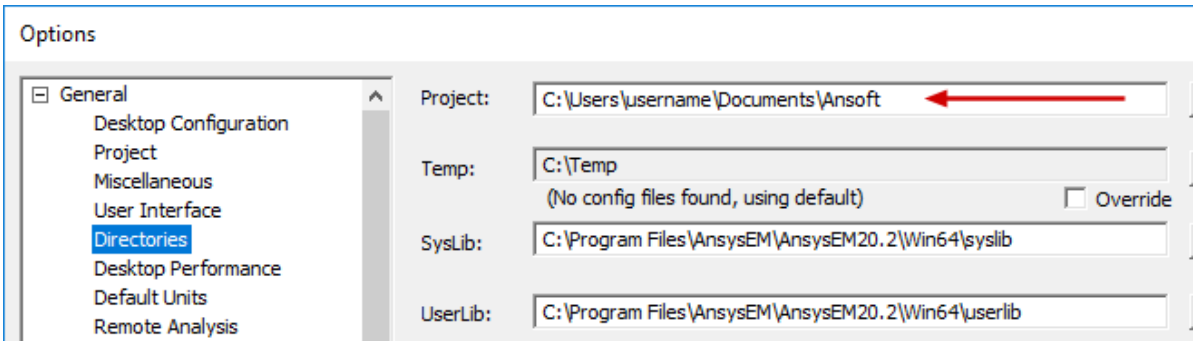


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

b. Skip to **step 4**.

3. Alternatively, if the data is to be imported from a design in a different project, clear the **Use This Project** option and complete the following substeps:
   a. The Mechanical design saves the relative path to the source project in one of two ways. Choose one of the following two options under *Save source path relative to*:
      - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):



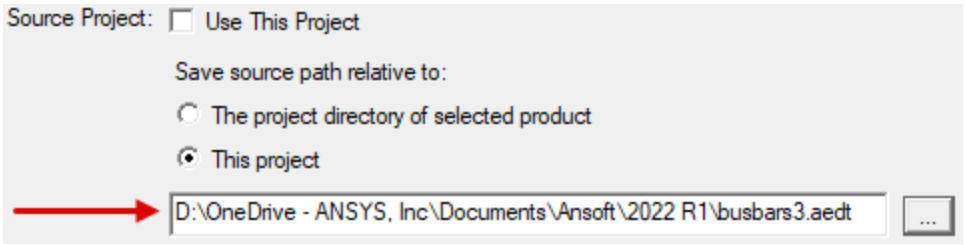
      - **This project:** Save source project path relative to the target project location.

   b. Click the ellipsis button (**...**) to the right of the *Source Project* text box.

      The *Open* dialog box appears.

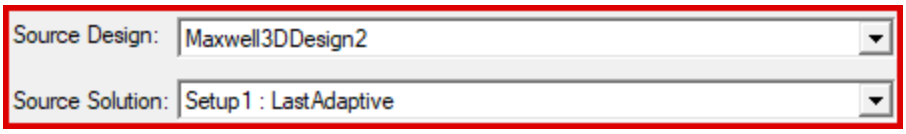
   c. Locate the source project file and select it.
   d. Click **Open**

      The selected **Source Project** is listed in the *Setup Link* dialog box:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:

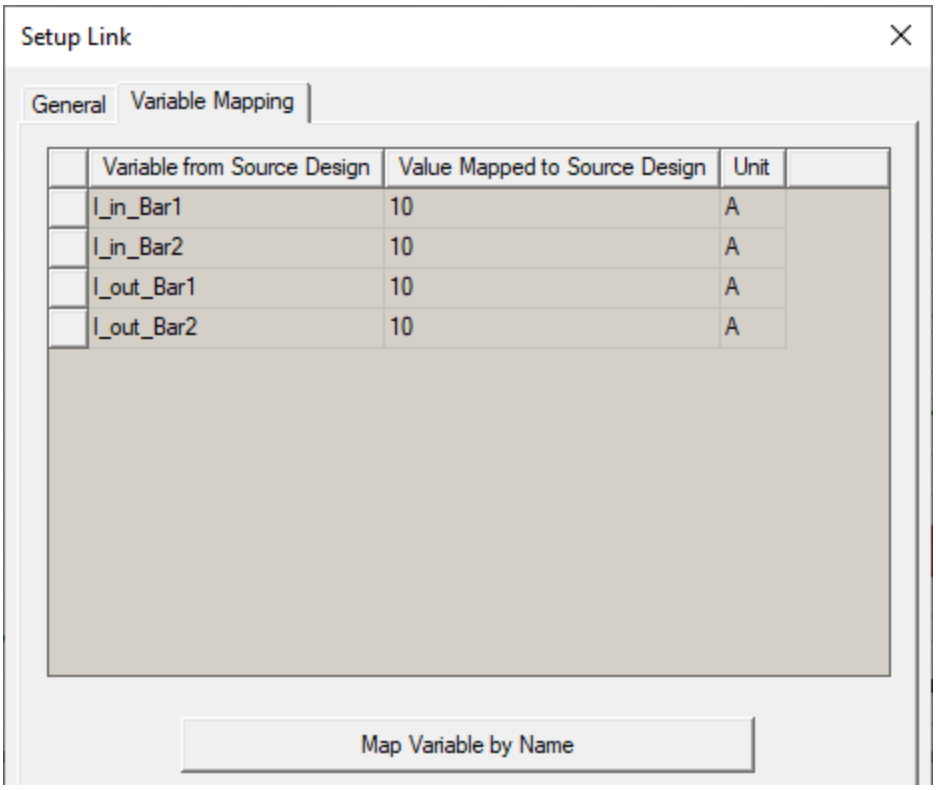


> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

5. Optionally, select one or both of the following two options:
   - **Simulate source design as needed:** Select this option if you want an updated solution to be linked to the *target* design when the *source* design has been modified.
   - **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option has no effect if the source design is in the same project as the *target* design.
6. Click the **Variable Mapping** tab.

   If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:

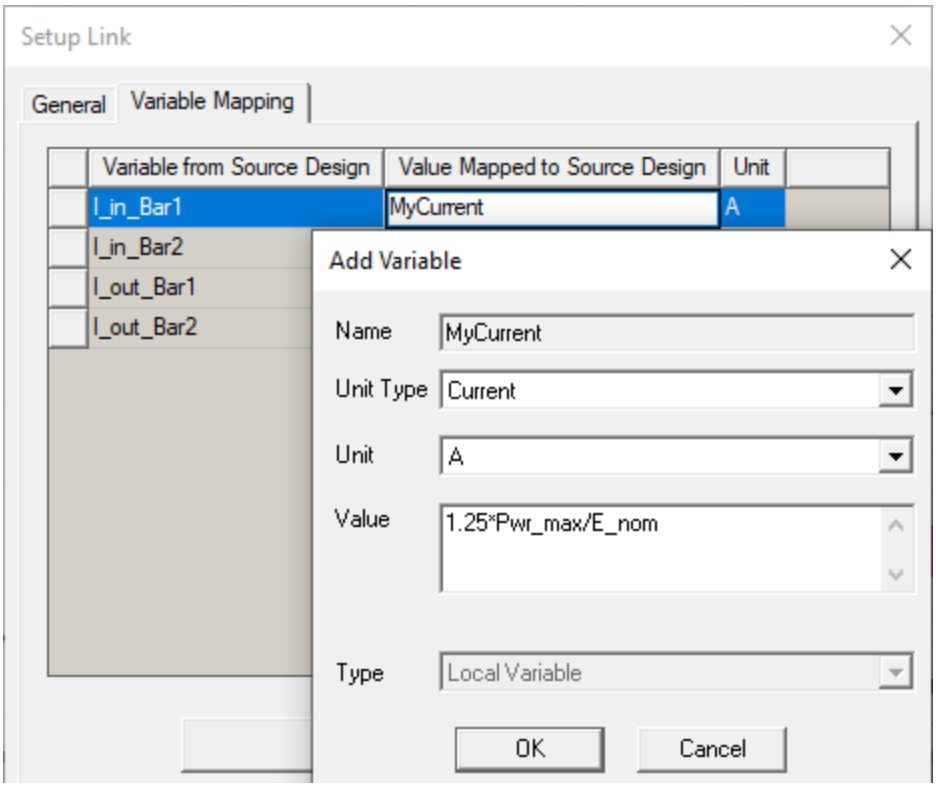
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



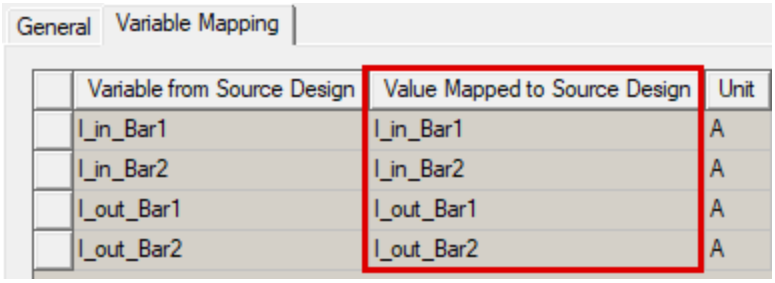
The values passed back to the source design from the mechanical design (second column) can either be constants (numeric values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

> **Note:**
>
> Automatic variable creation is supported in the *Variable Mapping* tab . You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the linked analyses.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

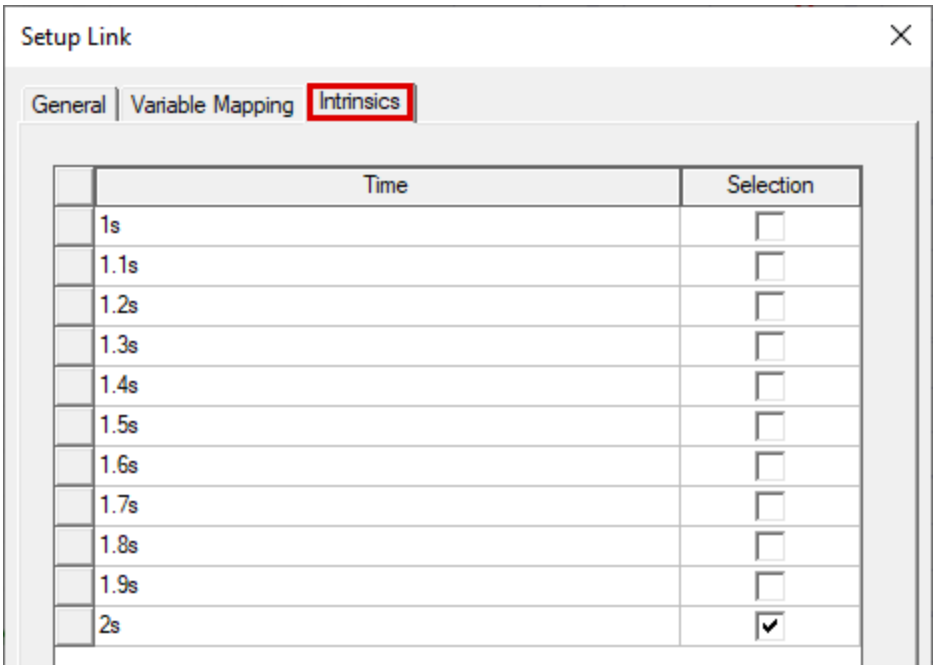
Important:

- Variables can only be mapped by name if they already exist in the Mechanical design. When you copy and paste geometry from the source, or import it from the clipboard, variables used in the definition of the geometry automatically come into the Mechanical design. However, for non-geometry-related variables (such as a voltage applied to an HFSS source excitation or a current assigned to a Maxwell conductor), you would have to manually define the variable within the Mechanical design.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

Mapping variables to the source design by name is useful for Optimetrics analyses (for cases where changing the value of a variable affects both the target design and the film coefficients from the source design). You can create an Optimetrics setup in the target design that sweeps both the target and source designs.

8. For source design with transient solutions or frequency sweeps, the **Intrinsics** tab will list each transient event time point or each frequency in the sweep. Select **only one** time or frequency as the basis of the imported forces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Currently, multiple time or frequency selections under *Intrinsics* are not supported for EM force coupling.

9. Click **OK** to accept the link settings and close the *Setup Link* dialog box.
10. Continue the force excitation setup at **step 4** of the *Force* topic.

> **Note:**
>
> If you later revisit the *Force* dialog box to verify or modify its properties, click the **Setup Link** button under the *General* tab to return to the *Setup Link* dialog box. This dialog box will reopen in **View Only** mode. To modify the properties, first select the **Edit Link** radio button.

Heat Flux

This excitation specifies a rate of heat flowing through object faces and is applicable to the following entities:

- **Faces** of solid or shell objects
- **Objects** (solids or shells)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Assigning a Heat Flux excitation to an object is equivalent to assigning the excitation to all faces of the object.

You can specify heat flux in either of two ways—*Total Power* or *Surface Flux*. Total Power is specified in units of Power, such as watts (W) or Btu_per_hr. See the *Important* note below to understand what happens when you assign Total Power to multiple faces and/or objects in a single excitation.

Surface Flux is a uniformly distributed heat flux, and it is specified in units of Irradiance (power per unit area), such as KW_per_m2 (which is, kilowatts/m^2).

The *Total Power* or *Surface Flux* specified for a *Heat Flux* excitation may be positive or negative:

- A positive value adds heat to the model (heat flows into the body at the selected face or faces).
- A negative value removes heat from the model, cooling it (heat flows out of the body at the selected face or faces).

> **Important:**
>
> - If you assign a *Total Power* heat flux to multiple faces, the specified power is applied to each face (not distributed over the selected faces). Therefore, the total combined power added to, or removed from, the model is equal to the specified power times the number of faces in the selection set.
> - If you apply a heat flux excitation to objects, it is equivalent to selecting every face of each object. When using the *Total Power* option, the combined power applied to the model is equal to the specified power times the total number of faces comprising the selected objects.
> - You can use the **Select** > **Multi** mode to assign heat flux to a combination of objects and faces in a single operation, if desired. The preceding two rules apply for mixed selections too.
>
> See the examples in step 3 of the following procedure.

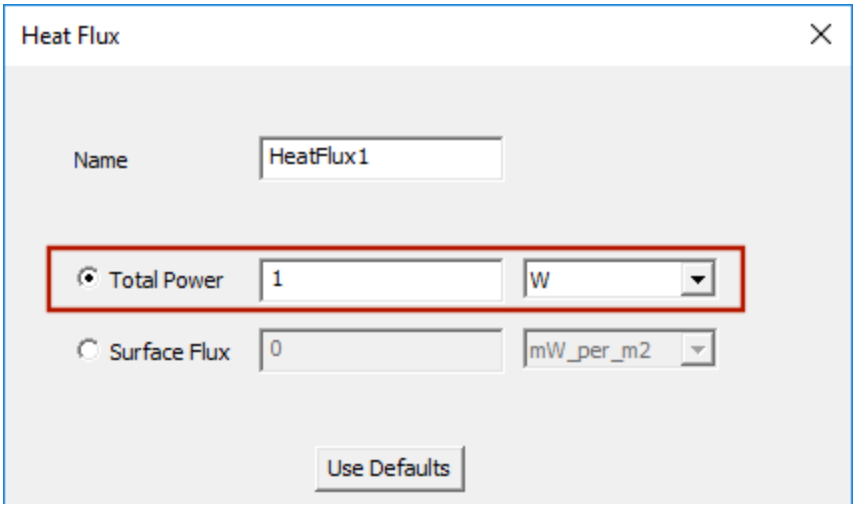
How to Assign a Heat Flux Excitation:

1. Select one or more faces, either graphically or using the **Select by Name** command on the **Draw** ribbon tab. In either case, the *Face* selection mode must be active.
2. Use *one* of the following three methods of accessing the *Heat Flux* dialog box:

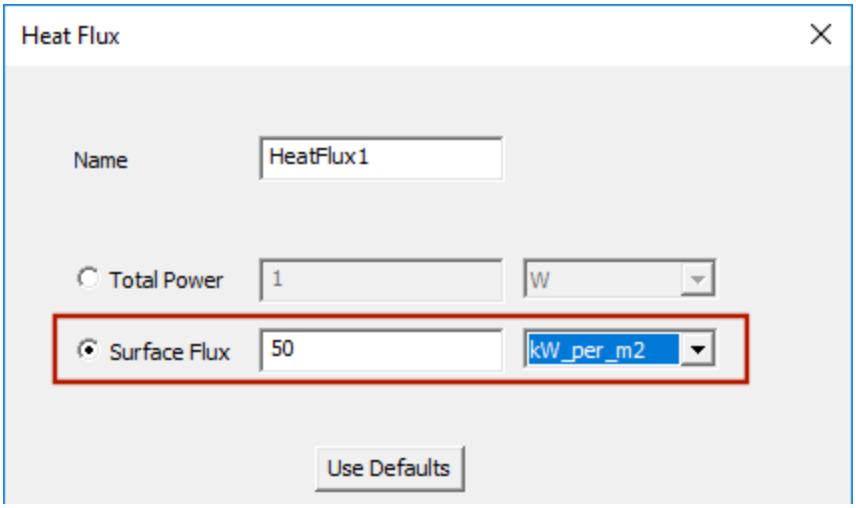
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Heat Flux**.
- Right-click in the Modeler window or right-click a selected item in the History Tree, and choose **Assign Excitation** > **Heat Flux** from the shortcut menu.
- Right-click **Excitation** in the Project Manager and choose **Assign** > **Heat Flux**.

3. Choose the desired heat flux type:
   - **Total Power:** Specify the magnitude of the total thermal power to be applied to *each* selected face and/or each face of any selected objects. Choose the units of power from the adjacent drop-down menu:



   - **Surface Flux:** Specify the uniform power per unit surface area to be applied to all selected faces and/or all faces of any selected objects. Choose the units of surface flux from the adjacent drop-down menu:

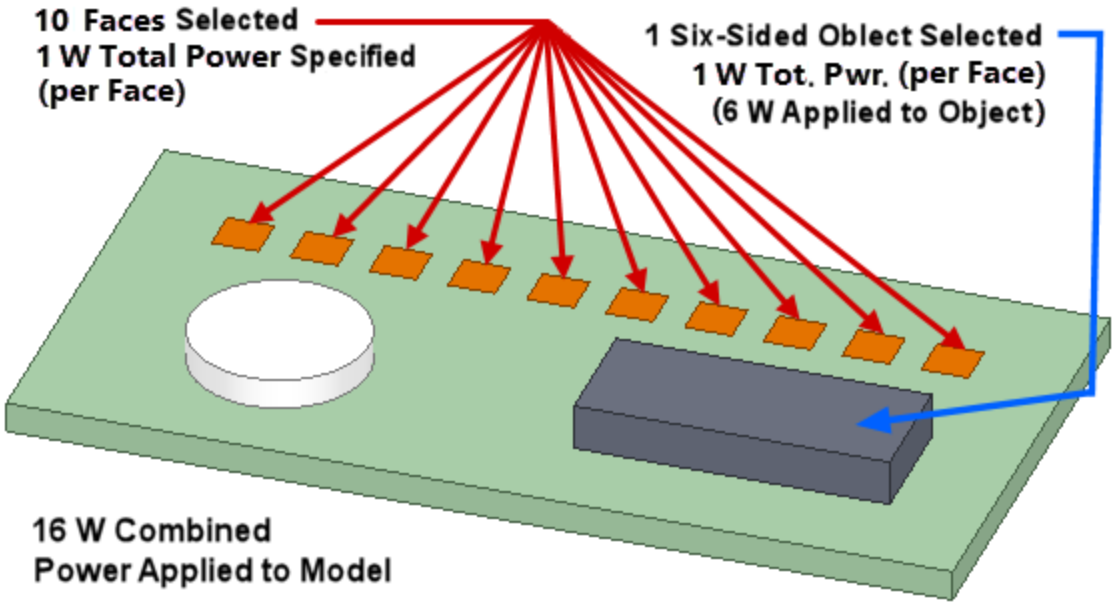


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

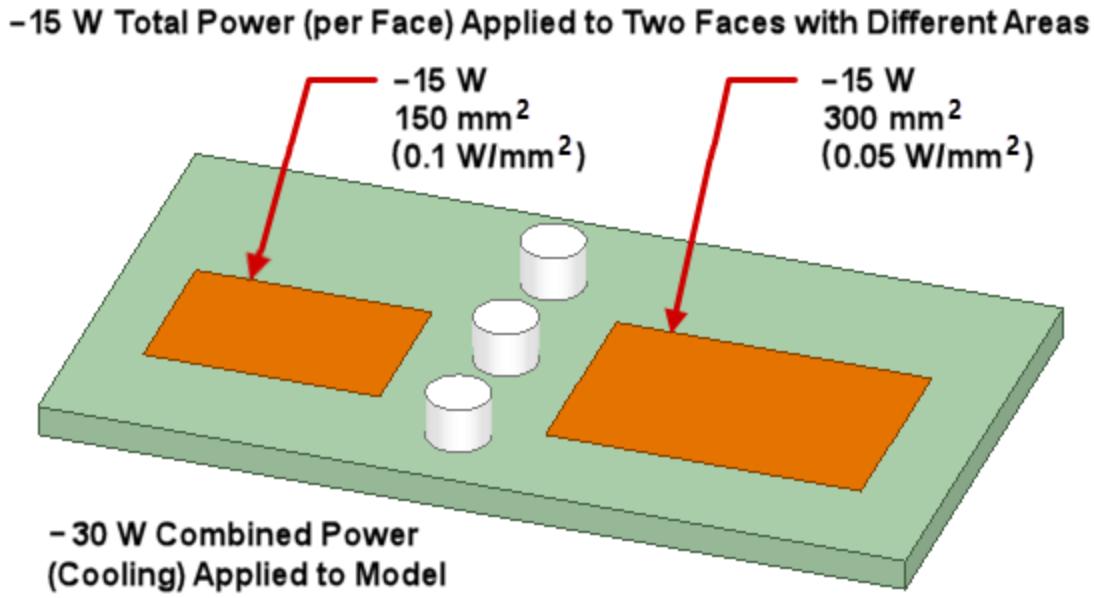
Regardless of whether you specify *Total Power* or *Surface Flux*, the value can be positive or negative and can be numeric, a variable, or an expression (consisting of constants, variables, and mathematical operators). Additionally, for transient thermal solutions, the value can vary with time by using a time-dependent dataset reference.

Example 1: Assume that ten faces and one six-sided object (a box) are selected. If you specify 1 W of *Total Power*, then each face will receive 1 W of heat. The model will receive a combined total thermal power of 16 W (1W x 16 faces). Because the power per face is constant but the areas of the faces differ, the heat flux density (that is, the power per unit area) will vary among the faces.

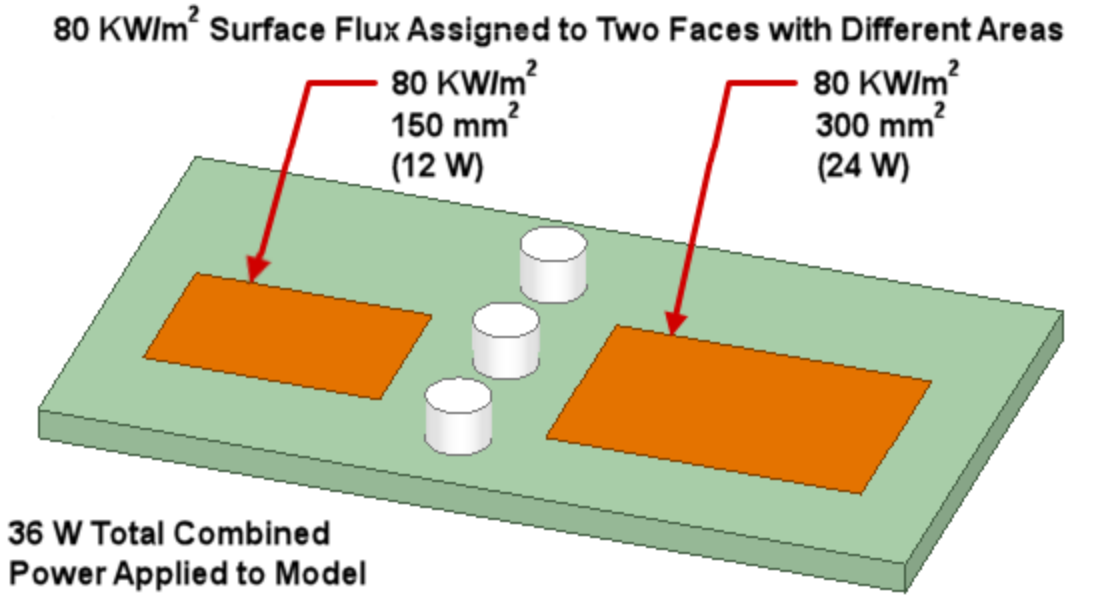


Example 2: This example shows the total combined power and the equivalent heat flux density when two faces with different areas are selected. One face has an area of 150 mm^2, and the other 300 mm^2. If you specify a *Total Power* of -15 W, the combined thermal power applied to the model is -30 W (-15 W x 2 faces). 15 W of heat is **removed** at each face (that is, the model is cooled) because the specified power is negative. The flux density of the smaller face is -0.1 W/mm^2 (-15 W / 150 mm^2). The flux density of the larger face is -0.05 W/mm^2 (-15 W / 300 mm^2).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



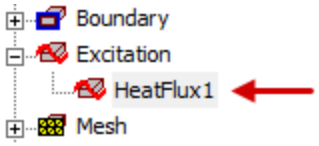
Example 3: This example shows the total combined power and the power per face when two faces with different areas are selected and you choose the *Surface Flux* option. One face has an area of 150 mm^2, and the other 300 mm^2. If you specify a *Surface Flux* of 80 KW/m^2, the combined thermal power applied to the model is 36 W. The thermal power entering the smaller face is 12 W (80,000 W/m^2 * 150 mm^2 /1,000,000 mm^2/m^2). The thermal power entering the larger face is 24 W (80,000 W/m^2 * 300 mm^2 / 1,000,000 mm^2/m^2).



4. Optionally, change the **Name** of the excitation. The default name is HeatFlux*x*, where *x* is a number that increments for each heat flux excitation that you apply.
5. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A heat flux entry appears under *Excitation* in the Project Manager:



Heat Generation

This excitation is applicable only to the following entity type:

- **Solid Objects**

Heat Generation is specified in units of power, such as watts (*W*) or *Btu_per_hr*. The *Total Power* specified for this excitation may be positive or negative. Optionally, for transient solutions, the power can vary with time:

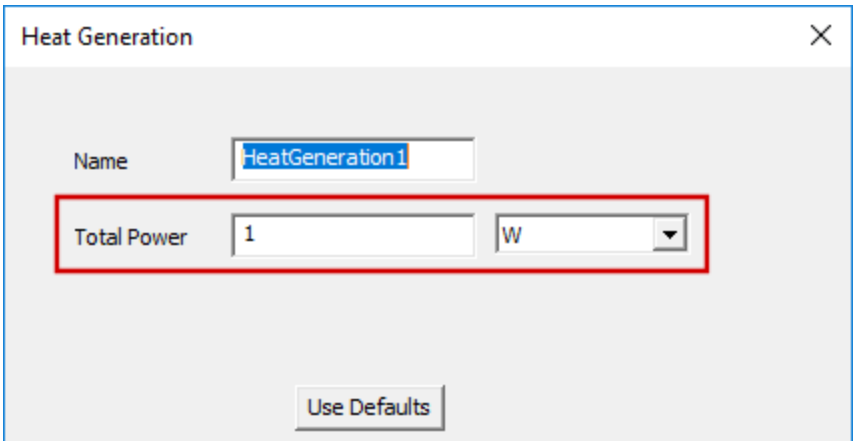
- A positive value adds heat to the model.
- A negative value removes heat from (that is, cools) the model.

> **Note:**
>
> If you select multiple objects, the specified *Total Power* is applied to each of the selected objects, not distributed among them. See the examples in the following procedure.

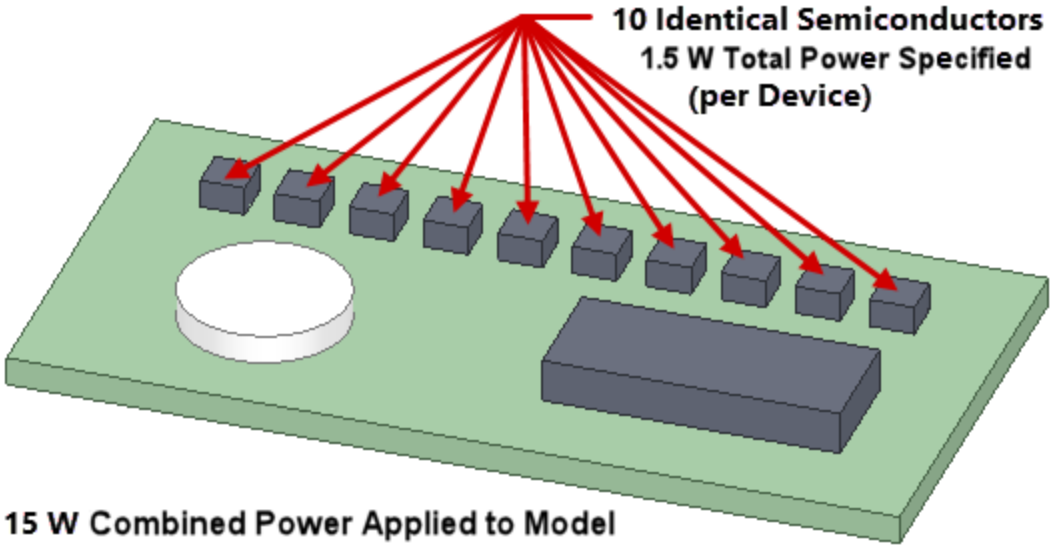
How to Assign a Heat Generation Excitation:

1. Select one or more solid objects, either graphically (in *Object* selection mode) or from the *Solids* branch of the History Tree.
2. Use *one* of the following three methods of accessing the *Heat Generation* dialog box:
   - Using the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Heat Generation**.
   - Right-click in the Modeler window or right-click a selected item in the History Tree, and choose **Assign Excitation** > **Heat Generation** from the shortcut menu.
   - Right-click **Excitation** in the Project Manager and choose **Assign** > **Heat Generation**.
3. Specify the total heat generation per selected object in the **Total Power** text box and select the units from the adjacent drop-down menu. The power value can be positive or negative and can be numeric, a variable, or an expression (with constants, variables, and mathematical operators). Additionally, for transient thermal solutions, the value can vary with time by using a time-dependent dataset reference.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

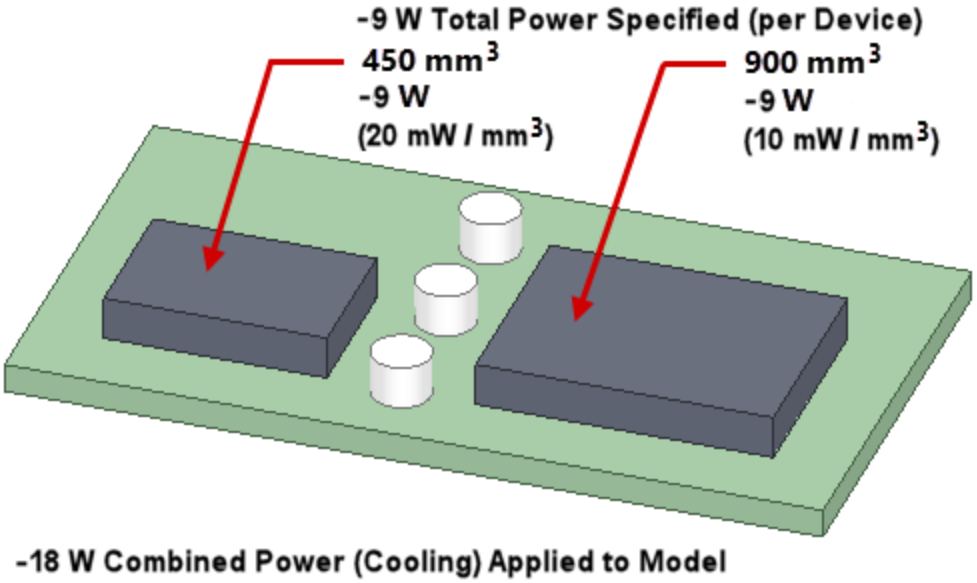


Example 1: Assume that ten components are selected. If you specify 1.5W of *Total Power*, then each object generates 1.5W of heat and the total power input to the model due to this assignment is 15W (1.5W total power x 10 objects). If the ten objects are identical, then the power density (that is, the power per unit volume) is the same for all of them. This example is a typical usage case for applying a heat generation excitation to multiple parts in a single operation.



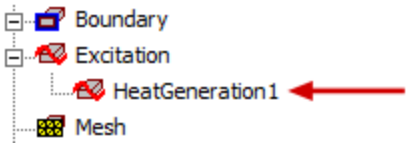
Example 2: Assume that two objects are selected, and their volumes are 450 mm^3 and 900 mm^3. If you specify -9W of *Total Power*, each object **removes** 9W of heat. Heat is removed (that is, the model is cooled) because the specified *Total Power* is negative. The total amount of heat removed due to this assignment is 18W (9W x 2 objects). Since the volumes differ, the power densities differ, with the smaller object having a higher cooling power per unit volume.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, change the **Name** of the excitation. The default name is HeatGeneration*x*, where *x* is a number that increments for each heat generation excitation that you apply.
5. Click **OK**.

   A heat generation entry appears under *Excitation* in the Project Manager:



Pressure

A *Pressure* excitation applies a uniform surface pressure (force per unit area), which acts normal to element faces by default. Positive pressures act into the face (compressive); negative pressures act away from the face (tensile). Alternatively, you can specify X, Y, and Z components to define a pressure that acts in any desired vector direction. Global and user-defined coordinate systems are supported.

You can assign *Pressure* excitations to only the following entity type:

- **Faces** of solid objects

The faces to which you assign a *Pressure* may be of any shape or orientation, flat or curved.

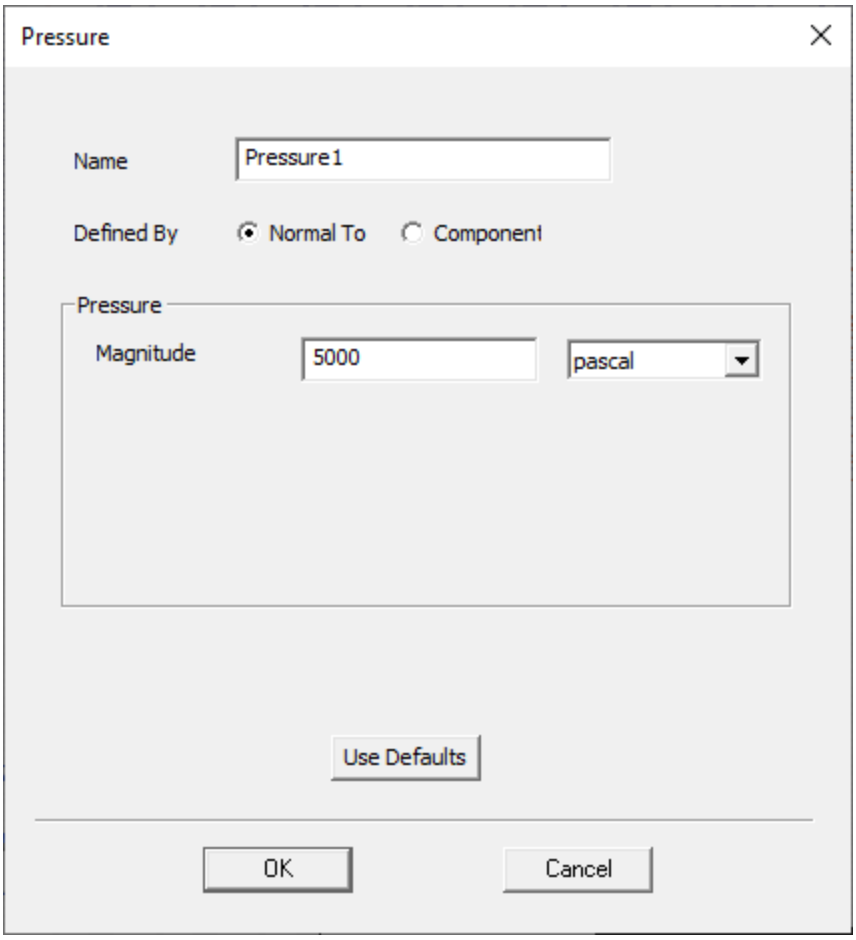
How to Assign a Pressure Excitation:

1. Using the *Face selection* mode, select one or more solid object **faces**.
2. Use *one* of the following three methods of accessing the *Pressure* dialog box:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click in the Modeler window and choose **Assign Excitation** > **Pressure** from the shortcut menu.
- Right-click **Excitations** in the Project Manager and choose **Assign** > **Pressure**.
- From the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Pressure**.

3. Optionally, change the **Name** of the excitation. (The default name is Pressure*x*, where *x* is a number incremented for each pressure you add.)
4. Choose one of the following two options from the **Define By** drop-down menu:
   - **Normal To:** Pressure magnitude acts normal to element faces
   - **Component:** Specify X, Y, and Z components of pressure (global and user-defined coordinate systems are supported)
5. Specify the pressure as follows, depending on your selection in the previous step:
   - If you selected the **Normal To** option, type in the normal pressure **Magnitude**.

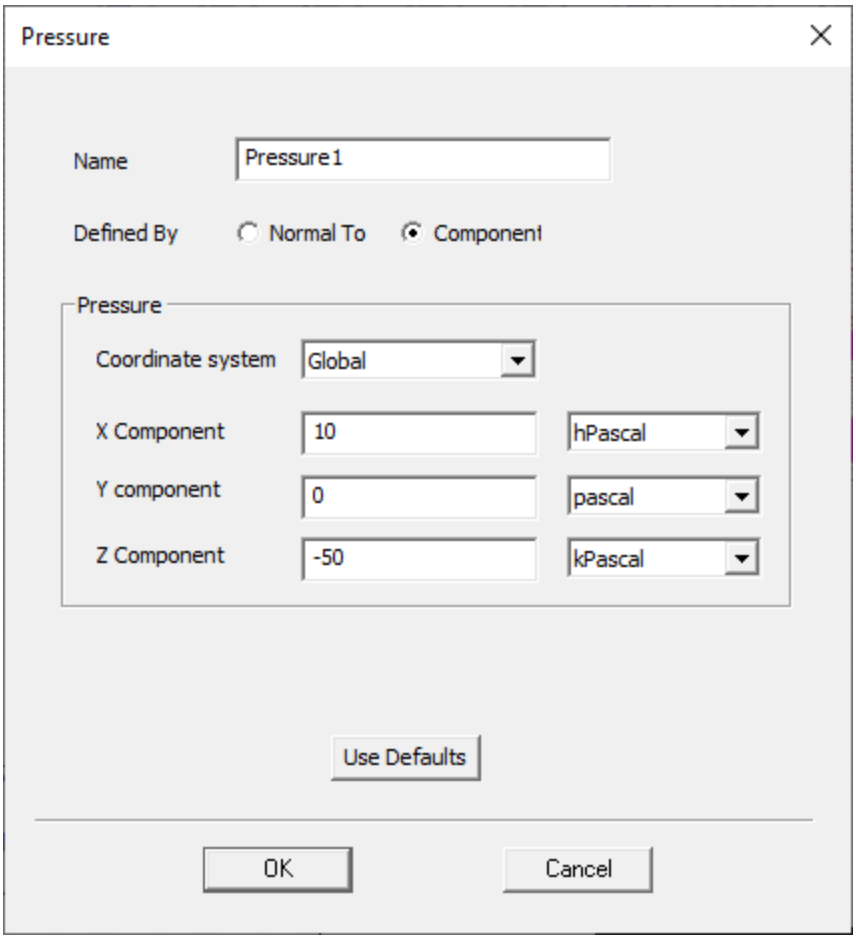


   - If you selected the **Component** option:
     a. From the **Coordinate System** drop-down menu, choose the CS on which the force components will be based.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The menu will list the Global CS and any user-defined CSs that exist in the design.

b. Type the pressure component magnitudes in the **X Component, Y Component,** and **Z Component** text boxes.



6. Choose the desired pressure units from the drop down menu(s) to the right of the *Magnitude* or *X, Y,* and *Z Component* text boxes. The following units are supported:

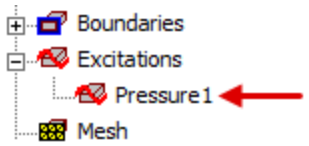
   n_per_meter_sq, n_per_meter_sq, kn_per_meter_sq, megn_per_meter_sq, gn_per_meter_sq, lbf_per_ft2, psi, kpli, megpsi, gpsi, mbar, bar, mmHg, mmh2o, uPascal, mPascal, cPascal, dPascal, pascal, hPascal, kPascal, megPascal, gPascal, stAtm, techAtm, torr, and in_water

   For zero-components, the units choice doesn't matter.

7. Click **OK**.

   The pressure appears under *Excitations* in the Project Manager:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Thermal Condition

This excitation is applicable only to the following entity type:

- **Solid Objects**

A *Thermal Condition* assigns a temperature to a body and can be one of the following two types:

- **Uniform:** A user-defined temperature that is constant throughout the selected objects. You can specify the value numerically, in degrees Celsius (cel), Kelvin (kel, mkel, ckel, dkel), Fahrenheit (fah), or Rankine (rank). You can also select the *EnvTemp* variable, which is the default condition. The EnvTemp value is specified by the *Temperature* value in the *Environment Conditions* tab of the *Mechanical Design Settings* dialog box. This value is the temperature at which all objects are in a stress-free condition. Additionally, you can specify a user-defined design or project variable name for the uniform temperature, or you can enter an expression based on constants and variables.
- **Non-Uniform:** A temperature distribution imported from the results of a Mechanical–Thermal or Icepak simulation of the same geometry. In this case, the *Setup Link* dialog box is used to specify the source design and coupling options.

> **Note:**
>
> Thermal expansion or contraction, and the resultant stresses and deformation, are calculated based on the difference between the assigned thermal condition and **EnvTemp** (the **Temperature** defined in the *Environment Conditions* tab of the *Mechanical Design Properties* dialog box). **EnvTemp** is the temperature at which the material is in a stress-free state, often referred to as the "stress-free temperature."

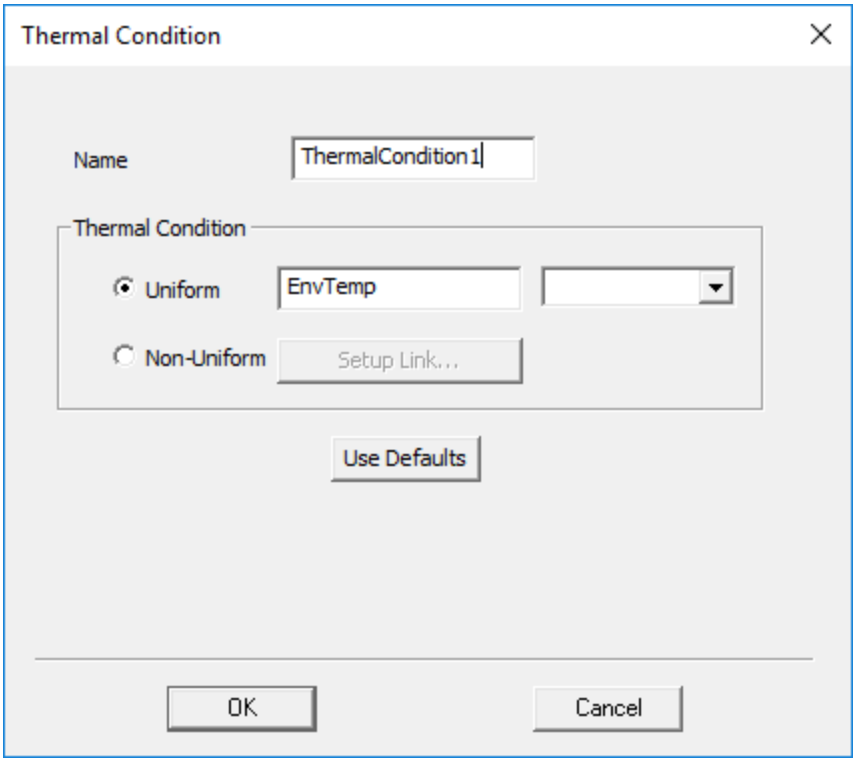
How to Assign a *Thermal Condition* Excitation:

1. Select one or more solid objects, either graphically (in *Object* selection mode) or from the *Solids* branch of the History Tree.
2. Use *one* of the following three methods of accessing the *Thermal Condition* dialog box:
   - Using the menu bar, click **Mechanical** > **Excitations** > **Assign** > **Thermal Condition**.
   - Right-click in the Modeler window or right-click a selected item in the History Tree, and choose **Assign Excitation** > **Thermal Condition** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click **Excitation** in the Project Manager and choose **Assign** > **Thermal Condition**.

The *Thermal Condition* dialog box appears.



3. Choose one of the following two *Thermal Condition* options:
   - **Uniform**, if you want to specify a temperature that is constant throughout the volumes of the selected objects.
   - **Non-Uniform**, if you want to import the resultant temperature distribution from a Mechanical–Thermal or Icepak simulation.
4. If you choose the **Uniform** option, specify the temperature to apply to the selected bodies and then click **OK**. The following four types of entries are supported:
   - **EnvTemp**, a predefined variable, which is the **Temperature** value defined in the *Environment Conditions* tab of the *Mechanical Design Properties* dialog box. This is the default value, but after defining a Thermal Condition, you can save a user-defined value as the new default.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You do not have to assign the environment temperature to objects that are not either hotter or colder than the *EnvTemp* value. Objects with no *Thermal Condition* assignment are assumed to be at the environment temperature (which is the stress-fee temperature).

- A numerical temperature value. Choose the units from the drop-down menu to the right of the text box.
- A user-defined project or design variable name.

  If you type in a new variable name (that is, one not previously defined), the *Add Variable* dialog box appears. Specify the **Unit Type, Unit,** and **Value** in this dialog box and then click **OK**.

- A mathematical expression, which can consist of constants and/or variables (for example, *"EnvTemp + 65cel"*).

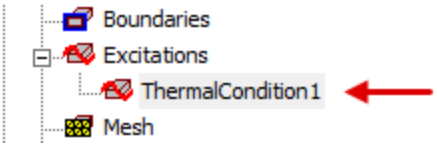
5. If you choose the **Non-Uniform** option, the *Setup Link* dialog box appears. Follow the instructions in the How to Set Up Temperature Linking section of the *Setup Link* subtopic before continuing to the next step of this procedure.

> **Note:**
>
> The algorithm used to map temperature values from the source model to the target model is the System Coupling (SyC) Mapper and is based on the Radial Basis Functions (RBF). For more information concerning system coupling details, visit the Ansys System Coupling homepage (Ansys account signon required). Then do the following:
>
> a. Under *Documentation*, click the **System Coupling User Guide** link.
> b. In the **Search** text box, type **radial basis functions** and press **Enter**.
> c. Click any of the links in the search results that are of interest.
>
> This user guide is comprehensive and includes coverage of loads that are not applicable to Mechanical – Structural analyses.

6. Optionally, change the **Name** of the Thermal Condition excitation.
7. Click **OK** to assign the excitation.

   The assigned Thermal Condition appears under *Excitations* in the Project Manager:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

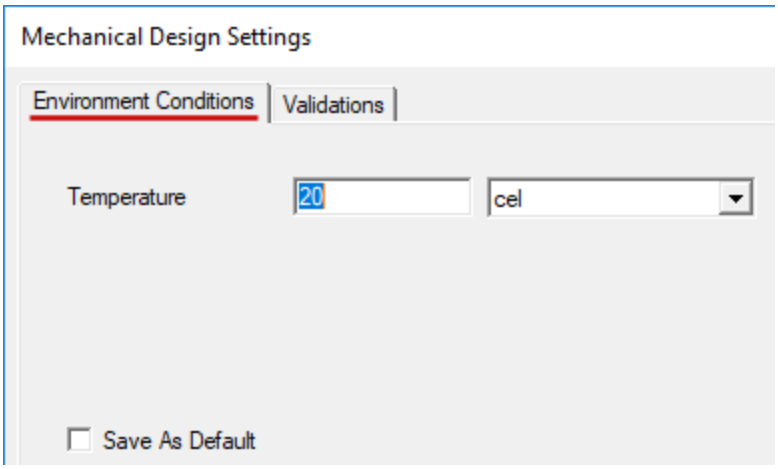


8. If you need to change the value of the *EnvTemp* variable, complete the following procedure:

> **Tip:**
>
> Sometimes, the stress-free temperature of an assembly may be considerably hotter than the temperature of the environment in which the assembly operates. For example, consider a printed circuit board heated by the molten solder applied during a wave-soldering operation. After the solder cools to its fusion temperature and the assembly continues to cool off, thermal stresses are induced by the shrinkage of the solder. In this case, the board is in a state of residual stress at room temperature, and the stress is reduced if the solder is heated. In such cases, you can set the EnvTemp value to a temperature near the point where the solder develops a significant stiffness, assign something close to room temperature to the model's objects, and thereby determine the residual stresses in the assembly.

   a. Access the *Mechanical Design Settings* dialog box using *one* of the following two methods:
      - Using the menu bar, click **Mechanical** > **Design Settings**.
      - Right-click the structural design heading in the Project Manager [**MechanicalDesign*x* (Structural)**, by default]. Then, choose **Design Settings** from the shortcut menu.
   b. In the **Environment Conditions** tab, specify the desired environment (stress-free) **Temperature** value and use the adjacent drop-down menu to choose the **Units**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



c. Click **OK**.

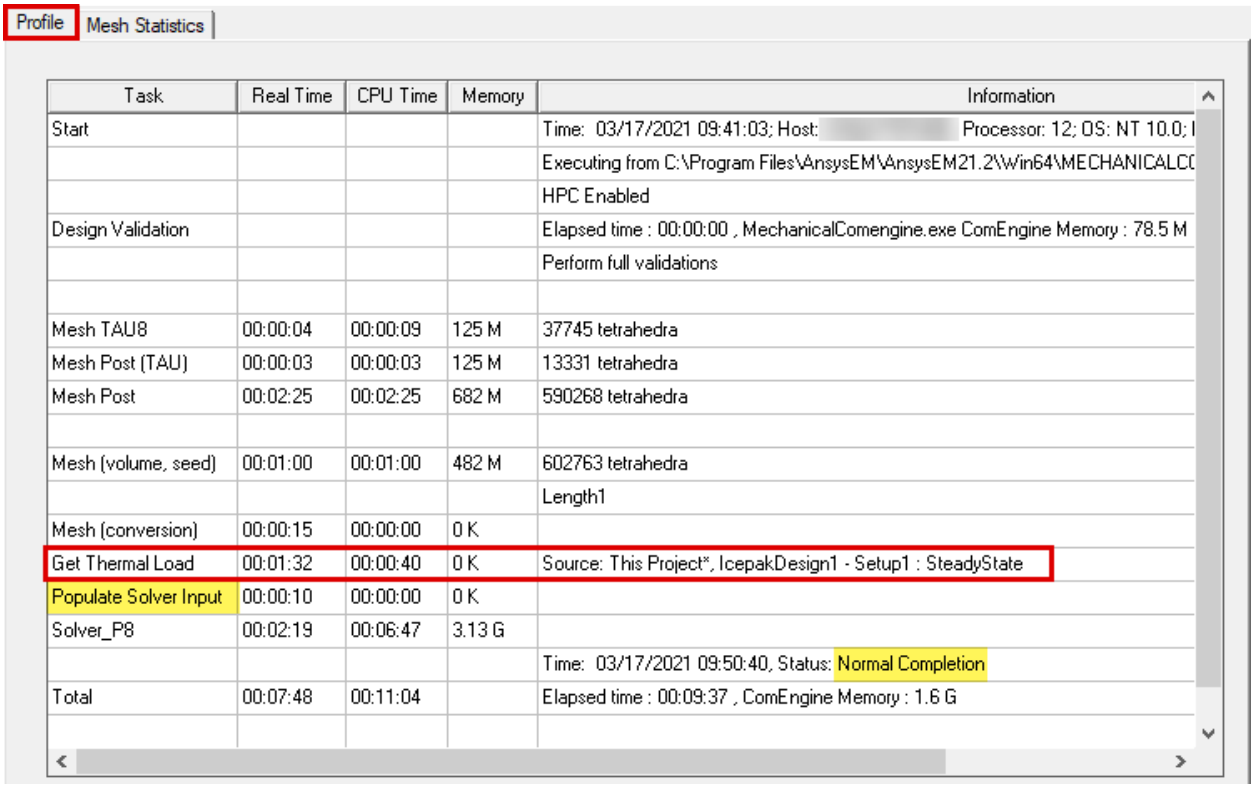
Verifying Thermal Condition Linking

When you have assigned a Non-Uniform Thermal Condition, you can verify that the temperature mapping has been performed successfully from the analysis profile:

- Under *Analysis* in the Project Manager, right-click the target setup and choose **Profile** from the shortcut menu.

  After *Mesh (conversion)*, the **Get Thermal Load** process is listed, including the source project and source design information. This profile entry represents the temperature mapping process. When this step is successfully completed, it is followed by *Populate Solver Input*, and the solution proceeds:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Profile | Mesh Statistics

Task	Real Time	CPU Time	Memory	Information
Start				Time: 03/17/2021 09:41:03; Host: [redacted] Processor: 12; OS: NT 10.0; I
				Executing from C:\Program Files\AnsysEM\AnsysEM21.2\Win64\MECHANICALCC
				HPC Enabled
Design Validation				Elapsed time : 00:00:00 , MechanicalComengine.exe ComEngine Memory : 78.5 M
				Perform full validations
Mesh TAU8	00:00:04	00:00:09	125 M	37745 tetrahedra
Mesh Post (TAU)	00:00:03	00:00:03	125 M	13331 tetrahedra
Mesh Post	00:02:25	00:02:25	682 M	590268 tetrahedra
Mesh (volume, seed)	00:01:00	00:01:00	482 M	602763 tetrahedra
				Length1
Mesh (conversion)	00:00:15	00:00:00	0 K	
Get Thermal Load	00:01:32	00:00:40	0 K	Source: This Project*, IcepakDesign1 - Setup1 : SteadyState
Populate Solver Input	00:00:10	00:00:00	0 K	
Solver_P8	00:02:19	00:06:47	3.13 G	
				Time: 03/17/2021 09:50:40, Status: Normal Completion
Total	00:07:48	00:11:04		Elapsed time : 00:09:37 , ComEngine Memory : 1.6 G

Best Practices:

When you assign a Non-Uniform Thermal Condition, the application does not enforce an identical geometry limitation between the source and target designs. The mapping scheme provides some flexibility. However, your results may be unpredictable and incorrect if there is not a very close correlation between objects in the source design and corresponding objects with imported temperatures in the target design. Follow these best practice guidelines for accurate and reliable results:

- Though not a requirement, it is best to keep the number of Thermal Condition objects in the source and target designs the same. Doing so keeps the designs straightforward and helps to minimize the likelihood of errors.
- The geometry (volume, shape, orientation, and absolute location in global coordinates) of each corresponding pair of source and target objects should be consistent between the source and target designs.
- The mesh quality should be comparable between the source and target objects.

  Source and target meshes cannot be identical between Icepak and other design types. Icepak employs a unique meshing process. However, the mesh sizing can be similar between the source and target designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using a Mechanical–Thermal solution for the temperature source, you can take advantage of mesh linking. This method provides identical meshes between source and target designs and also reduces the solution time by skipping a second mesh generation phase.

> **Tip:**
>
> Thermal Condition mapping is performed on the basis of the locations of the element centroids in the source and target objects. Therefore, you do not have to specify the source object that is associated with each target object. For this reason, you can assign a Thermal Condition excitation to multiple target objects in a single operation.

Cached Link Data and Solution Invalidation

The temperatures from the source design are cached in the target design after its simulation. You can remove this cached data (instructions below).

If you've updated the source design and want the imported temperatures in the target design to reflect the changes in the source project, you need to clear the linked data. Doing so invalidates the previous solutions. Clear the linked data using one of the following two methods:

Method 1: (to clear linked data and/or solutions)

1. Using the menu bar, click **Mechanical** > **Results** > **Clean Up Solutions**.

   The *Clean Up Solutions* dialog box appears.

2. In the Solutions section of the dialog box, choose one of the following sets of options:
   - Select the **Linked Data Only** option.
   - Select the **Fields and Meshes** and **Include Linked Data** options.
   - Select the **All Solutions** and **Include Linked Data** options.
3. Click **Do Deletions**.

Method 2: (to clear linked data only)

- Under *Analysis* in the Project Manager, right-click a solution setup and choose **Clear Linked Data** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Current Thermal Condition temperatures and previously solved results are invalidated when the cached data is cleared.
- Temperatures are cached for each solution setup with a Thermal Condition excitation specified.
- The cached data is automatically cleared when the target Mechanical design is edited in a way that invalidates its results (such as adding, removing, or modifying geometry).
- Cached link data in the target (Mechanical) design is *not* automatically removed when the source design is edited.

Setup Link (Thermal Condition)

When importing *Non-Uniform* temperature results from a Mechanical–Thermal or Icepak design to a *Thermal Condition* excitation in a Mechanical–Structural design, use the *Setup Link* dialog box to specify the source project, source design, and other linking options.

Imported temperatures improve the accuracy of Mechanical–Structural solutions because stresses are based on realistic thermal gradients. Typical temperature distributions are difficult to represent with user-defined, uniform object temperatures.

In a Mechanical–Thermal solution, you specify heat flux or heat generation excitations and convection boundaries to calculate steady-state temperature results. You can also import non-uniform convection film coefficients from an Icepak simulation to improve the accuracy of convection boundaries in a Mechanical–Thermal solution.

Icepak Background Information:

In Icepak, a fluid region surrounds the solid objects, and you specify various flow parameters, such as:

- Openings (flow inlet and outlet faces)
- Grills (openings with screens or perforations that partially inhibit the flow)
- Velocity of the fluid (for forced convection)
- Ambient pressure of the fluid

The remaining Icepak model setup involves specifying heat sources, selecting materials, and so on. The source Icepak model setup should duplicate the materials and geometry in the target Mechanical design, except for the fluid region that surrounds the solid parts in the Icepak design. A fluid-thermal multiphysics analysis is performed. Fluid motion can be driven by natural convection (buoyancy) alone, specified fluid velocities, or a combination of both. Fluid velocities, convective cooling effects, optionally radiative cooling effects, and resultant temperatures are calculated for each element.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refer to the Icepak help for more information.

Important:

- The Mechanical–Structural design objects that are assigned thermal conditions should have geometrically identical counterparts in the Mechanical–Thermal or Icepak design. Temperatures are mapped according to the locations of the elements. Identical geometry ensures that mapping of temperatures from source elements to target elements is performed successfully and accurately. Additionally, the surface mesh size should ideally be similar between the two designs.

  Though Icepak element faces are quadrilateral, and Mechanical faces triangular, the mapping results will be best if the element lengths are comparable.

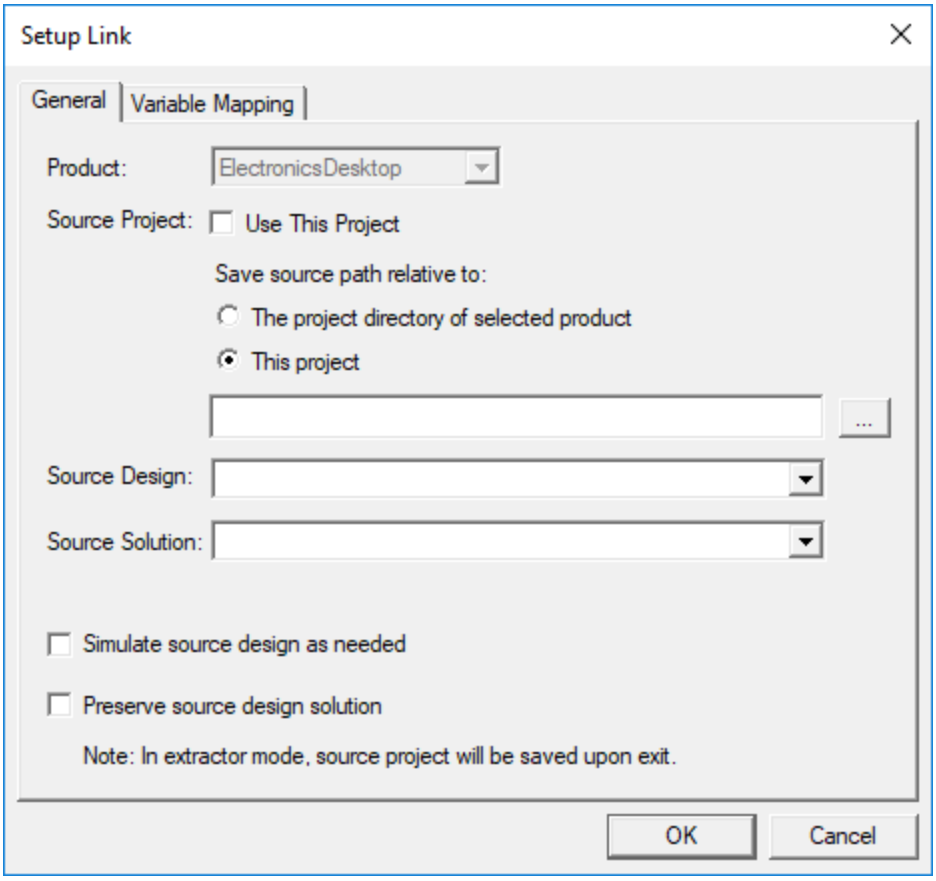
  Optionally, when the temperature source is a Mechanical–Thermal solution, you can take advantage of mesh linking to produce identical source and target design meshes. This option also reduces the solution time by eliminating the mesh generation phase from the structural analysis. Copying a completed mesh takes less time than generating a new one.

- When using an Icepak design as the non-uniform temperature source, the fluid region in the Icepak design must be omitted from the Mechanical–Structural design. A fluid region around the parts would interfere with the thermal expansion and resulting deformation of the solid objects in the Mechanical–Structural solution.
- The link between source and target designs is a one-way coupling operation. Changes you make in the target Mechanical–Structural design have no effect on the source Mechanical–Thermal or Icepak design's results.

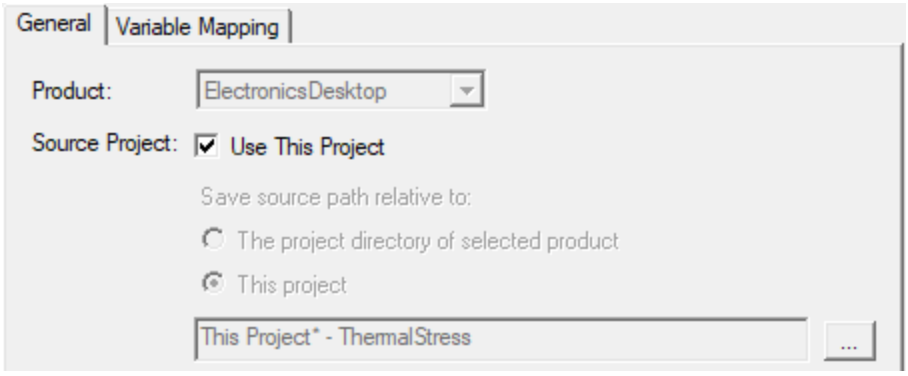
How to Set Up Temperature Linking

1. Follow steps 1 through 3 of the How to Assign a *Thermal Condition* Excitation procedure in the parent topic, selecting the **Non-Uniform** option in the third step. The *Setup Link* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If the data is to be imported from a Mechanical–Thermal or Icepak design in the same project, complete the following substeps. Otherwise go to **step 3**:
    a. Select **Use This Project**.

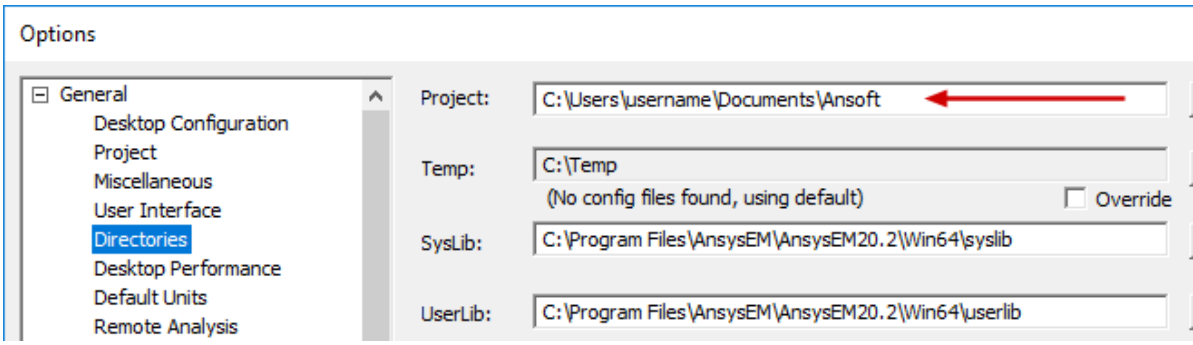


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

b. Skip to **step 4**.

3. Alternatively, if the data is to be imported from a Mechanical–Thermal or Icepak design in a different project, clear the **Use This Project** option and complete the following substeps:
   a. The Mechanical–Structural design saves the relative path to the source project in one of two ways. Choose one of the following two options under *Save source path relative to*:
      - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):



      - **This project:** Save source project path relative to the target project location.

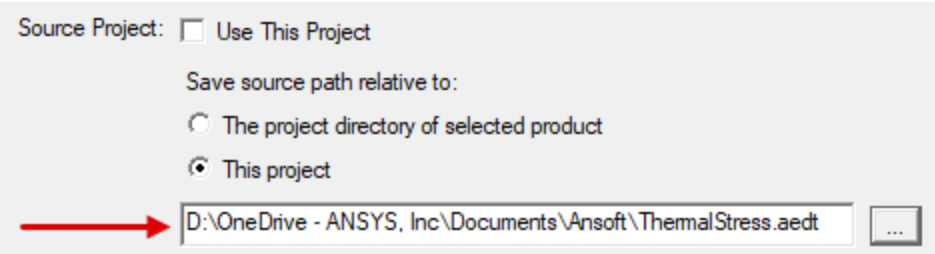
   b. Click the ellipsis button (**...**) to the right of the *Source Project* text box.

      The *Open* dialog box appears.

   c. Locate the source project file and select it.
   d. Click **Open**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected **Source Project** is listed in the *Setup Link* dialog box:



4. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:

> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

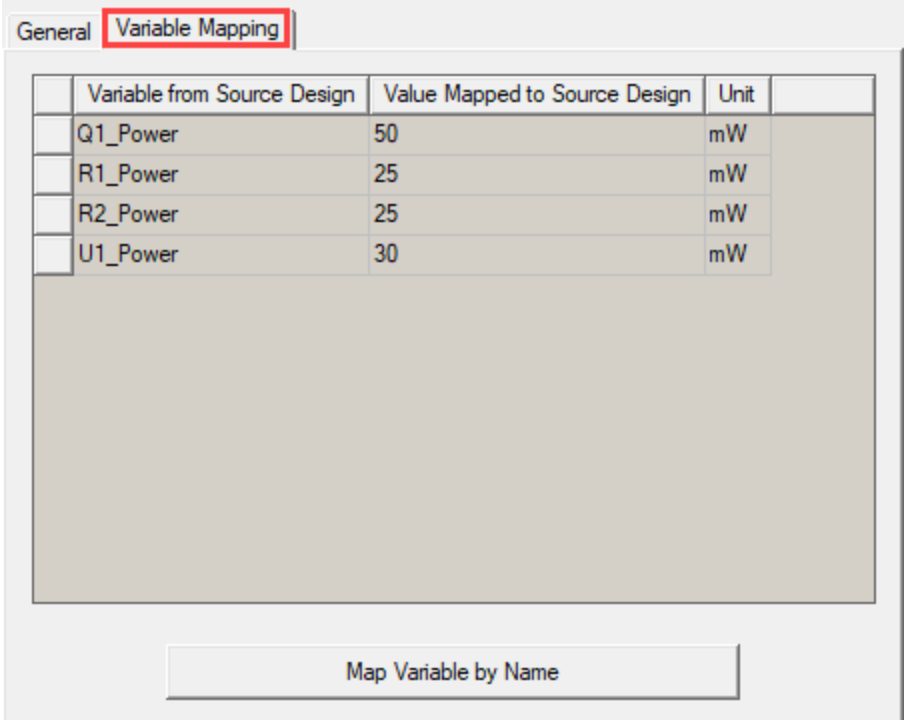
5. Optionally, select one or both of the following two options:
   - **Simulate source design as needed:** Select this option if you want an updated solution to be linked to the *target* design when the *source* design has been modified.

     > **Note:**
     >
     > Certain changes to the source design may not invalidate its solution. In these cases, a new source solution will only be run if the previous one did not converge. To achieve convergence, you may have to increase the specified maximum number of iterations or relax the convergence criterion in the source solution setup.

     (Also, see Cached Link Data and Solution Invalidation in the *Thermal Condition* topic for additional considerations).
   - **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option has no effect if the source design is in the same project as the *target* design.
6. Click the **Variable Mapping** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

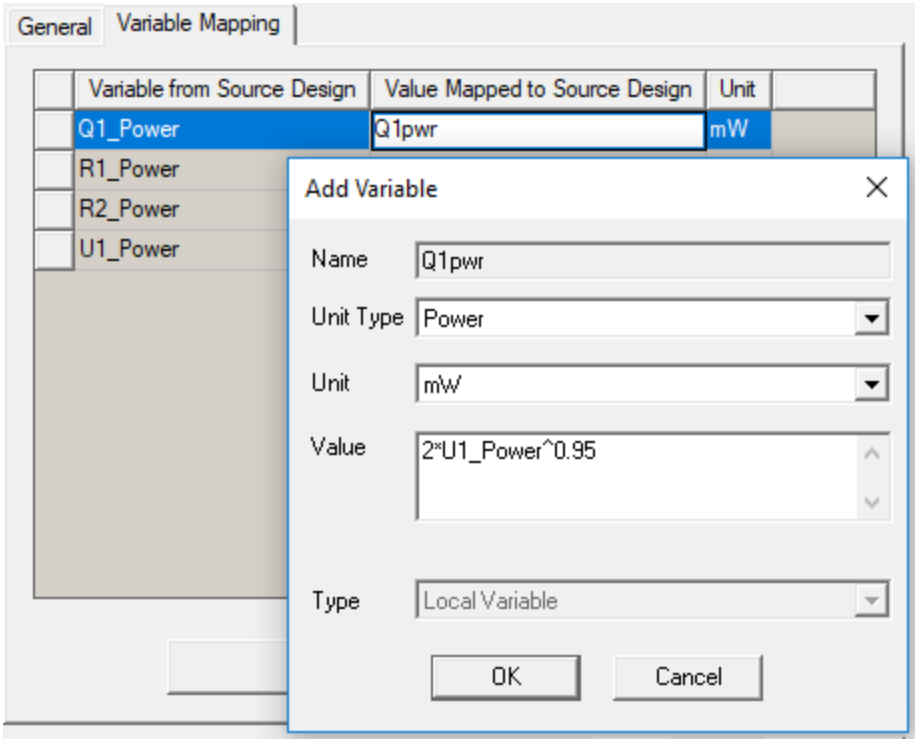
If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:



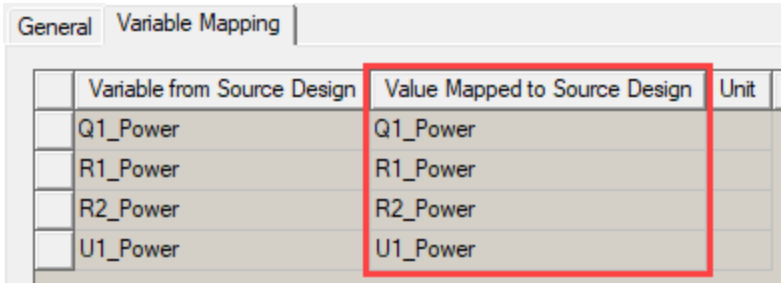
The values passed back to the source design from the mechanical design (second column) can either be constants (numeric values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

> **Note:**
>
> Automatic variable creation is supported in the *Variable Mapping* tab . You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the linked analyses.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- Variables can only be mapped by name if they already exist in the target Mechanical–Structural design. When you copy and paste geometry from the source, or import it from the clipboard, variables used in the definition of the geometry automatically come into the target design. However, for non-geometry-related variables (such as a wattage applied to a thermal source in an Icepak design), you would have to manually define the variable within the Mechanical–Structural design.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

Mapping variables to the source design by name is useful for Optimetrics analyses (for cases where changing the value of a variable affects both the target design and the film coefficients from the source design). You can create an Optimetrics setup in the target design that sweeps both the target and source designs.

8. Click **OK** to accept the link settings and close the *Setup Link* dialog box.
9. Continue at step 6 of the Thermal Condition topic.

Note:

If you later revisit the *Thermal Condition* dialog box to verify or modify its properties, click the **Setup Link** button under the *General* tab to return to the *Setup Link* dialog box. This dialog box will reopen in **View Only** mode. To modify the properties, first select the **Edit Link** radio button.

Editing Boundaries, Excitations and Other Assignments

This topic describes the features available for verifying boundaries, excitations, contacts, or initial temperatures after they have been assigned, renaming them or changing other properties. It also covers changing the scope of the assignment (that is, the faces or objects to which a boundary, excitation, contact, or initial temperature is assigned).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Rename or Change Properties of a Boundary, Excitation, Contact, or Initial Temperature

You can change the *Name* or other properties using one of the following four methods:

- Select the boundary, excitation, contact, or initial temperature in the Project Manager to display its properties in the docked *Properties* window. Then, type a new **Name** or other property in the *Value* column and press **Enter**.
- Double-click the boundary, excitation, contact, or initial temperature in the Project Manager or right-click it and choose **Properties** from the shortcut menu. Then, change the **Name** or other properties in the dialog box that appears and click **OK**.
- For renaming only: Click the boundary, excitation, contact, or initial temperature in the Project Manager to select it. Then, press **F2**, type the new name, and press **Enter**.
- For renaming only: Right-click the boundary, excitation, contact, or initial temperature in the Project Manager and choose **Rename** from the shortcut menu. Then, type the new name and press **Enter**.

To Verify or Change the Scope of a Boundary, Excitation, Contact, or Initial Temperature

To verify or change the assignment faces or objects, use the following commands. These commands are available from the shortcut menu that appears when you right-click a boundary, excitation, contact, or initial temperature in the Project Manager:

- **Select Assignment:** Selects the currently assigned faces or objects (for verification purposes or in preparation for another operation).
- **Reassign:** Replaces the previously specified assignment faces or objects with the currently selected entities.
- **Add to Assignment:** Adds the currently selected faces or objects to the previous assignment set, expanding the scope of the assignment.
- **Remove from Assignment:** Removes the currently selected faces or objects from the previous assignment set, reducing the scope of the assignment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

In certain cases, faces or objects may both be applicable to a boundary or excitation. For example, you can apply thermal convection boundaries or thermal heat flux excitations to faces and objects. In the latter case, the boundary or excitation is applied to all faces of the selected objects.

The *Remove from Assignment* command is only applicable when the selected entities are of the same type as the original assignment. For example, if you apply a thermal convection boundary to an *object*, you cannot remove any *faces* from the assignment to exclude them. In this case, you would have to select all desired convection faces and use the *Reassign* command.

Assigning a Boundary or Excitation to a Portion of a Face

It is a good practice to simplify finite element analyses by eliminating non-essential parts. You can apply boundaries and excitations that produce the equivalent effects of the parts they represent. In other cases, it would be incorrect to assign a boundary or excitation to an entire face of a solid model, and smaller subfaces are needed. A few application examples follow:

Modal Example:

- Consider a part mounted to a rigid base plate by spot-welding at two or three points. Applying a Fixed Support boundary to the entire bottom surface of the part, would overly stiffen the model and increase the natural frequency results. Instead, you could imprint small circular objects onto the bottom of the part, located at the welding areas. Then, apply the boundary to only the resulting small circular subfaces.

Thermal Examples:

- Omit surface-mounted components that are heat sources from a thermal analysis model. Do this by applying heat flux excitations to small faces on the printed circuit board model that represent the footprint of the components. You can make these small faces by imprinting sheets or non-model solid objects onto the face of the board, creating subfaces that are split from the original larger face.
- Consider a heat sink with a rectangular face on which a semiconductor is mounted. It would be incorrect to apply a Convection boundary to the entire face, since the portion of the face underneath the semiconductor is not exposed to the ambient environment. Imprint the shape of the semiconductor onto the rectangular face, splitting the semiconductor's footprint from the rest of the face. Then, apply the Convection boundary to the outer portion of the face only. You could also apply a Heat Flux excitation to the inner subface and eliminate the semiconductor from the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The imprinting process is different for flat and curved faces. Refer to the following subtopics for more information:

- Imprinting on a Flat Face
- Imprinting on a Curved Face

Imprinting on a Flat Face

The following procedure details the process of imprinting on a flat face of a solid object and assigning a boundary or excitation to one of the resultant subfaces. You can use an existing object that contacts the part to be imprinted, or you can draw a sheet object to use as an imprinting tool:

1. Optionally, if there is no model object to use as a tool for imprinting the face you want to split, draw a sheet object as follows:
    a. Optionally, to provide a drawing grid on the target face, create an offset coordinate system based on the appropriate global plane.
    b. Using the available tools on the **Draw** ribbon tab (or equivalent commands in the **Draw** menu), draw a planar object to use as an imprinting tool. This object will define the outline shape (that is, the edges) of the desired subface.

       If drawing polylines, be sure to terminate the operation with the **Close Polyline** shortcut menu command.

       A closed planar shape appears as a sheet object in the History Tree.
2. In **Object** selection mode, select the solid object to which the face you are splitting belongs.
3. While holding down the **Ctrl** key, select the sheet or solid object to imprint onto the previously selected object. You can select multiple tool objects as needed.
4. On the **Draw** ribbon tab, click **Imprint** or, using the menu bar, click **Modeler** > **Boolean** > **Imprint**.
5. In the *Imprint* dialog box, select **Clone tool objects before operation**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Currently, when using the default *Classic* mesher, a sheet object used for imprinting must remain present in the model for the surface mesh to properly conform to the subface on the solid object. Otherwise, the classic mesher would simplify the geometry, ignoring the imprinting. As such, the subface to which a boundary or excitation is applied would not exist in the solution, and a solver error would be encountered.
- While sheet objects used as imprinting tools ensure a mesh that conforms to the imprinted faces, they are otherwise ignored by the solver. The exception is any sheet object in a Thermal solution that has the **Shell Element** attribute selected. Shell Objects do participate in Thermal solutions.
- Alternatively, you could skip this step (that is, choose to *not* clone the tool object), in which case you must select the **Tau** mesher within the *Initial Mesh Settings* dialog box. The Tau mesher will not simplify the geometry by merging a coplanar or tangential subface with the surrounding face.
- If you are using an existing solid object (such as a semiconductor) as the imprinting tool, you most likely want to retain the object, and tool cloning is necessary.

6. Ensure that the solid object to be imprinted is listed under *Blank Parts* and the sheet or solid tool object, or objects, are listed under *Tool Parts*. You can move items from one list to the other by selecting them and pressing the appropriate arrow button.
7. Click **OK** to complete the imprinting operation.
8. If you need to apply a boundary or excitation to the subface just created, select the subface. If it is underneath a sheet object (that is, you chose *not* to skip step 5), use one of the following two selection methods:
    - Hide the sheet objects:
        a. Select the sheet object, or objects, in the History Tree or, in **Object** selection mode, select them graphically.
        b. Hide the sheet objects.
        c. In **Face** selection mode, select the desired subfaces on the solid (those to which you wish to assign a boundary).
    - Use the *Next Behind* selection command:
        a. In **Face** selection mode, click a single sheet object to select its face.
        b. Press **B** to select the solid object's subface *Next Behind* the sheet object face.
        c. Optionally, to select additional boundary assignment subfaces, repeat steps a and b, except hold down **Ctrl** while selecting each additional sheet object face in step a. Do **not** hold *Ctrl* while pressing *B*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, if the boundary or excitation is to be applied to the portion of the original face outside of the imprinted area, simply select the outside face.

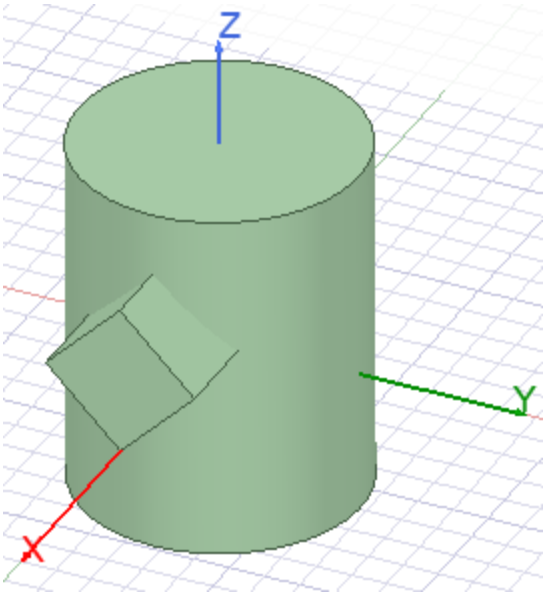
9. Assign the desired Modal Boundary, Thermal Boundary, or Thermal Excitation.

Imprinting on a Curved Face

The following procedure details the process of creating a temporary part to use as a cutting tool for imprinting on a curved face of a solid object. Then, you can assign a boundary or excitation to one of the resultant subfaces:

1. Use any of the available solid object drawing tools, drawing planes, and user-defined coordinate systems (as needed) to produce a solid object of the desired shape. This object must intersect the face of the solid object you wish to split.

   In the following example, a box object intersects a larger cylindrical object, and it does not pass through the opposite side of the cylindrical face. That is, it only intersects the cylinder in one area:



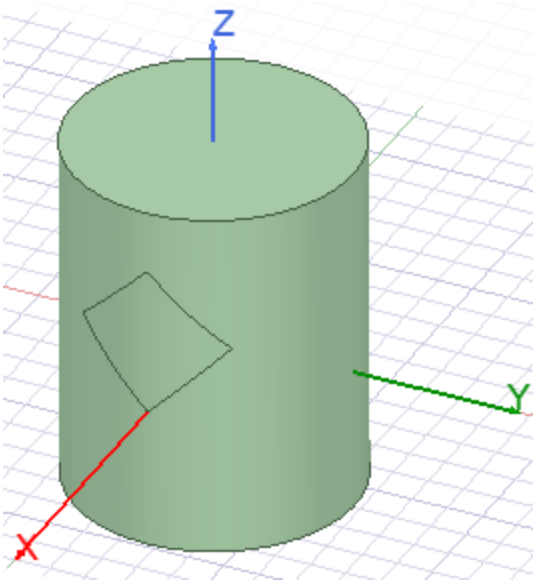
2. In **Object** selection mode, select the solid object that contains the face you want to split.
3. While holding down the **Ctrl** key, select the intersecting object to imprint onto the first solid object.
4. In the *Imprint* dialog box, ensure that **Clone tool objects before operation** is cleared (not selected). For this example, we do not want to retain the tool part after the imprinting operation.
5. Ensure that the solid object with the curved face to be divided is listed under *Blank Parts* and the intersecting object is listed under *Tool Parts*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The first object selected before clicking the *Imprint* command is the *Blank Part*, and any subsequent selections are *Tool Parts*. You can move items from one list to the other by selecting them and pressing the appropriate arrow button.

6. Click **OK** to complete the imprinting operation.

   The following image shows the result for the previously shown example objects:



7. Select the subface just created.
8. On the Draw ribbon tab, click **Surface** > **Create Object From Face**.

   A sheet object is added to the History Tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Currently, when using the default *Classic* mesher, an adjacent object must remain present in the model for the surface mesh to properly conform to the subface on the solid object. Otherwise, the classic mesher would simplify the geometry, ignoring the imprinting. As such, the subface to which a boundary or excitation is applied would not exist in the solution, and a solver error would be encountered.
- While sheet objects overlaid on imprinted faces ensure that the mesh conforms to the imprinted faces, they are otherwise ignored by the solver. The exception is any sheet object in a Thermal solution that has the **Shell Element** attribute selected. Shell objects do participate in Thermal solutions.
- Alternatively, you could skip steps 7 and 8 (that is, choose to *not* create an object from the subface), in which case you must select the **Tau** mesher within the *Initial Mesh Settings* dialog box. The Tau mesher will not simplify the geometry by merging a coplanar or tangential subface with the surrounding face.

9. If you need to apply a boundary or excitation to the subface just created, select the subface. If it is underneath a sheet object (that is, you chose *not* to skip steps 7 and 8), use one of the following two selection methods:
   - Hide the sheet objects:
     a. Select the sheet object in the History Tree or, in **Object** selection mode, select it graphically.
     b. Hide the sheet object.
     c. In **Face** selection mode, select the desired subface for boundary assignment.
   - Use the *Next Behind* selection command:
     a. In **Face** selection mode, click a single sheet object to select its face.
     b. Press **B** to select the solid object's subface that is *Next Behind* the sheet object face.

   Alternatively, if the boundary or excitation is to be applied to the portion of the original face outside of the imprinted area, simply select the outside face.
10. Assign the desired Modal Boundary, Thermal Boundary, or Thermal Excitation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10 - Layout Components in Mechanical [Beta]

In Mechanical designs, support for layout components is a beta feature supported for the following solution types:

- *Steady-State Thermal*
- *Transient Thermal*
- *Structural*

You do not have to enable any beta option to expose the *Layout Component* functionality; it is always available for the supported solution types.

One of the primary purposes of layout components is for trace mapping, which is a means of representing printed circuit boards (PCBs) in a simulation model. The electrically conductive, and structurally stiffer, traces of a PCB (including pads, vias, and conductor pour areas) are accounted for without actually including the conductor geometry in the mechanical design. Layout components can also represent semiconductor substrates, electronic components, heat sinks, and more. Use a previously created *HFSS 3D Layout* design as a source for Adding a Layout Component to your Mechanical design. The conducting objects are visualized in the layout component, but they are not directly included as 3D geometry. The geometry consists only of the **dielectric** layer material (such as FR4) and the **dielectric** fill material specified for signal layers. The conductors are accounted for by locally modifying the material properties (thermal conductivity, density, and specific heat for thermal solutions; Young's modulus, Poisson's Ratio, density, and thermal expansion coefficient for structural solutions). The History Tree lists the source layout component's conducting and dielectric layers and their materials, not the material properties used in the mechanical design.

The signal layers exists in the mechanical design as a contiguous volume of the layer's dielectric fill material. The solver modifies the fill material's properties locally, element-by-element, based on a **metal fraction map**, which is derived from the source layout's conductor geometry. The **resolution** of the metal fraction map is user-defined and needs to be fine enough to accurately capture the metal distribution of the board's signal layers. Metal fraction values are also mapped along the full length of vias, through dielectric and signal layers. See Technical Notes: Metal Fraction Map for more details.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The fill material for signal layers is specified in the *Edit Layers* window of the source HFSS 3D Layout design. Though this fill material is not listed in the History tree of the Mechanical design, it is used in the solution of the layout component.

Similarly, the via plating material is specified in the *Edit Padstack Definition* dialog box of the source HFSS 3D Layout design. If *"None"* is specified, the plating material is taken from the first (topmost) signal layer of the layout. The plating material is not listed in the History tree of the Mechanical design, but it is used in the layout component solution.

For structural solutions, solid dielectric material properties are substituted for air. Gases and fluids cannot support structural boundary conditions. The substitution ensures that these layers are able to support force and pressure excitations or fixed support and frictionless support boundaries.

In HFSS 3D Layout designs, cutouts and via holes are voids and are not meshed. As such, no fill material specification exists in the source design for these items. For thermal solutions, you define these materials as part of the target layout component properties. Air is the default fill material for cutouts, and copper is the default fill material for via holes. For structural solutions, cutouts are handled as non-solve-inside objects. These objects exist but are not meshed, representing voids in the model, similar to the source HFSS 3D Layout design. Therefore, the cutout fill material option is not applicable to structural solutions and does not appear when adding a structural layout component.

Significance of the metal fraction map resolution: You define the number of columns and rows into which the board's length (X dimension) and width or height (Y-dimension) are to be divided. You can specify the resolution numerically or using a slider. Each cell in the resulting grid is assigned a value between 0 and 1, inclusively. The numbers represent the decimal portion of each cell that consists of conducting metal. For example, a value of 0.45 indicates that 45 percent of the cell is metal and therefore, 55 percent is dielectric. The thermal or structural material properties of all elements will be somewhere between the dielectric/fill material and the conductor material, inclusively. When an element extends into two or more grid cells, the element's metal fraction value is interpolated, and the material properties are based on the interpolated metal fraction value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- A general guideline for good results is to set the number of columns and rows such that the cell width and height are about **half of the narrowest trace width** or **half of the smallest via diameter**, whichever result is **smaller**. For example, assume you have a 200 mm x 100 mm PCB with a minimum trace width of 0.5 mm and a minimum via diameter of 0.8 mm. Since the minimum trace width is less than the minimum via diameter, trace width controls the grid resolution. For a target resolution of 0.25 mm (half the trace width), the number of columns and rows would be 800 (200 mm / 0.25 mm) and 400 (100 mm / 0.25 mm), respectively. A finer resolution (more columns and rows), though it might slightly further improve the results, should not be necessary. There is a diminishing return as the grid is made finer and a significant increase in solution time and storage requirements.
- The material property mapping for layout components is mesh-sensitive. The number of elements in the X and Y directions should be **no fewer than one-fourth of the specified number of columns and rows in the grid**. For the example just given, the recommended minimum number of elements along the X and Y directions would be 200 (800 / 4) and 100 (400 / 4), respectively.
- To ensure an adequate mesh density, assign a **length-based mesh operation** to all layers of the layout component. Specify a maximum length of **three to four times the smaller of the grid cell width or height**.

There are several ways to create a layout component in a Mechanical design:

- Select an existing Ansys ECAD Database (*.aedb) folder previously created by, or imported into, HFSS 3D Layout.
- Any layout component that already exists in the current project's Component Library (under *Definitions* > *Components* in the Project Manager) also appears in the **Browse Layout Component** flyout menu. This menu is accessible when you right-click **3D Components** in the Project Manager. Pick the listed name to add another instance of the component or to re-add a previously deleted component.
- Select a layout component file (*.aedbcomp) previously exported from an HFSS 3D Layout design.

Heat Sources

You can define heat sources for steady-state or transient thermal solutions in any combination of the following four ways:

- Specify the **Power Dissipated** in the *Boundary* tab of the *Insert Layout Component* or *Edit Layout Component* dialog box when adding or editing the component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This method distributes the specified power uniformly throughout the component volume. It is a convenient way to account for the heat input of many small parts mounted all over a PCB without having to individually include the components and their heat generation excitations.

- Select **Use from Layout Design DCIR Analysis** in the *Boundary* tab of the *Insert Layout Component*, or *Edit Layout component* dialog box when adding or editing the component.

  If the source HFSS 3D Layout design includes a DCIR analysis, this method imports the non-uniform power dissipated as heat within the layout component.

- Add objects on the top or bottom faces and assign **Heat Generation** excitations to them:

  This method localizes the primary heat sources and also can account for convective heat dissipation from the faces of the mounted components. Currently, this is the best way to account for major heat contributors, such as integrated circuits and power transistors, since the capability of importing components from the source layout is not yet implemented for layout components in mechanical desings.

  While adding a layout component, you can specify convection boundaries on the board's top or bottom faces (in the *Insert Layout Component* dialog box). When you do so, the area under any adjacent solid objects you add to the board will be filtered out of the convection boundary. This is the desired behavior since the board surface under the mounted electronic parts is not exposed to the ambient air. Select exposed faces of the electronic components you add to the design and assign convection to them.

- Overlay shell objects on the top or bottom board face and use these objects to assign **Heat Flux** excitations at selected areas:

  Like the second method, this one localizes the primary heat sources. Heat flux at a shell object can represent the heat conducted to a PCB from a discrete component, or a group of components, without including additional solid geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You cannot imprint faces of a layout component to split them into subfaces for the purpose of localized boundaries or excitations. To perform imprinting, if it were supported, you would have to draw an object anyway. So, as an alternative to adding solid parts, you can draw shells overlaid on the top or bottom PCB face for localized boundaries or excitations. Keep the default infinitesimal thickness of the shell object (1e-7 mm) so that the shell doesn't provide a significant parallel heat conduction path. Alternatively, set the shell thickness and material to approximate the thermal conductivity of the mounted component(s) it represents.

Additionally, if you specify convection at the board faces when adding it, do *not* also assign convection to overlaid shells. Convection boundary areas are not automatically filtered out at shell-to-solid object intersections as they are for solid-to-solid intersections. Assigning convection to the board face and the shell would exaggerate convective heat loss by providing a parallel and redundant heat dissipation path.

You cannot assign boundaries or excitations to layout components in the same way as you do for general objects or faces. That is, you cannot select the layout component or its faces in the Modeler window and execute a boundary or excitation assignment command.

Thermal Boundaries

Two thermal boundaries are supported, and you can assign these to the top or bottom face of the board only as part of the layout component properties. Specifically, the following boundaries are supported. You can assign only one of these boundaries at a time to a given face; they are mutually exclusive:

- Convection

Note:

For layout component (LC) top and bottom faces, you can optionally include radiation effects when defining a convection boundary, regardless of whether you select the *Non-Uniform* film coefficient option. This capability differs from convection boundaries assigned to general objects or faces, which can only indirectly represent radiation when importing non-uniform coefficients from an Icepak solution in which radiation was enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> When choosing to import *Non-Uniform* film coefficients from Icepak, and **if radiation was enabled in the source Icepak solution setup**, do *not* enable radiation in the Mechanical LC definition. Doing so would cause redundant radiation effects to be included in the solution, exaggerating the radiative effect. However, if the Icepak setup did *not* have radiation enabled, you can enable it in the target Mechanical LC definition to account for radiative heat transfer in addition to the convective transfer.

- Temperature

  Use this boundary to fix the temperature of the top or bottom face of the layout component to a specified value. The temperature distribution is uniform.

Structural Boundaries and Excitations

There are two structural boundaries and two structural excitations supported for layout components, and you can assign these to the top or bottom face of the board only as part of the layout component properties when adding it. You can assign only one of these boundaries or excitations at a time to a given face; they are mutually exclusive:

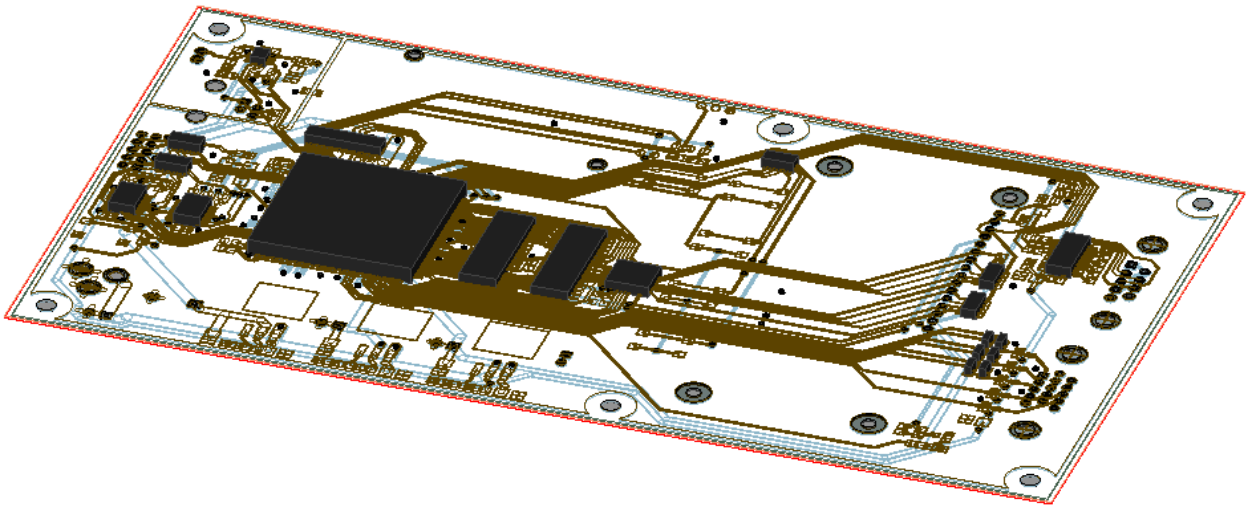
- Boundaries:
  - Fixed Support
  - Frictionless Support
- Excitations:
  - Force
  - Pressure

The behaviors of these structural LC boundaries are the same as when they are assigned to regular 3D geometry. See the appropriate subtopics under Boundaries for more information.

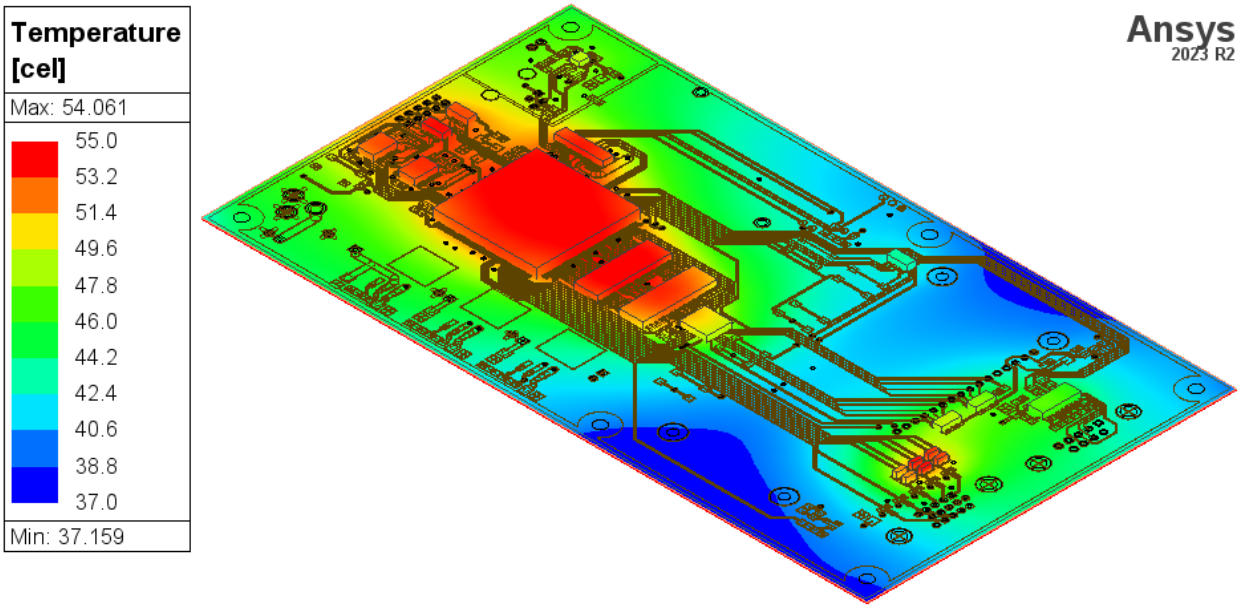
Example

The following figure is a PCB layout component in a mechanical design. The solution type is Steady-State Thermal. In the *Object Attributes*, the signal layer visualization has been set to *Wireframe* mode and dielectrics have been hidden. Twenty components (black objects) have been added to the top surface of the board:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



And the following figure is a temperature results overlay of the same model:



For a workflow video of the transient thermal analysis process using a trace mapping example, see Video: Transient Thermal Analysis with Trace Mapping.

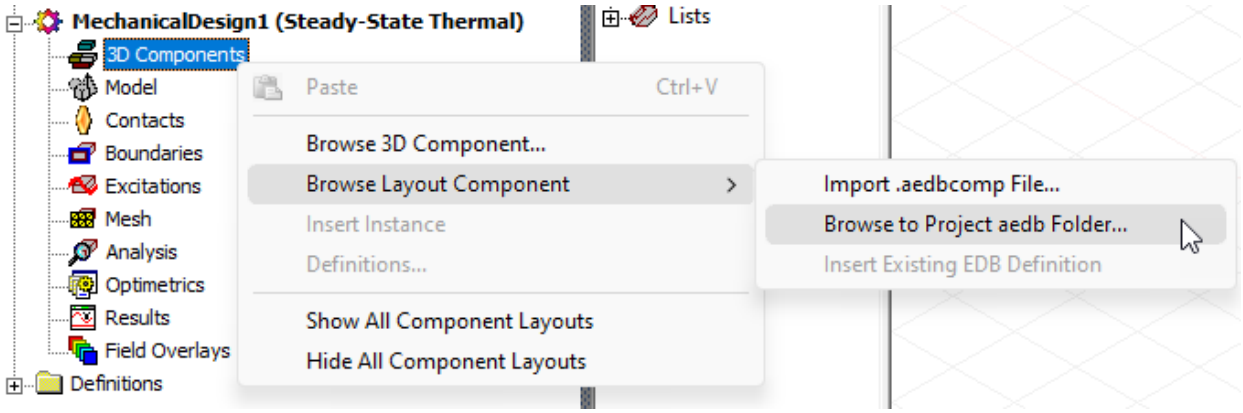
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Layout Component

This topic outlines the procedure for adding layout components to a Mechanical design. See the parent topic, Layout Components in Mechanical, for information on the purpose of these components and the Mechanical design types in which they are supported.

The following instructions assume that an HFSS 3D Layout design, or an exported layout component file, of a printed circuit board (PCB) to be analyzed already exists:

1. In the Project Manager, right-click **3D Components** and access the **Browse Layout Component** flyout menu:

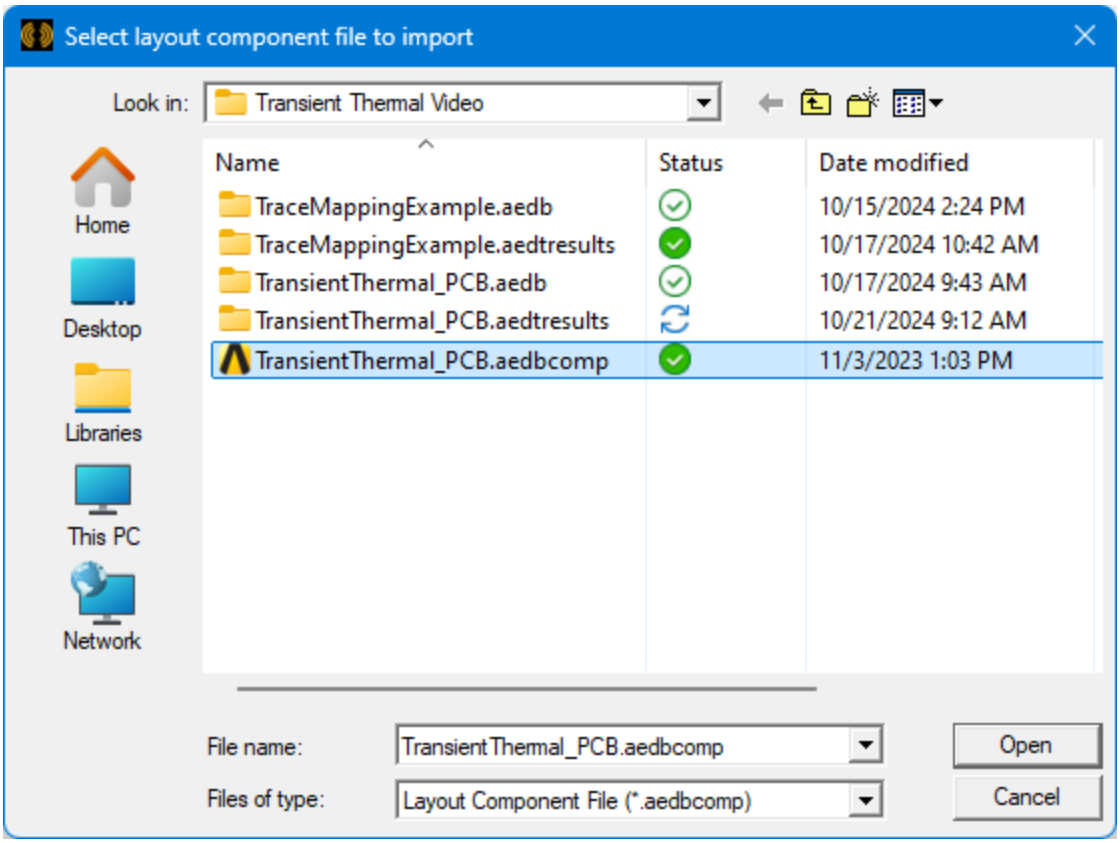


   Then, choose *one* of the following three commands and follow the associated instructions:

   - **Import .aedbcomp File:** Choose this option if you have exported an HFSS 3D Layout as a single-file Layout Component (*.aedbcomp)

     A **Select layout component file to import** dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

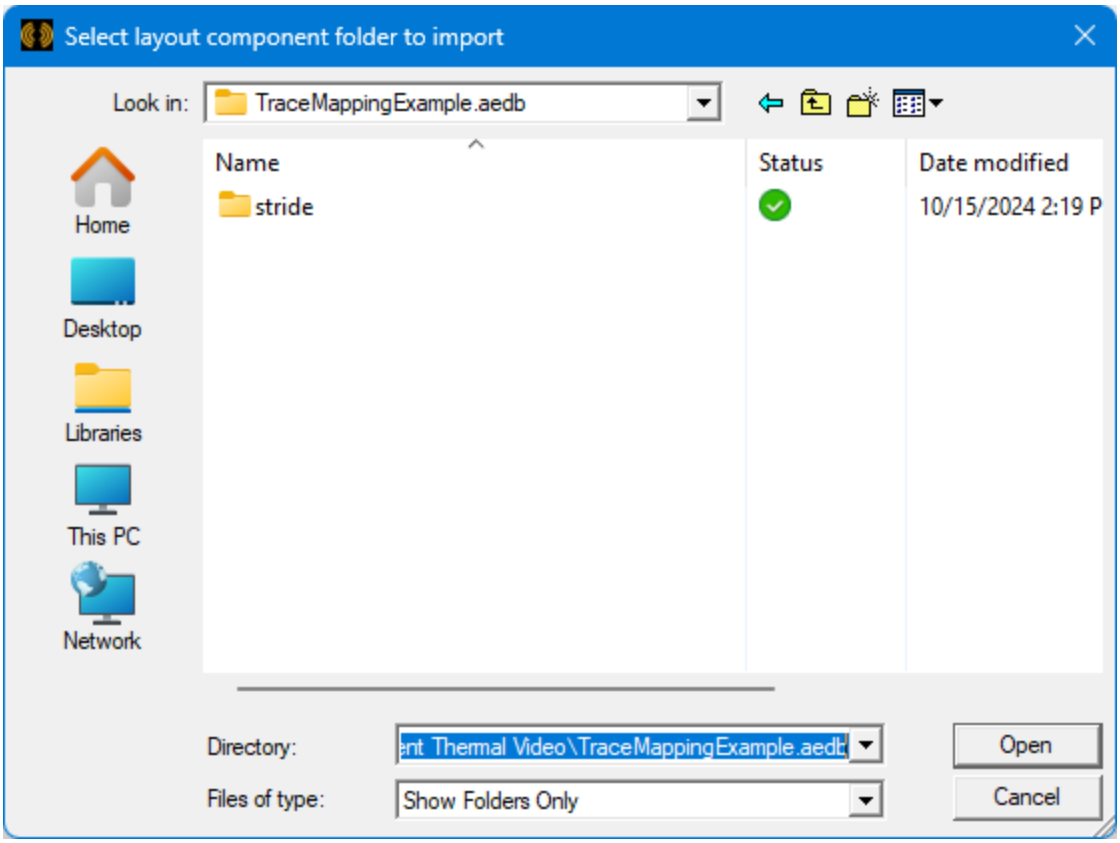


a. Browse to the location of the desired *.aedbcomp file.
b. Select the file.
c. Click **Open** to complete the component source file selection.

- **Browse to Project .aedb Folder:** Choose this option to specify the location of an Ansys electronics database folder (*.aedb) to use in the creation of the layout component. This folder is where HFSS 3D Layout design filesets are stored.

  A **Select layout component folder to import** dialog box appears:

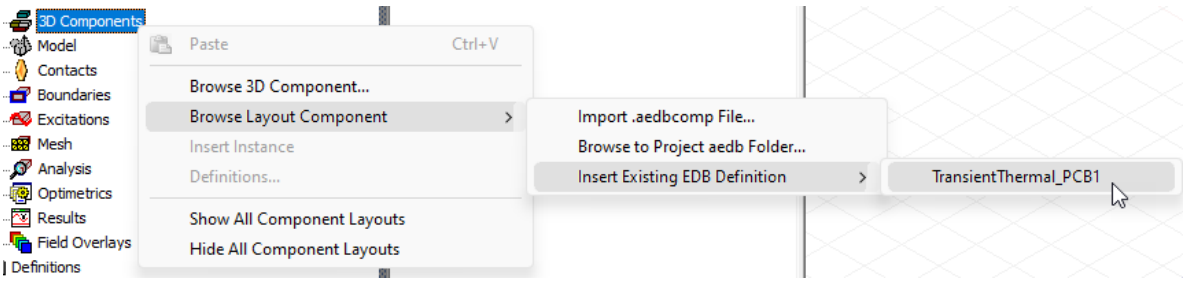
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Browse to the *.aedb folder that contains the desired layout.
b. Double-click the folder to open it.
c. Click **Open** to complete the component source folder selection.

- **Insert Existing EDB Definition:** This option will only be available if you have already imported at least one layout definition into the current project.

  If a layout component definition has previously been added to the current project, the component will be identified by the project name with a serial number appended. It will be listed in the flyout menu even if the component has since been deleted from the active Mechanical design:



  You can select the component from this flyout menu to re-add it to the design. If you were to select it while the component already exists in the design, a duplicate layout component, overlapping the original one, would be added.

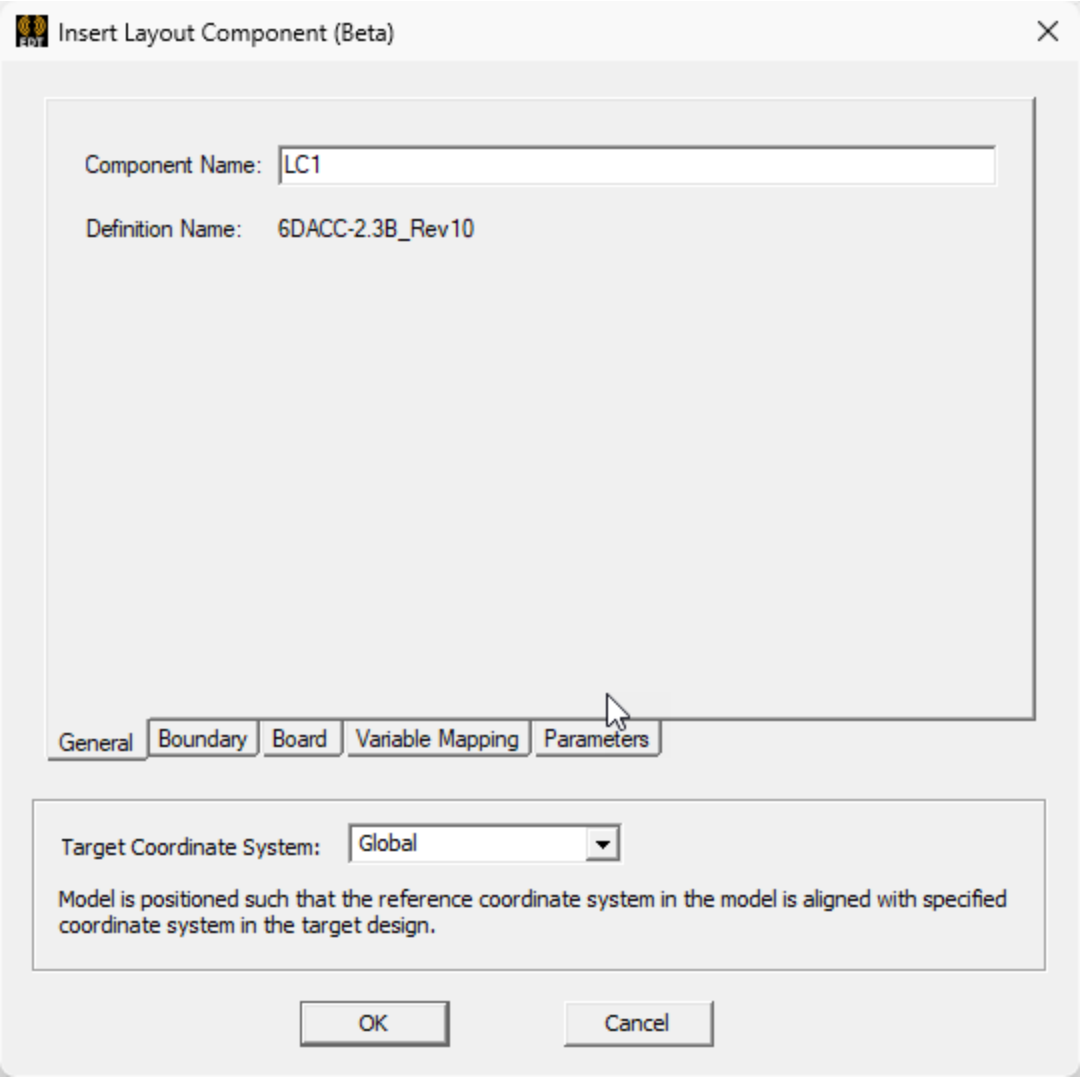
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, avoid doing so unless you want a duplicate component and plan on moving it to a separate location. You can also define a relative coordinate system (RCS) and choose to place the new component according to the RCS instead of dropping it on top of the previous layout component and then moving it.

> **Note:**
>
> Unused layout components that remain in the flyout menu can be removed by editing the *Components Library*. To do so, right click **Components** under *Definitions* in the Project Manager and choose **Edit Library** from the shortcut menu. Select one or more rows of the *Components* table and click **Remove Component(s)**. Click **Yes** to confirm removal for each selected component entry. Finally, click **OK** to exit the *Edit Library* dialog box.

The *Insert Layout Component* dialog box appears with its **General** tab displayed:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

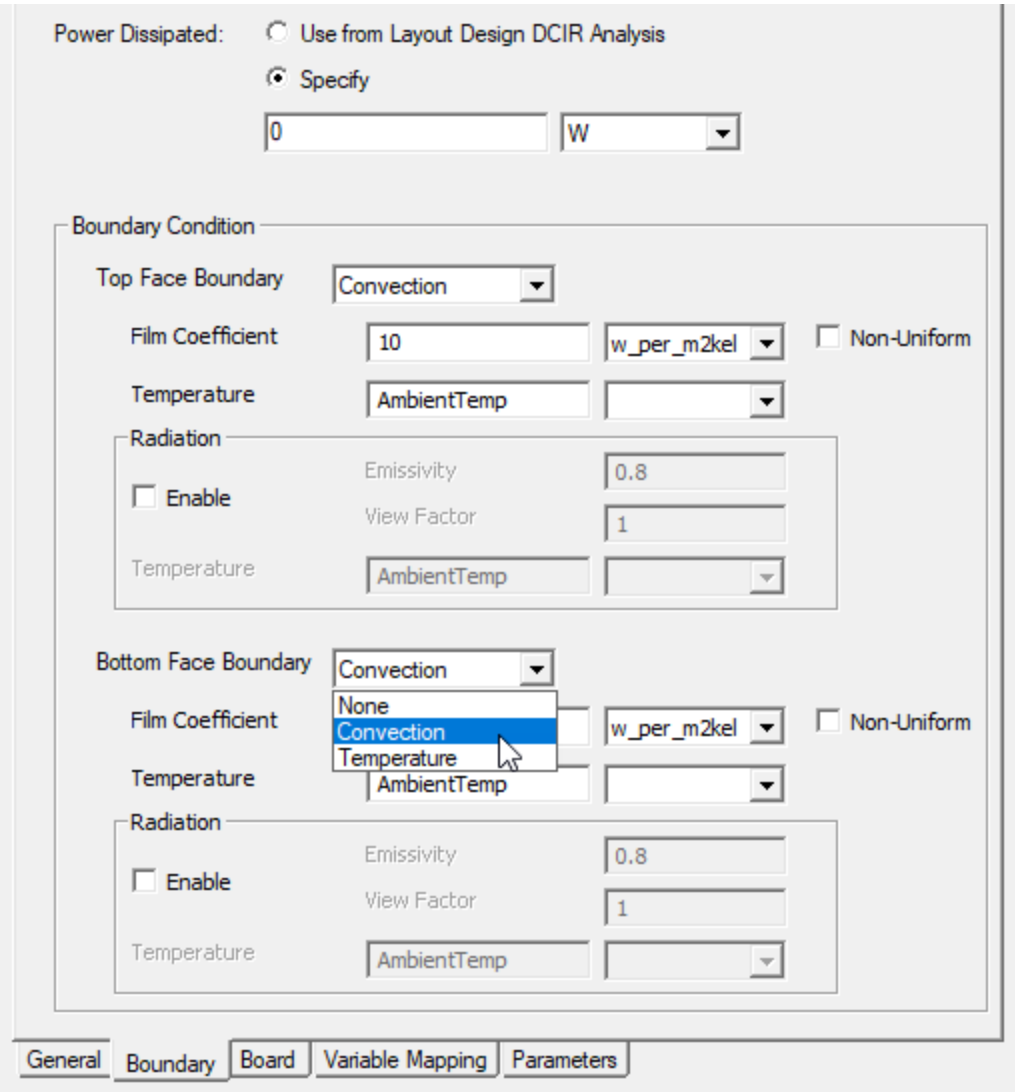


2. Perform the following steps within the **General** tab:
    a. Optionally, change the **Component Name** from its default value (LC*x*, where *x* is an automatically incremented serial number).
    b. Choose the **Target Coordinate System** on which to base the layout component placement. Relative coordinate systems, defined in the target design prior to starting the *Browse Layout Component* process, appear in the drop-down menu in addition to the *Global* CS.

       The *Target Coordinate System* setting is visible regardless of which dialog box tab is currently selected.
3. Select the **Boundary** tab. The contents of this tab depend on the solution type of the active mechanical design.

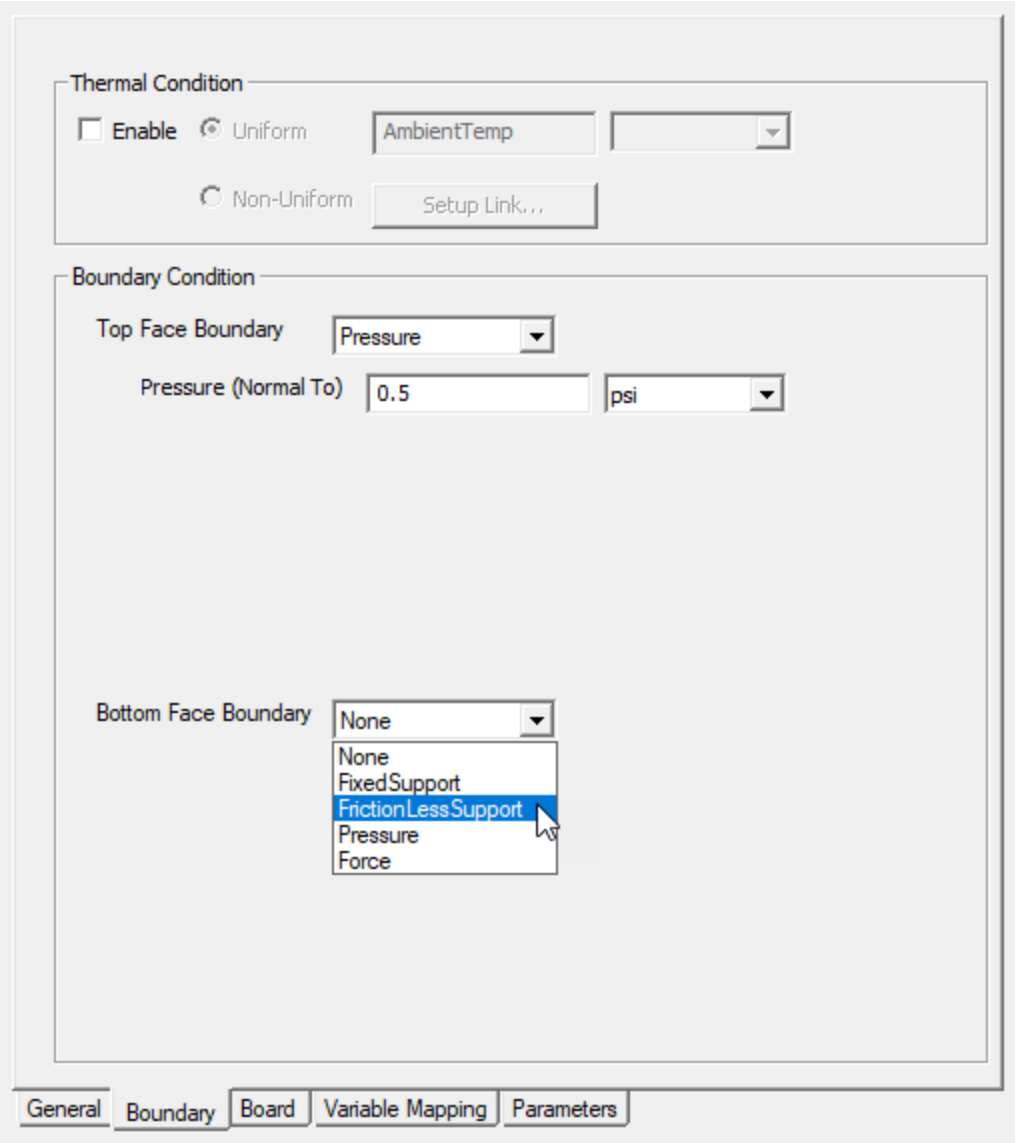
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **Steady-State Thermal** or **Transient Thermal** solutions, the *Boundary* tab contains the following settings:



For **Structural** solutions, the *Boundary* tab contains the following settings:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify the desired boundary options as follows:

a. For thermal solutions, specify the **Power Dissipated** (as heat) by the PCB using one of the following two methods:

   - If the source *HFSS 3D Layout* design includes a **DCIR** setup, and you wish to use the results of that solution as a true, nonuniform heat source for the layout component in the mechanical–thermal solution:

     Select the **Use from Layout Design DCIR Analysis** option.

   - Alternatively, if you wish to specify the power dissipation for the PCB numerically:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Specify**, type the desired total power value in the provided text box, and choose the units of power from the adjacent drop-down menu.

> **Note:**
>
> In this case, the specified power is distributed uniformly throughout the PCB volume. This power can be used to represent the heat input of all mounted components; only smaller, low-power electronic components; or left at zero to include only heat generation you will later assign to PCB-mounted electronic components. For the most accurate results, heat from primary sources should be applied locally, (by assigning heat generation to the mounted components or heat flux to an overlaid shell object at their mounting locations).

b. For structural solutions, you can **Enable** a **Thermal Condition** that either applies a user-specified, **Uniform** temperature to the layout component or imports **Non-Uniform** temperatures from a separate thermal solution of the same layout component.

   When choosing the **Non-uniform** option, you must set up a link to the source *Steady-State Thermal* or *Transient Thermal* solution (either Mechanical or Icepak). Except for the selection of assignment objects, defining a thermal condition is the same for layout components as for general solid objects. For transient source solutions, the temperatures are taken from the final timestep. See Thermal Condition and Setup Link: How to Set Up Temperature Linking for more information.

c. Optionally, in the *Boundary Condition* section, assign boundaries or excitations to either the top, bottom, or both faces of the layout component. Choose the desired boundary condition from the **Top Face Boundary** and **Bottom Face Boundary** drop-down menus. The default selection for both faces is **None**. The other available choices depend on the solution type.

   - **Steady-State Thermal** and **Transient Thermal** solutions have the following *Boundary Condition* choices:
     - **Convection:** Specify the type of convection *Film Coefficient* (**Uniform** or **Non-Uniform**), its value (if Uniform), the **Temperature** of the ambient environment, and choose the units of each value from the provided drop-down menus. Optionally, choose whether or not to **Enable** *Radiation*.

       When choosing the **Non-uniform** option, you must also set up a link to a source Icepak design containing the same layout component. For transient solutions, the heat transfer coefficients will be taken from the *final* timestep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, see Convection Boundary and Setup Link: How to Set Up the Icepak Link. However, be aware that the option to **Enable** *Radiation* is only applicable to layout components. This feature is not mentioned in the context of convection boundaries assigned to general objects or faces.

- **Temperature:** Specify the **Temperature** of the surrounding environment for convection purposes and select its units from the adjacent drop-down menu.
- **Radiation:** Optionally, select **Enable** to include radiative heat transfer effects along with convection.

  If you choose to enable radiation, specify the **Emissivity** of the PCB surface, the **View Factor** between the PCB and surrounding radiation path, and the fixed **Temperature** to which the layout component surface radiates heat.

  A view factor of 1 means that radiation can occur unobstructed in all directions (that is, 100% of the surface area fully participates). Use a non-zero value less than 1 when radiation is partially obstructed ($0 < \text{View Factor} \leq 1$).

  The global *AmbientTemp* variable is the default fixed temperature to which the layout component surface radiates heat. You can also specify a numeric value or user-defined variable, as needed.

  > **Warning:**
  >
  > When choosing to import *Non-Uniform* film coefficients from Icepak, and **if radiation was enabled in the source Icepak solution setup**, do *not* enable radiation in the Mechanical LC definition. Doing so would cause redundant radiation effects to be included in the solution, exaggerating the radiative effect. However, if the Icepak setup did *not* have radiation enabled, you can enable it in the target Mechanical LC definition to account for radiative heat transfer in addition to the convective transfer.

  For either boundary type, if you are keeping the default *AmbientTemp* variable or using a different variable, you need not select the temperature *units* from the adjacent drop-down menu. In either of these cases, the units are specified as part of the variable definition.

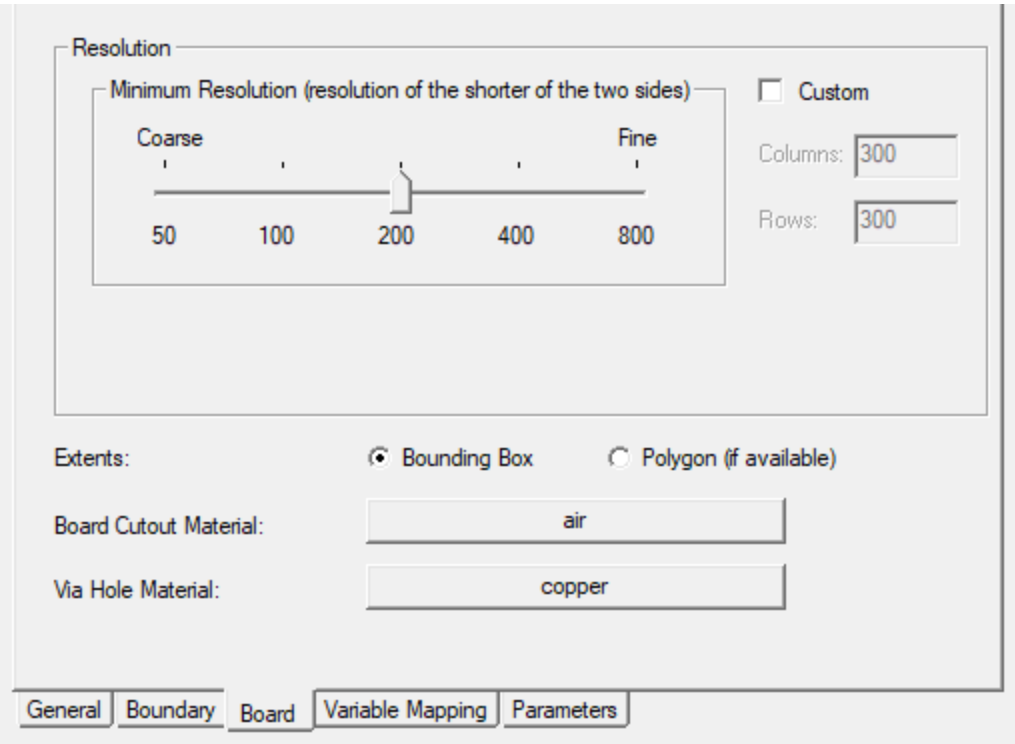
- **Structural** solutions have the following *Boundary Condition* choices:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fixed Support:** Prevent X, Y, and Z translation of the face.
- **Frictionless Support:** Prevent normal translation of the face but allow in-plane motion.
- **Pressure:** Apply a uniform pressure normal to the face (in units of force per unit area). Positive **Pressure** values push into the face; negative values pull the face outward. Choose the units from the adjacent drop-down menu.
- **Force:** Apply three components of uniformly distributed force to the face. Type the desired values in the **Force X**, **Force Y**, and **Force Z** text boxes and choose the units from the adjacent drop-down menus.

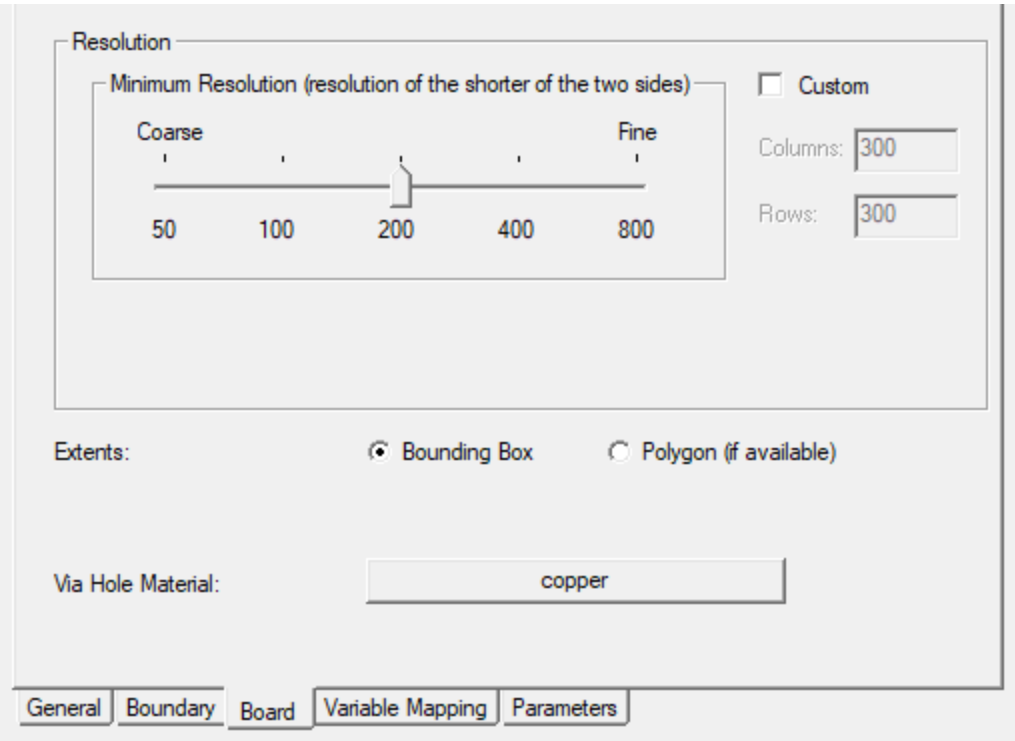
4. Select the **Board** tab. The contents of this tab depend on the solution type of the active mechanical design.

   For **Steady-State Thermal** or **Transient Thermal** solutions, the *Board* tab contains the following settings:



   For **Structural** solutions, the *Board* tab contains the following settings:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify the desired board options as follows:

a. Specify the metal fraction map's *Resolution* using one of the following two methods:
   - Drag the **slider** left (coarser) or right (finer) to specify the number of divisions along the **shorter** dimension of the PCB, whether in the X or Y direction.

     The resolution along the longer dimension is proportional to the specified slider value. Specifically, it is an integer calculated from the slider value times the long/short dimension ratio of the PCB. Therefore, each cell will always be approximately square (width ≈ height) when using the slider.

   - Select **Custom** and specify the number of **Columns** and **Rows** to divide the board into for mapping purposes.

     Columns refers to the number of divisions along the X direction, and *Rows* refers to the divisions in the Y direction, regardless of which dimension is larger.

   See the *"Important"* guidelines in the *Layout Components in Mechanical* topic for resolution recommendations.

b. Choose the desired **Extents** option:
   - **Bounding Box:** The extents of the source layout drawing
   - **Polygon (if available):** A polygon or closed polyline shape indicating the PCB perimeter (when included in the *Outline* layer of the layout)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The HFSS 3D Layout drawing on which a layout component is based will not necessarily have its dielectric layer geometry extended to the perimeter of the board it represents. A polygon or closed polyline is typically drawn to indicate the outline of the board. In the HFSS solution, a specified fill material fills any vacant space in the layout.
>
> Using the *Extents* option, you can choose whether the board in the mechanical design is limited to the extents drawn in the original layout or extends to the outline polygon (assuming one is available).

c. Optionally, choose alternative fill materials:
   - For **Steady-State Thermal** or **Transient Thermal** solutions, choose the desired **Via Hole Material** and **Board Cutout Material**.
   - For **Structural** solutions, choose the desired **Via Hole Material**.

> **Note:**
>
> The default material for via holes is copper.
>
> In thermal solutions, the default board cutout material is air, and the cutout represents an opening through which cooling air can flow. However, in structural solutions, cutouts are modeled as non-solve-inside objects (representing voids in the model), and no material assignment applies.

5. Select the **Variable Mapping** tab.

   What appears at this point depends on whether variables are defined in the source layout and are applicable to the current solution type. Choose one of the following two actions:

   - If there are no applicable variables in the source layout, no variables will be listed in this tab, and you can **skip to step 7**.
   - When applicable variables are present in the source layout, they are listed on this tab, and the following actions are possible:
     a. Optionally, you can **substitute a different numerical value** or **substitute variables** (instance parameter names) for the numeric values in the *Values Mapped to EDB* column. This substitution is particularly useful if you wish to include multiple instances of the layout component in a single design and have geometry variations between the instances. See Instance Variation for more information. Additionally, you can use variables to manipulate the geometry during an optimization or parametric analysis.

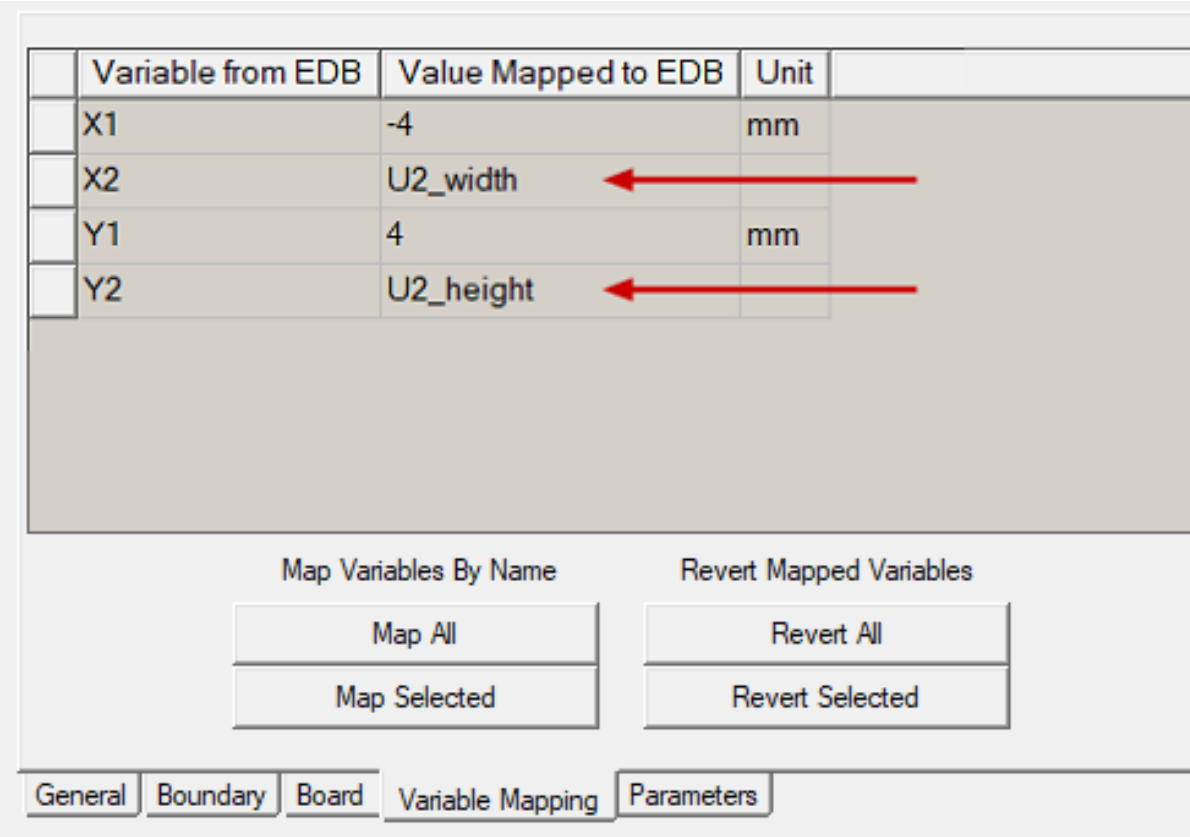
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In-place definition of new variables is supported. If you specify a previously undefined variable name, the *Add Variable* dialog box will appear when you press **Enter** or advance to a different table cell so that you can define its unit type, units, and initial value.

b. Alternatively, you can use the following buttons to automate the variable mapping task or to revert to the original values:
    - **Map All:** For all source layout variables, this command creates instance parameters, each with the same name and value as the source variable.
    - **Map Selected:** For selected layout variables, this command creates instance parameters, each with the same name and value as the source variable. Before clicking the *Map Selected* button, click a table row to select a variable to map. Ctrl+click to select multiple variables or to toggle the selection state of a row.
    - **Revert All:** For all variables, this command reverts the **Value Mapped to EDT** entry to the original source layout value.
    - **Revert Selected:** For selected table rows, this command reverts the **Value Mapped to EDT** entries to the original source layout values. Before clicking the *Revert Selected* button, click a table row to select a variable to revert. Ctrl+click to select multiple variables or to toggle the selection state of a row.

    In the following example, two of the four variables in the source ECAD database (X2 and Y2) have had instance parameters substituted for the original numeric values:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

In the *Parameters* tab (next step), you can choose whether to create local *Design* or *Project* variables from instance parameters. For the latter type, the name will be automatically prefixed with a dollar sign ($) when the local variable is created. Therefore, you do *not* have to manually include a dollar sign at the beginning of the name you type.

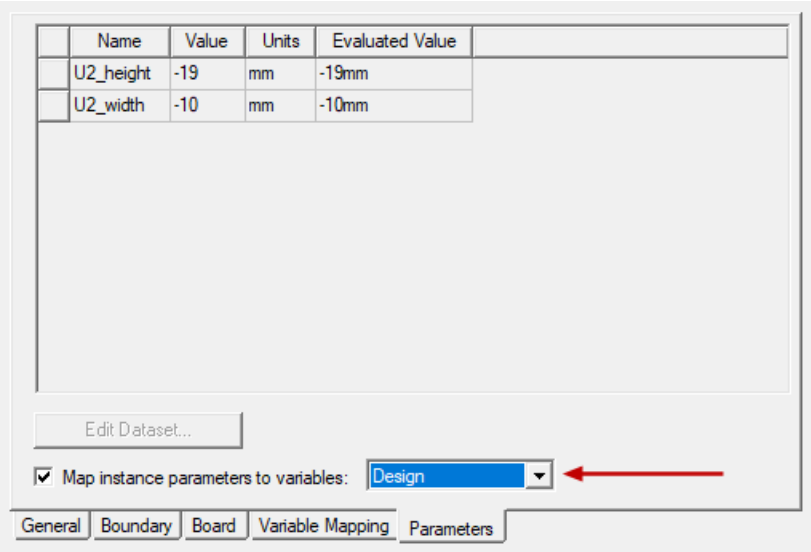
6. Select the **Parameters** tab.

   If the source design contains no applicable variables, or if you chose not to define local instance parameters in the *Variable Mapping* tab, nothing will be listed in the *Parameters* tab. **Skip to step 7**.

   Otherwise, when you have defined local instance parameters in the *Variable Mapping* tab, they will be listed in the *Parameters* tab along with their values and units.

   The following figure shows the two instance parameters that were substituted for numeric values (as shown in the preceding figure):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can choose whether to keep these mapped variables only as instance parameters or to also create local *Design* variables or *Project* variables, as follows:

i. Optionally, toggle the selection of **Map instance parameters to variables** as preferred.

> **Note:**
>
> If this option is cleared, mapped geometry variables are exposed in the mechanical design only as layout component parameters, and you can vary the values between multiple instances of the component in the same design.
>
> Alternatively, if you also wish to include the mapped variables as mechanical *design* or AEDT *project* variables, select this option and choose the desired local variable type (next substep). Local design or project variables are needed if you wish to expose and manipulate the values in a parametric or optimization analysis. Otherwise, if you only need to vary a parameter for each inserted instance of a layout component, local design or project variables are *not* necessary.

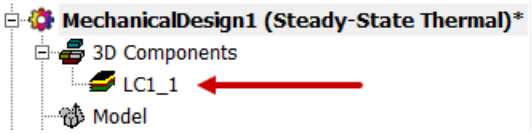
ii. If the *"Map instance parameters to variables"* option is selected, choose either **Design** or **Project** from the adjacent drop-down menu to specify the type of local variable to create.

As previously stated, the variable name will be prefixed with a dollar sign ($) automatically when you choose to create local *Project* variables.

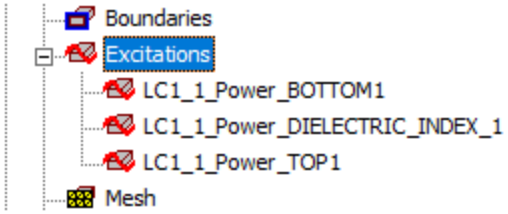
7. Click **OK** to close the *Insert Layout Component* dialog box and insert the component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The layout component name appears under *3D Components* in the Project Manager. An instance number ("_1" in this case) has been appended to the component name:



Additionally, if your component definition included *Power Dissipated* (thermal solutions) or a *Thermal Condition* (structural solutions), excitations will appear in the Project Manager for each PCB layer:



8. Optionally, to change the visualization characteristics of the layout component, do the following:
   a. Under *3D Components* in the Project Manager, select the ***<layout_component_name>***.
   b. In the docked *Properties* window, select the **Visualization** tab

      See Visualization Tab for details about the visualization options and object attributes.

   c. If you visited the *Object Attributes* dialog box, click **OK** to apply your settings and to close the dialog box.

9. In **object** selection mode, select all layers of the layout component and assign a length-based mesh operation **on** the selection. Set the maximum element length to be **three to four times the width or height of the grid cells**, whichever is the **lesser**.

   Typically, you need not enforce the maximum element length *inside* the selection. For the majority of printed circuit boards, there will only be one element through the thickness of any layer due to their thinness.

Inserting a Layout Component Instance

Once you have added a layout component to a design, you can insert additional instances of it easily, duplicating the definition of the first one. Access the command using one of the following two methods:

- Right-click **3D Components** in the Project Manager and choose **Insert Instance** > ***{Layout_Component_Name}***

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the menu bar, click **Draw** > **3D Component Library** > **Insert Instance** > ***{Layout_Component_Name}***

Additional instances of a layout component will be inserted at the same location as the original one. Select each additional instance under *3D Components* in the Project Manager and move it to the desired location, eliminating any overlap between the layout components.

If you later decide to change any property in the layout component definition, use the Edit Definition command to edit the source HFSS 3D Layout design. However, be aware that changing the definition in this manner affects all instances of the component referenced in all designs of the project. You can also change the settings that were specified when adding the component by using the Replace Component command. Be aware that changing the settings in this manner affects all instances of the component in the current design.

For variations to occur between instances of the same component, at least one project or design variable must be defined and used in the source layout. Instance variation is achieved as described below.

Instance Variation

- Prerequisites:
  - At least one project or design variable must be defined in the HFSS 3D Layout source design.
  - The variable must be used to define a model property (such as a layer thickness, trace width, or object location).
- Defining a layout component with variable mapping:

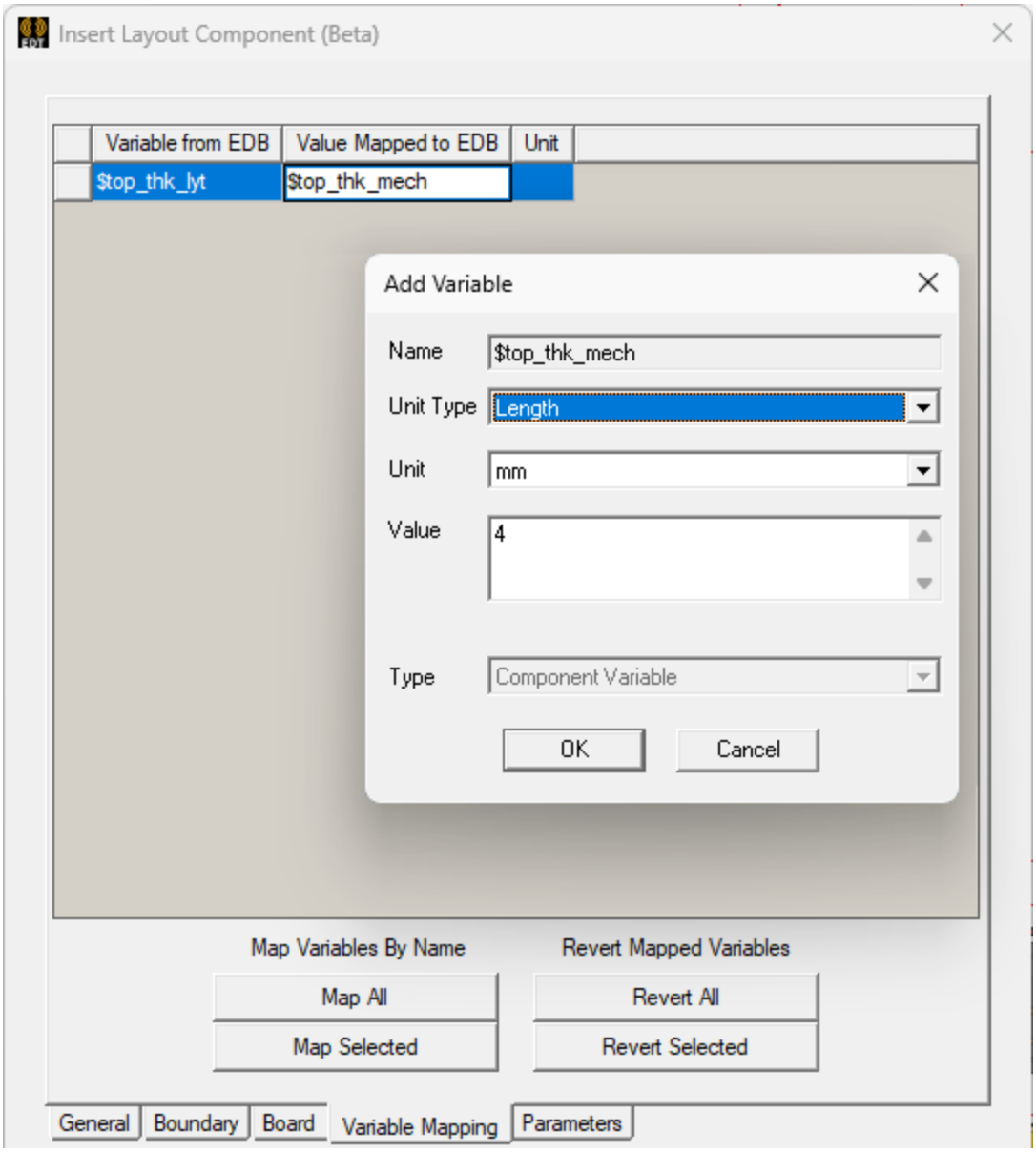
  While adding the layout component to your mechanical design, do the following in the **Variable Mapping** tab of the *Insert Layout Component* dialog box:

  1. In the *Value Mapped to EDB* column, replace one or more numerical values with the name of a local variable and press **Enter**.

     If the name you specified has not already been defined, the *Add Variable* dialog box will appear.

     a. Select the **Unit Type** and **Unit** from the drop-down menus.
     b. Optionally, change the prepopulated **Value**, which initially equals the value assigned to the associated variable in the source layout.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



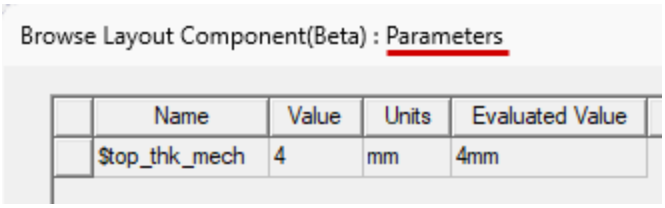
Note:

Alternatively, you can map **All** variables or **Selected** variables to local instance parameters using the buttons near the bottom of the dialog box. Variables specified in this manner have their initial defined value automatically set to the source variable's current numerical value. You can also revert instance parameter variables to their original numerical values if you change your mind.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Repeat steps 1–4 for any other variables you wish to map between the target and source designs.
3. Select the **Parameters** tab.

   This tab lists any local instance parameters that have been mapped in the preceding tab along with the current numerical value of each:



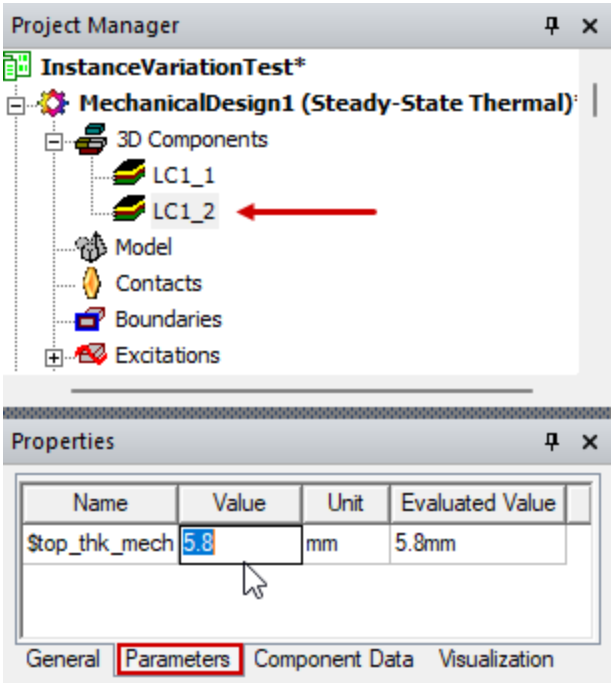
4. Optionally, choose whether or not to **Map instance parameters to variables**. If you select this option, use the drop-down menu to choose whether to create local **Design** or **Project** variables. Doing so exposes the variables so that they can be manipulated in Optimetrics analyses in the local design. Otherwise, clear the option to keep the variables solely as instance parameters that you can use to vary the geometry manually from one instance to another.

   Regardless of whether you choose to map parameters to local variables or not, each parameter will be exposed as a property of the layout component instance, and the value can be manipulated independently for each instance.

5. Click **OK** to insert the completed layout component.

- Inserting two or more instances of the layout component:
  1. Insert one or more additional instances of the layout component as detailed near the top of this page.
  2. Move the additional instances of the component, or insert them with a different coordinate system active, so that they don't overlap previously added layout components. Locate as desired.
  3. Under *3D Components* in the Project Manager, select the instance for which you want to change a mapped parameter (such as *"LC1_2"*).
  4. Optionally, in the *Parameters* tab of the docked *Properties* window, change the **Value** of one or more parameters and press **Enter** to vary geometry between instances.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The geometry changes should be immediately visible in the Modeler window. Position the viewpoint and zoom in/out as required to clearly compare the geometry for each instance.

Replacing or Editing Layout Components

Use the command, **Replace Component** to edit the properties that were specified when a layout component was inserted into your design or to replace the component with a different one. This command is located in the shortcut menu that appears when you right-click a layout component instance in the Project Manager. A multi-tab dialog box appears, *Edit Layout Component*, which is very similar to the *Insert Layout Component* dialog box discussed in the topic, Adding a Layout Component. The available tabs, and the options that appear in each, are identical to the *Insert Layout Component* dialog box except for the following differences:

- The **General** tab of the *Edit Layout Component* dialog box includes a **Definition** drop-down menu for selecting an AEDB folder, *.aedbcomp file, or existing component definition in the current design. Use this drop-down menu to replace the current component with a different one.

  When originally adding a layout component to a Mechanical design, you choose which one to insert using the *3D Components* shortcut menu and the *Browse Layout Component* flyout menu it contains. Therefore, the *General* tab of the *Insert Layout Component* dialog box lacks the **Definition** drop-down menu, because the source layout choice has already been made before the dialog box appears.

- In the **Variable Mapping** tab of the *Edit Layout Component* dialog box, you cannot revert previously mapped instance parameter variables to the original numeric values as you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can in the *Insert Layout Component* dialog box. Once the mapping is committed by completing the insertion process, the variables remain as instance parameters.

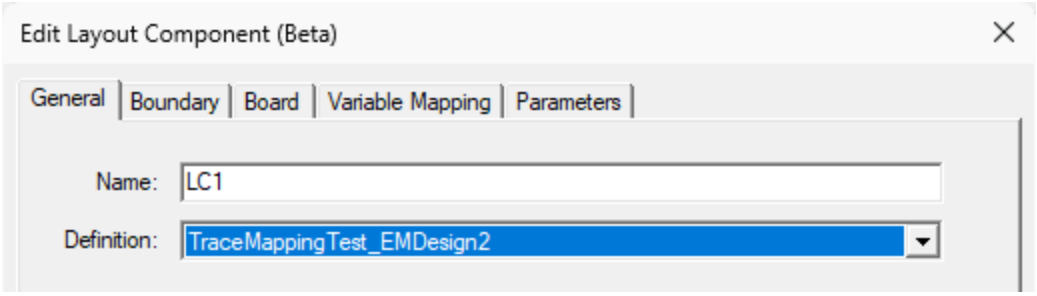
- The **Parameters** tab of the *Insert Layout Component* dialog box includes an option to make local design or project variables from component instance parameters along with a drop-down menu to choose the type of variable to save. These settings are absent from the *Edit Layout Component* dialog box. You can only create local design or project variables when originally adding the layout component to your Mechanical design. Even if you have chosen to map additional source layout variables to instance parameter variables when editing a component, you cannot automatically create associated local design or project variables from the *Edit Layout Component* dialog box.
- The tab locations differ between the two dialog boxes. For *Insert Layout Component*, the tabs are located near the bottom of the dialog box. For *Edit Layout Component*, the tabs are located near the top of the dialog box. However, the same tabs are included in both dialog boxes, and they appear in the same order.

Edit/Replace Procedure

To edit an existing layout component in your Mechanical design, or to replace it with a different component, do the following:

1. Under *3D Components* in the Project Manager, right click the ***<Layout_Component_Name>*** and choose **Replace Component**.

   The *Edit Layout Component* dialog box appears with the **General** tab initially selected:



2. Optionally, to select a different component to replace the current one, choose *one* of the following three options from the **Definition** drop-down menu:

   > **Note:**
   >
   > See Adding a Layout Component for more information on the following options.

   - **Import .aedbcomp File**
   - **Browse to Project .aedb Folder**
   - Previously added layout component definitions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

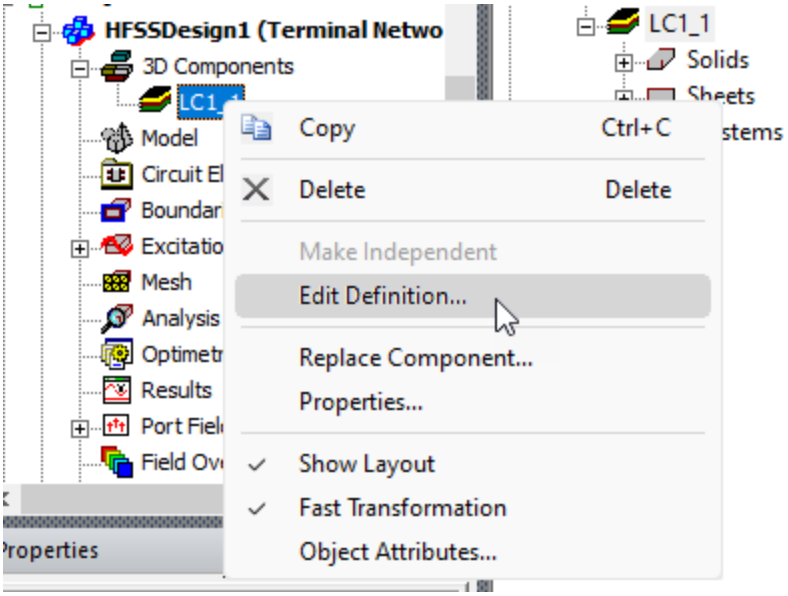
3. Progress through the **Boundary**, **Board**, **Variable Mapping**, and **Parameters** tabs of the *Edit Layout Component* dialog box in the same manner as detailed for the *Insert Layout Component dialog box* in the Adding a Layout Component topic. Take into account the limitation described above for the *Variable Mapping* and *Parameter* tabs.
4. Click **OK** when you are done editing the layout component.

   If you selected a new source file or folder for the layout component, a new layout component is added under *definitions* > *Components* in the Project Manager. Additionally, the layout component in the Modeler window is regenerated.

> **Note:**
>
> If the component's visualization options were customized in the Mechanical design, they will be reset to their default configuration whenever the layout component is replaced or regenerated after editing. When this occurs, repeat step 8 in the *Adding a Layout Component* procedure to restore your preferred visualization options.

Editing a Layout Component Definition

All layout components in Mechanical designs have a command in the right-click menu called **Edit Definition**.

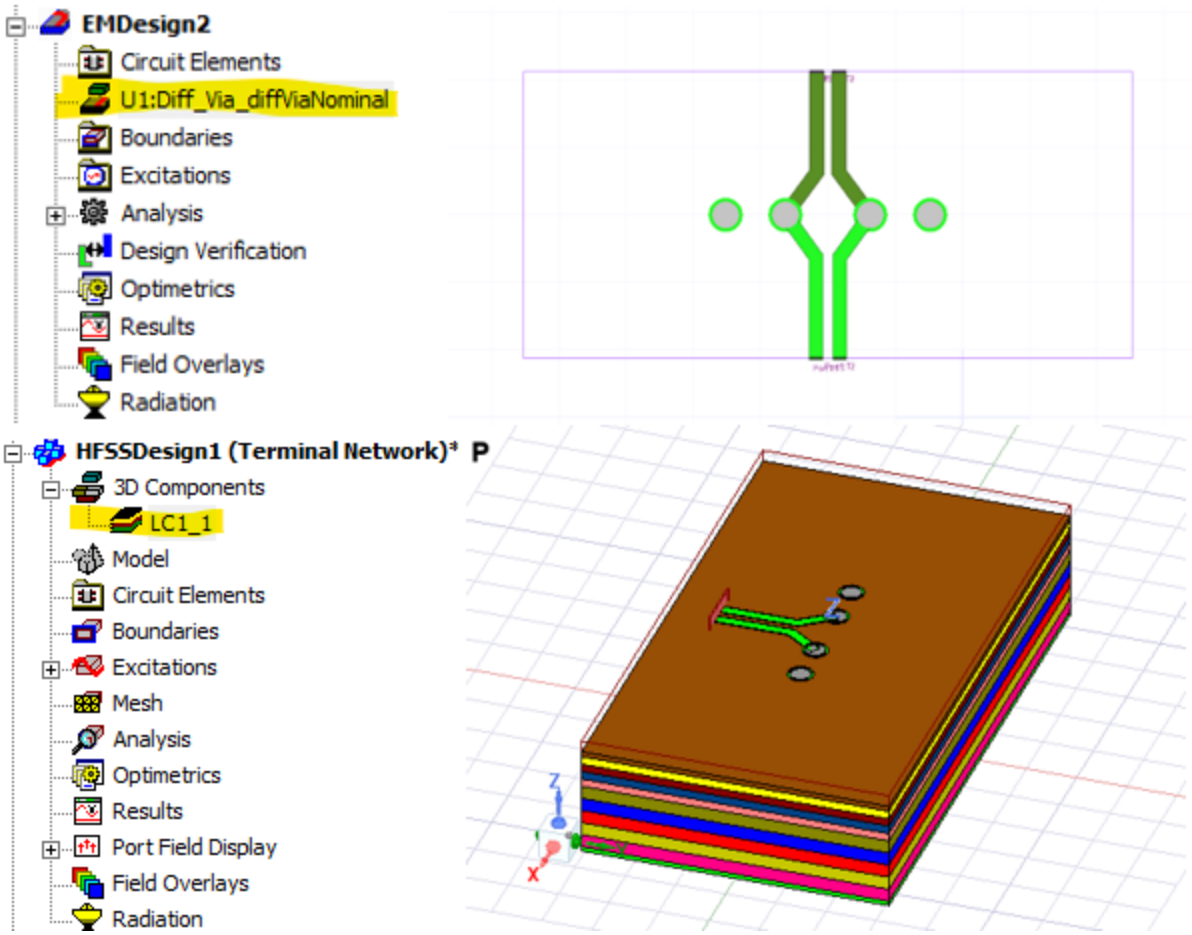


Once clicked, it opens the design containing the EDB definition of the layout component in a separate project to allow for any editing. Once a layout component is edited, either through a

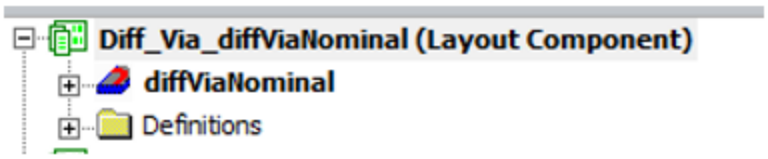
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parent 3D layout design or through this command, all designs of different types that use this component update it accordingly.

For example, in an Ansys Electronics Desktop project, multiple designs of different types can have layout components of the same definition. Below is an example project where both an HFSS 3D Layout design and an HFSS design have a layout component with the definition “Diff_Via_diffViaNominal”.

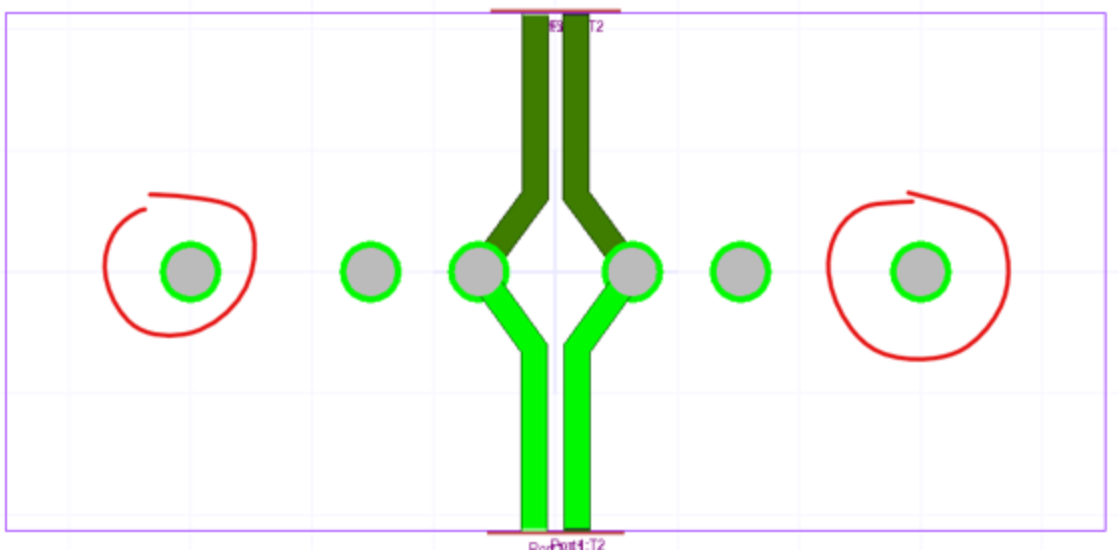


In addition to editing the source layout design using the **Edit Definition** command, you can edit the layout component's local settings, or replace it with a different component, using the **Replace Component** command.



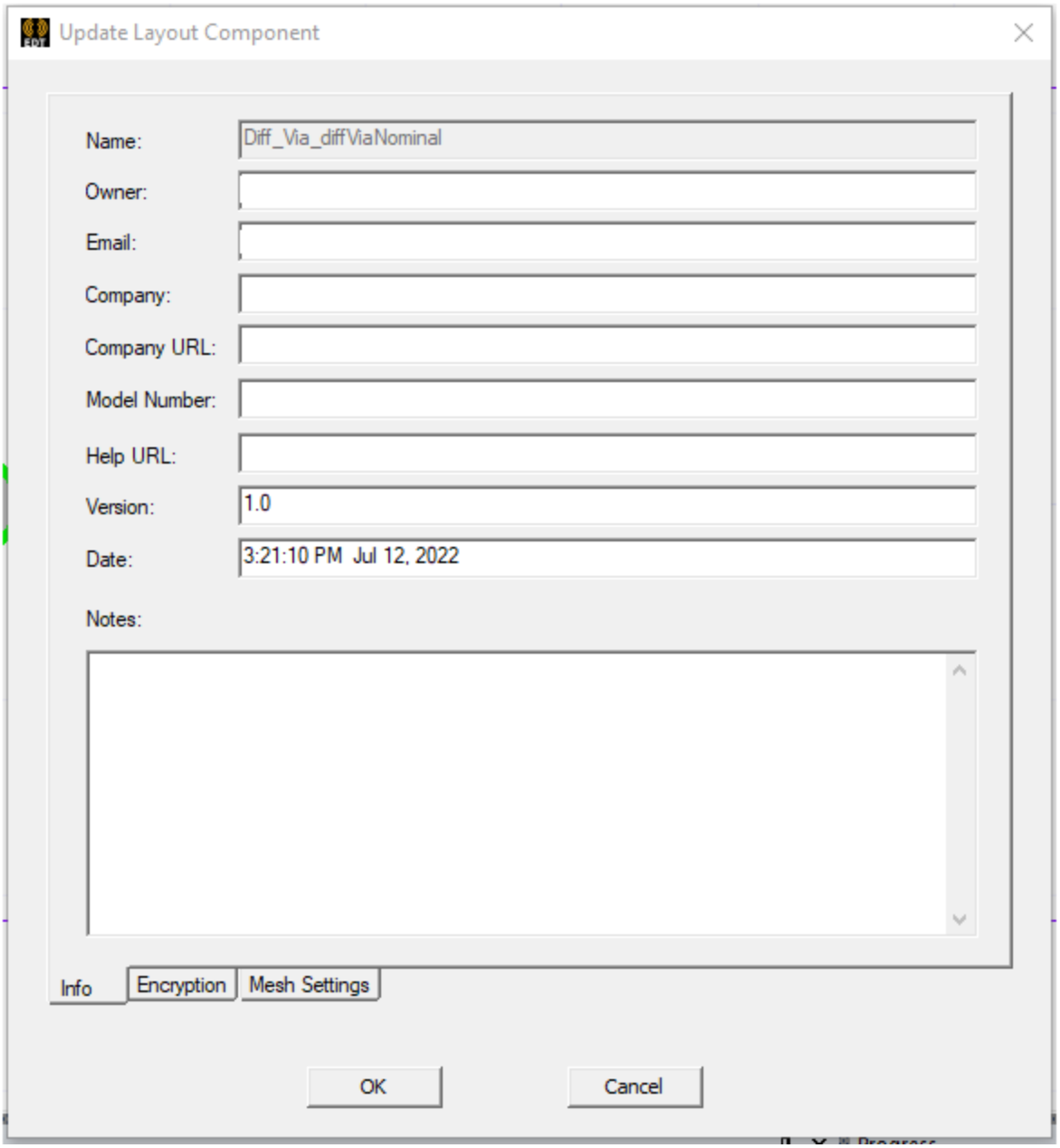
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the source design, you can edit the layout component however you want. This example adds two more vias on the left and right side as shown in the figure below.



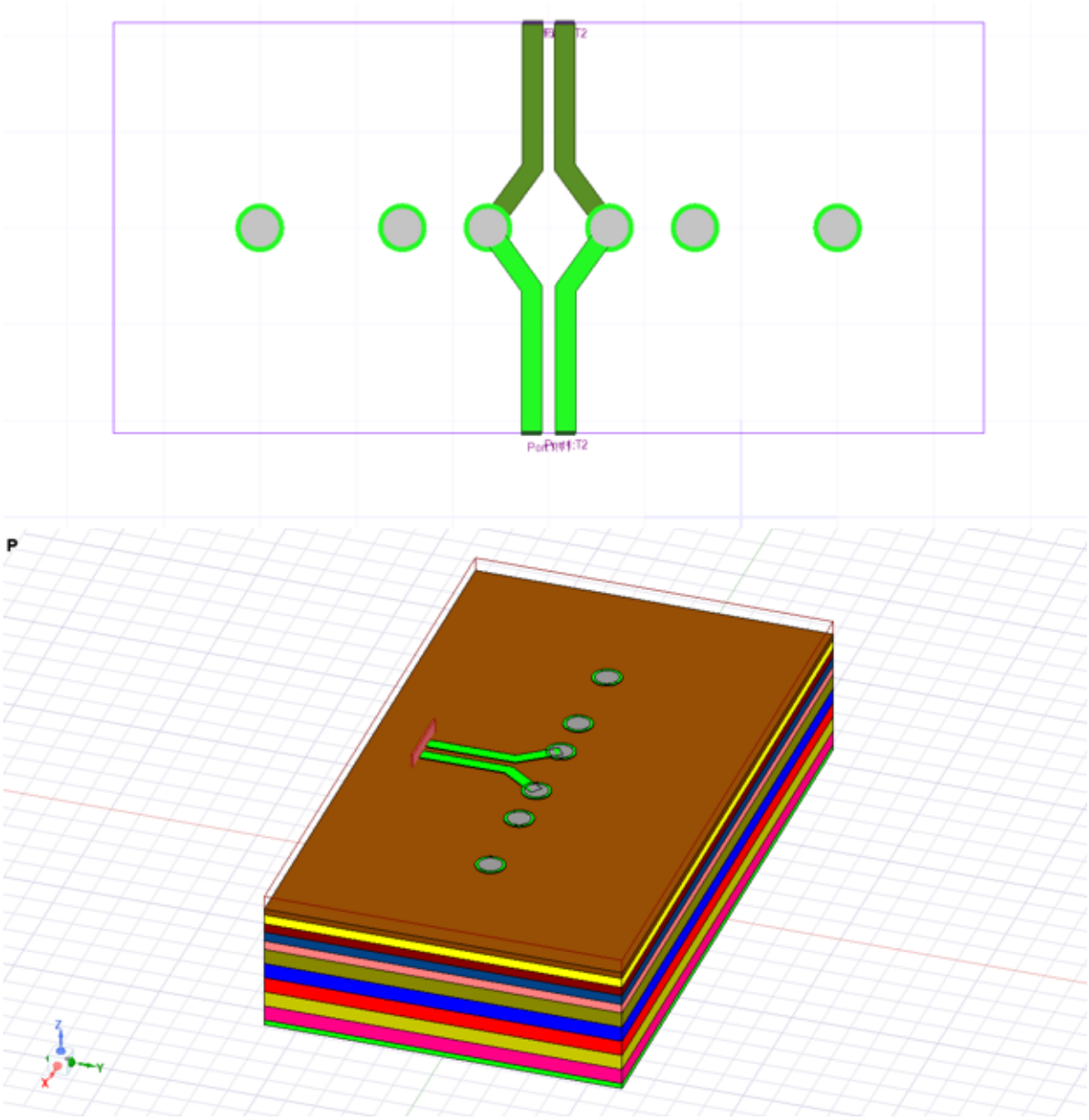
After editing, you can save the source design, and a dialog box opens for you to confirm the changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once the changes are confirmed, both the HFSS 3D Layout design and the Mechanical design containing this layout component update to reflect the changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

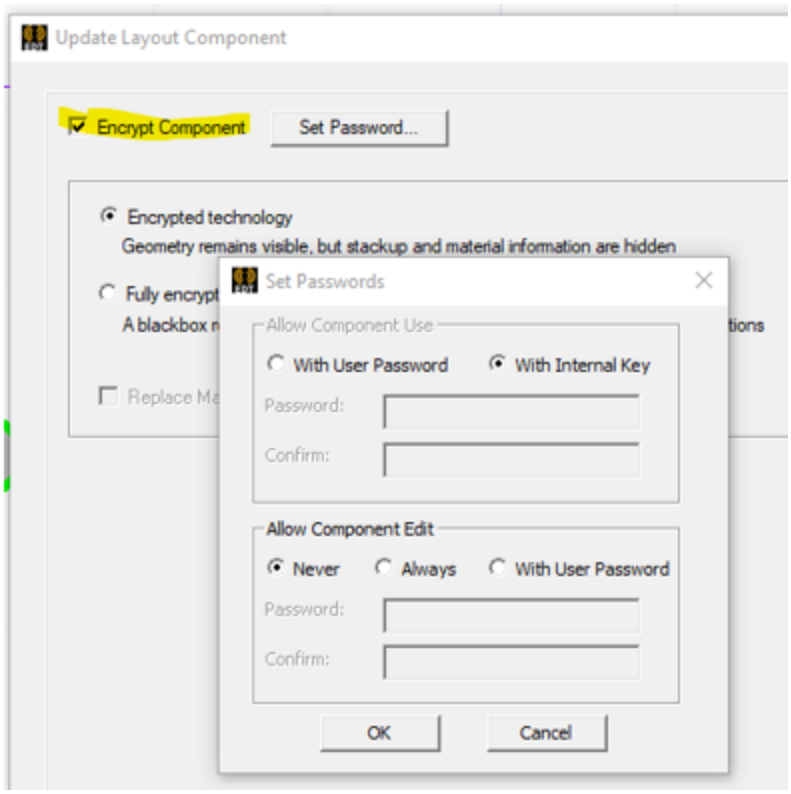


Note:

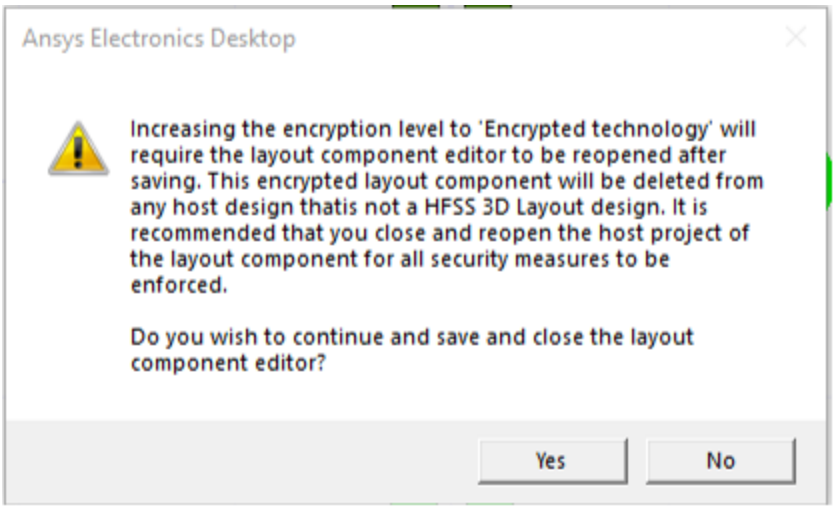
The editing is not undoable. After saving the source model, the target project is automatically saved, and the undo stack is cleared.

While editing, you may choose to encrypt the layout component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

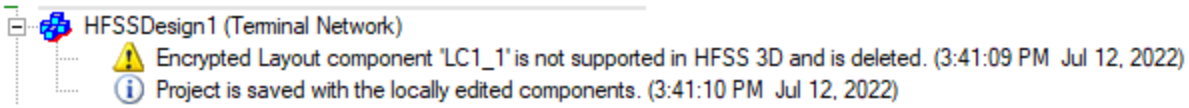


If the layout component is encrypted, a dialog box with warning messages alerts you about the potential impact to the designs containing this layout component.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After editing, all encrypted layout components are deleted from any target designs as encrypted layout components are not currently supported. A warning message appears for each deleted layout component.



Related Topics

Replacing or Editing Layout Components

Plotting the Mesh for Layout Components

Layout Component Properties

You must create a layout component using the *Insert Layout Component* dialog box (see Adding a Layout Component) and use the *Edit Layout Component* dialog box to replace or edit an existing layout component. However, you can also display the component's information and modify its visualization options using the docked *Properties* window. To do so, first select the layout component either in the Project Manager (under *3D Components*) or in the History Tree.

The topics below explain the contents of each tab of the docked *Properties* window:

- *General* Tab
- *Parameters* Tab
- *Component Data* Tab
- *Visualization* Tab

You can also access the same tabs, information, and settings in the *Properties* dialog box by right-clicking the layout component name in the Project Manager and choosing **Properties**.

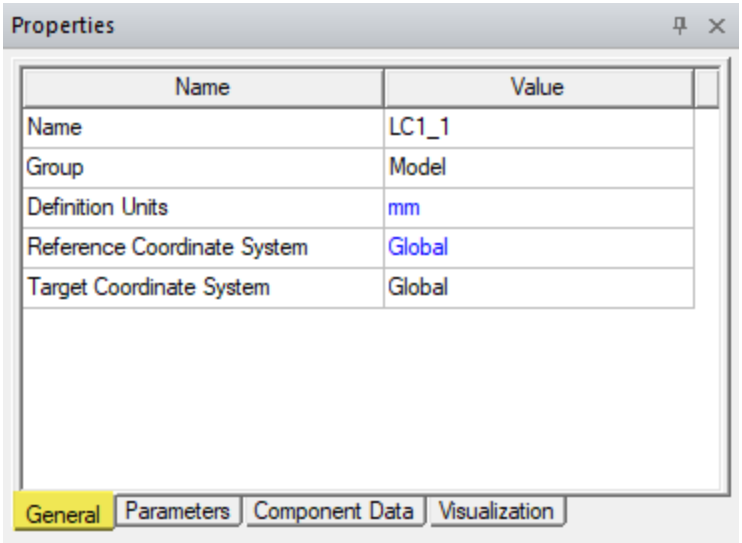
General Tab

The following items are shown in the *General* tab of the docked *Properties* window when a Layout Component is selected:

- **Name:** The name of the layout component (such as LC_1)
- **Group:** The group name within the History Tree where the component is listed.
- **Definition Units:** The units of the source HFSS 3D Layout design on which the layout component is based (not the current units of the Mechanical design)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reference Coordinate System:** The coordinate system of the source HFSS 3D Layout design
- **Target Coordinate System:** The coordinate system in the Mechanical design into which the layout component was inserted



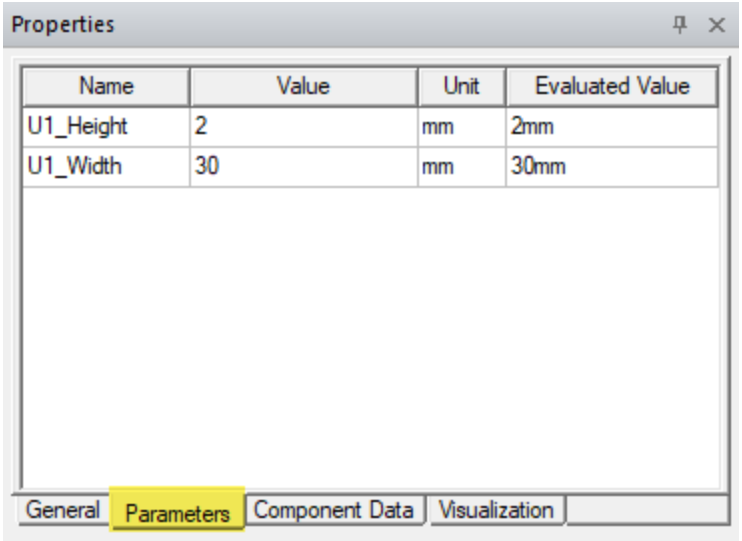
The items in this tab are for information only and cannot be edited from the docked *Properties* window.

Parameters Tab

This tab will only have contents when the source HFSS 3D Layout model has design variables defined, and those variables have been mapped to design variables in the target Mechanical model.

The following example shows the appearance of the *Parameters* tab when the layout component has two parameters defined:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

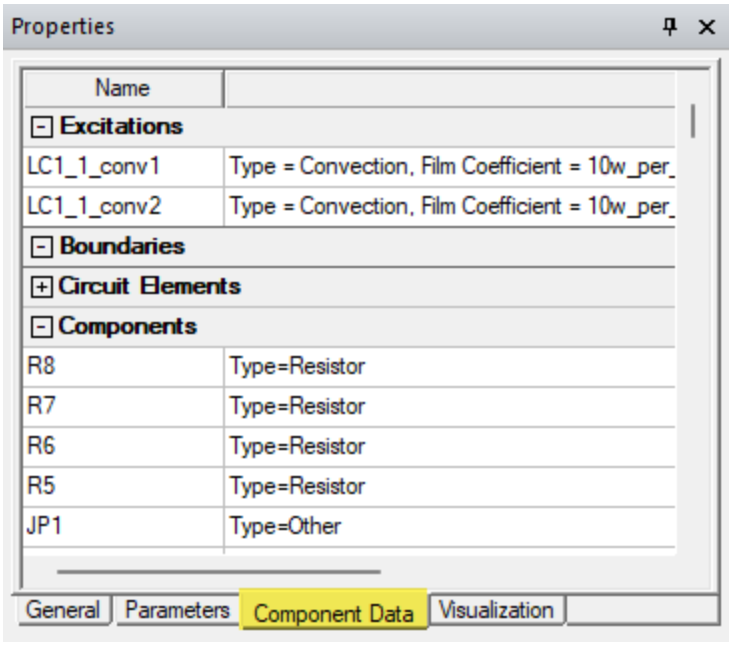


The **Name** column lists the name of the variable in the target Mechanical design. The name of the associated source design variable can only be seen, and the target variable assignment defined or changed, in the *Variable Mapping* step of the *Browse Layout Component* wizard. See Adding a Layout Component for more information.

Component Data Tab

This tab lists all **Boundaries** and **Excitations** assigned to the selected layout component as part of its definition. It also lists **Circuit Elements** (such as RLC networks) and individual **Components**. Circuit elements from the source design are not applicable to mechanical–thermal or –structural solutions. Similarly, components are not currently applicable to mechanicals design except for the conductor footprint (pads on a signal layer) that are associated with them.

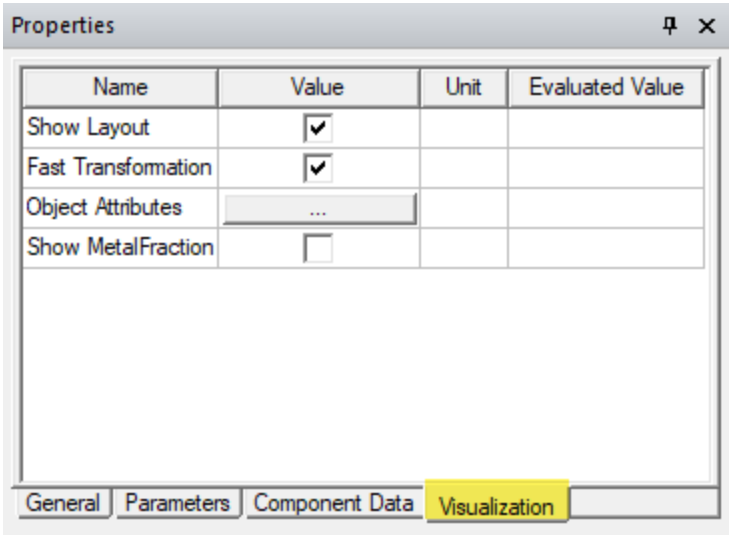
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The items in this tab are for information only and cannot be edited from the docked *Properties* window.

Visualization Tab

This tab contains various options controlling the visibility of layout component layers, objects, and the metal fraction map derived from the conductors:



The settings are defined as follows:

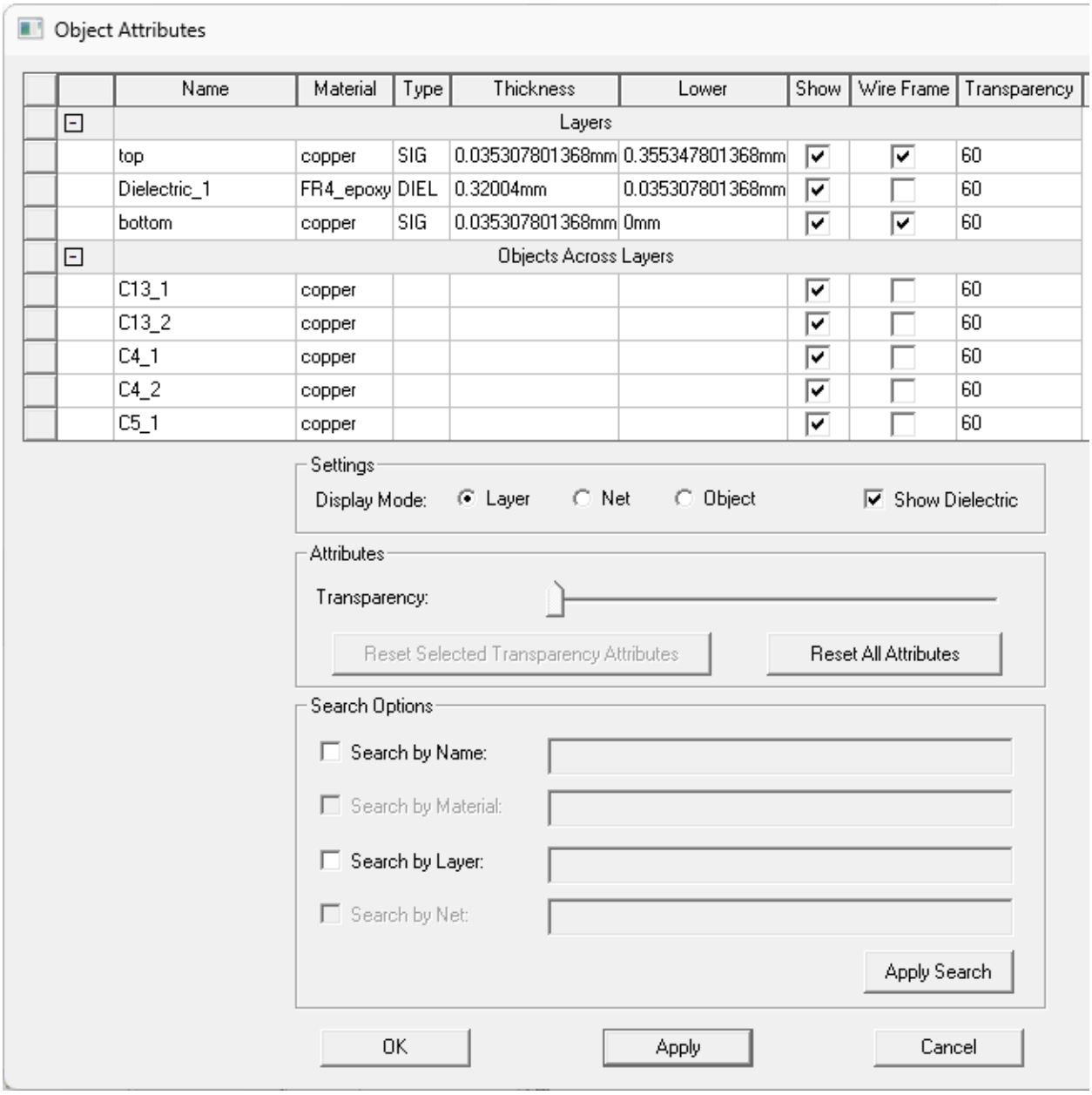
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Show Layout:** Clear this option to suppress the visibility of all layers and objects comprising the layout component, leaving only its outline wireframe visible.
- **Fast Transformation:** Improves graphical performance during view manipulation (zoom, pan, rotate) by suppressing the visibility of certain layout component objects (such as vias and cutouts). During view manipulations, both the outlines and shading of these objects is suppressed when the option is enabled.
- **Show MetalFraction:** Display a color overlay of the metal fraction data derived from the source layout's conductors. Blue represents 100% dielectric, and red represents 100% conductor, with the colors in between represent various partial conductor values. For example, consider a cell of the grid specified in the layout component definition that has a copper trace covering half of its area. In this case, the cell would be green (the middle color between the two extremes).
- **Layer:** This option is only visible when *Show MetalFraction* is selected (preceding bullet). Choose the printed circuit board layer on which to base the metal fraction contours from the drop-down menu that appears when you click in the *Value* column.

Object Attributes:

Click the ellipsis button (**...**) in this row to access the *Object Attributes* dialog box, from which you can control the visibility of objects comprising the layout component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- You can list objects by **Layer**, **Net**, or **Object** by selecting the desired **Display Mode** radio button.
- For each layer, net or object, specify the following visibility attributes:
  - **Show:** Select this option to make the item visible; clear to hide it.
  - **Wireframe:** This option is only applicable when *Show* is selected for the same item. Select it to display the item as an outline only (without shading). Clear to show a shaded view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Transparency:** Specify the percent transparency of the item; 0 is opaque, and 100 is fully transparent. This value has no effect unless *Show* is selected and *Wireframe* is cleared.

  > **Note:**
  >
  > For convenience, a common transparency can be specified for a set of selected items in a single operation. To do so, select the row of the table for which you wish to specify a common transparency value. Then, use the **Transparency** slider in the **Attributes** box to set the value for all selected rows.
  >
  > Similarly, you can **Reset Selected Transparency Attributes** for selected rows or **Reset All Attributes** to the default value (60) with a single click of one of these two buttons.

- **Show Dielectric:** Controls the visibility of all dielectric layers and dielectric fill materials in the layout component. When cleared, all dielectrics are hidden, and the listing of dielectric layers is suppressed (that is, they don't appear in the table at the top of the dialog box when hidden). Select the option to list the dielectrics, in which case the visibility attributes can be specified for each layer individually.
- **Search Options:** The available search options depend on your selected *Display Mode*. Select the checkbox for the desired **Search by** choice, type the string you want to search for in the adjacent text box, and click **Apply Search**. The options are:
  - **Search by Name**
  - **Search by Material**
  - **Search by Layer**
  - **Search by Net**

  After clicking *Apply Search*, the table at the top of the dialog box is filtered according to the search string.

  To restore the full, unfiltered list, clear the selected search mode checkbox and click **Apply Search** again. You need not clear the search string.

Technical Notes: Metal Fraction Map

When you add a layout component to a mechanical design, the materials listed in the History Tree are those contained in the source ECAD design on which the mechanical layout component is based. These materials typically include at least a dielectric (such as FR4) and the conductor in signal layers and vias (such as copper). However, the History Tree materials do not directly represent the thermal or structural material properties that are passed to the solver for the mechanical simulations. The mechanical layout component is comprised of the dielectric

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

materials with their local properties altered element-by-element to account for the conducting traces, pads, and vias. Electrical conductors have much higher thermal conductivity than dielectrics and are also much stiffer under structural loads. A metal-fraction map derived from the source layout is used to determine the average material properties.

The metal fraction map is divided into a grid with the number of columns and rows specified when you add the layout component to the mechanical design. The metal fraction is calculated for each cell in this grid. The method of accounting for the conductor is straightforward. For example, if 65% of a cell's volume is conductor, and 35% is dielectric, the resulting metal fraction for that cell is 0.65.

Vias pass through both dielectric and signal layers. For mechanical layout components, only cylindrical vias are currently supported. The *Plating Percent* specified in the source HFSS 3D Layout's padstack definition determines how much of the via's **radius** is filled with the plating material.

Example: For 25% plating, the outer 40% of the radius is the via plating material, and the inner 60% of the radius is the specified via fill material. These % radius values correspond to 54% (plating) and 36% (fill) express as portions of the via's cross-sectional area.

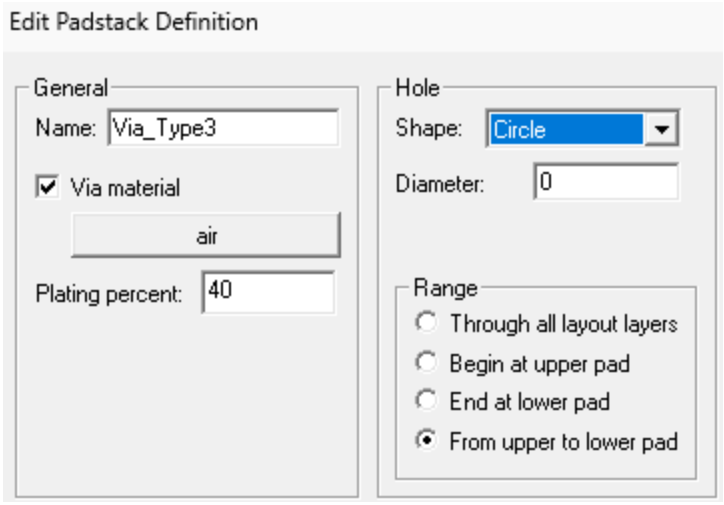
> **Note:**
>
> Vias are handled differently from traces, pads, and conductor pour areas. The metal fraction value for a grid cell containing a via is always 1.0, regardless of the *Plating Percent* specified in the source layout. Rather than reducing the metal fraction for vias with dielectric fill materials, the material properties are modified. Conducting and dielectric material properties are averaged to represent the true composition of the via, as detailed below.

A weighted average conductivity for vias is calculated based on the ratios of the plating and hole areas to the total area, according to the following equation:

$$k_{Total} = (k_{ViaPlating} * Area_{ViaPlating} + k_{ViaHole} * Area_{ViaHole}) / Area_{Total} \quad [Eq. 1]$$

The figure below shows the definition of the via's *Percent Plating*value in the *Edit Padstack Definition* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The weighted average material properties are calculated for each grid cell based on the metal fraction of the cell and the properties of the constituent conductor and dielectric materials. For cells containing vias, the material properties are calculated as described in the preceding note.

The same method as shown in Eq. 1 for thermal conductivity is used to determine the average of all other applicable material properties. Simply substitute the desired property for **k**. The applicable material properties depend on the solution type, as follows:

- **For Thermal Solutions:**
  - Thermal Conductivity (k)
  - Mass Density (ρ) – Relevant only for transient solutions
  - Specific Heat (c) – Relevant only for transient solutions
- **For Structural Solutions:**
  - Young's Modulus (E)
  - Poisson's Ratio (v)
  - Mass Density (ρ) – Though this property is provided to the solver, in the absence of gravity, acceleration, or rotational velocity excitations (currently unsupported), the material's mass density is not relevant to linear static structural solutions.
  - Shear Modulus (G)
  - Thermal Expansion Coefficient (α)

Finally, the weighted average material properties are mapped from the grid cells to the elements comprising the model mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 - Assigning Materials

For Mechanical Designs:

The information in this topic and its subtopics is general and applicable to multiple design types. For **Mechanical** designs, the material properties depend on the solution type (*Modal, Thermal,* or *Structural*). Therefore, separate *Assigning Materials* topics are included within the *Modal Solutions, Thermal Solutions,* and *Structural Solutions* branches of the Mechanical help. Access these topics using the following links to see the mechanical-specific material assignment information:

- Assigning *Modal* Materials
- Assigning *Thermal* Materials
- Assigning *Structural* Materials

General Information:

You can add, remove, and edit materials in two main ways:

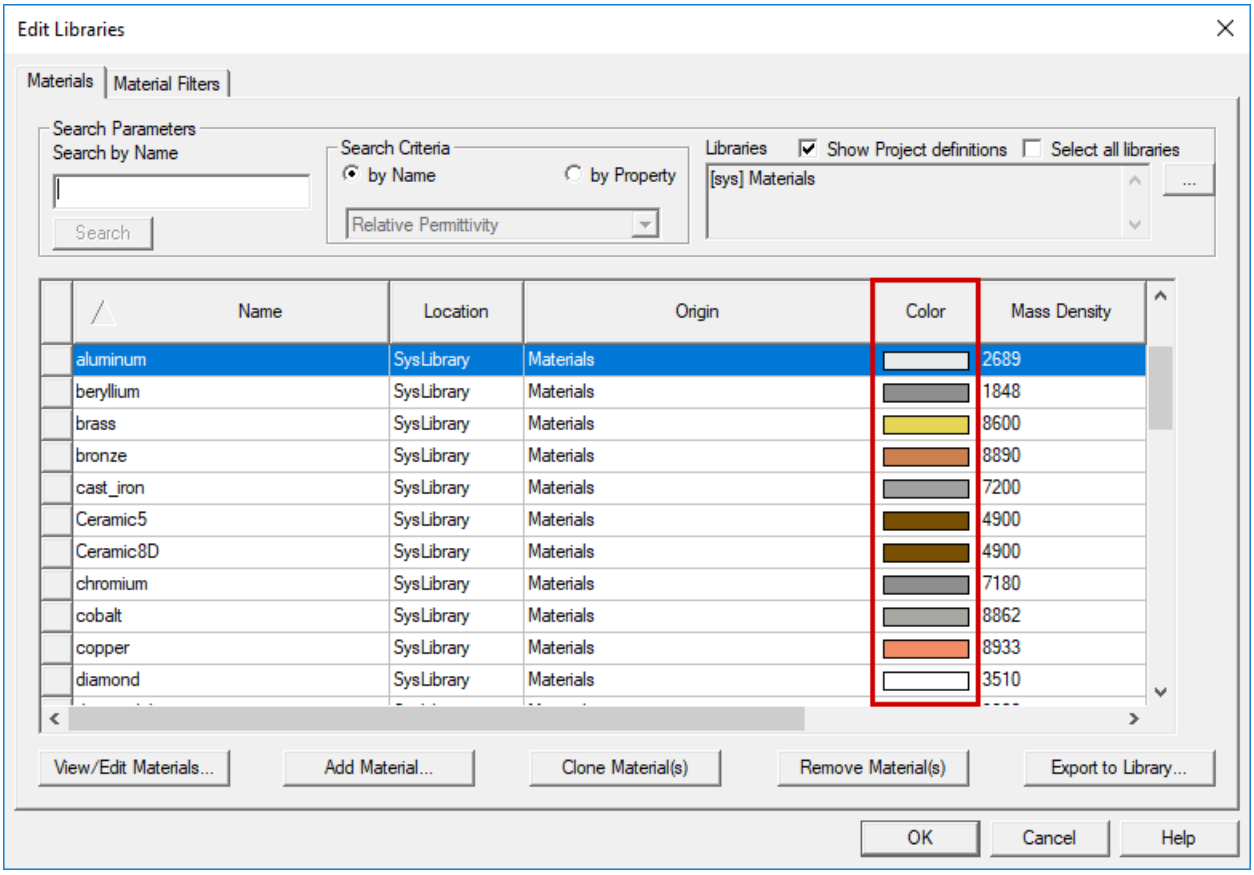
- Using the **Tools** > **Edit Libraries** > **Materials** menu command.
- Under **Definitions** in the Project Manager, right-click **Materials** and select **Edit Library**.

Regardless of which of the preceding two methods you use to edit a library, any new material you create exists only in the current project. Similarly, if you edit an existing material, the edited version becomes a *Project* material and exists only within the current project. To make a new or modified material available for other projects, you must export it to a user library and choose that library (or select **Show all libraries**) within the **Select Definition** dialog box.

> **Note:**
>
> In the History Tree, by default, objects are grouped by material. You can easily change the default grouping. Using the menu bar, click **Modeler** > **History Tree Layout** and clear **Group Objects by Material** along with any other listed option you may wish to disable. You can also perform this action from the **Tree View** drop-down menu on the **View** ribbon tab or from the shortcut menu that appears when you right-click most of the items in the History Tree.

If you check **Show Material Colors** on the Material Filters tab of the **Select Definition** or **Edit Library** dialog box, the **Materials** tab will include a **Color** column showing a color swatch for each listed material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

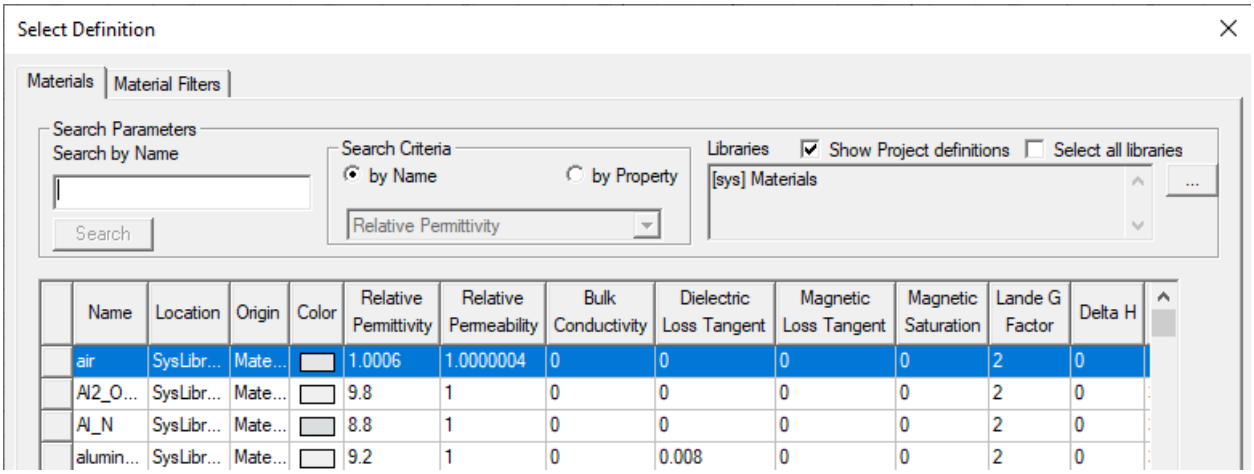


You can edit the color and transparency values for materials in the View/Edit Material dialog box.

Searching for Materials

You can search for materials in the **Select Definition** dialog box. The default Search Criteria is **by Name**, which is shown in the following example. Alternatively, you can choose to search by Property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

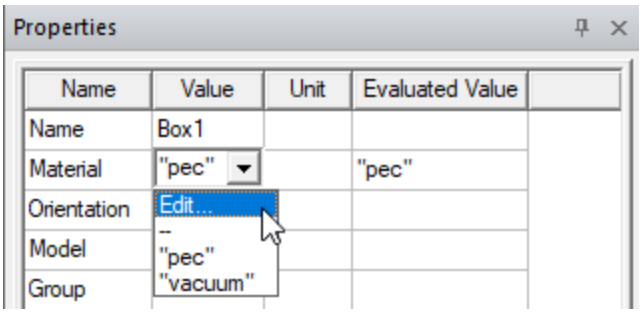


To search for a material **by Name**:

1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**
   - Click the **Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.

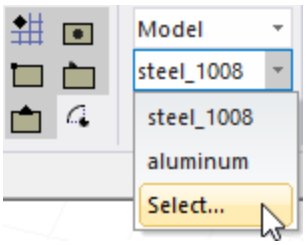


   - On the **Draw** ribbon tab, click **Assign material**.

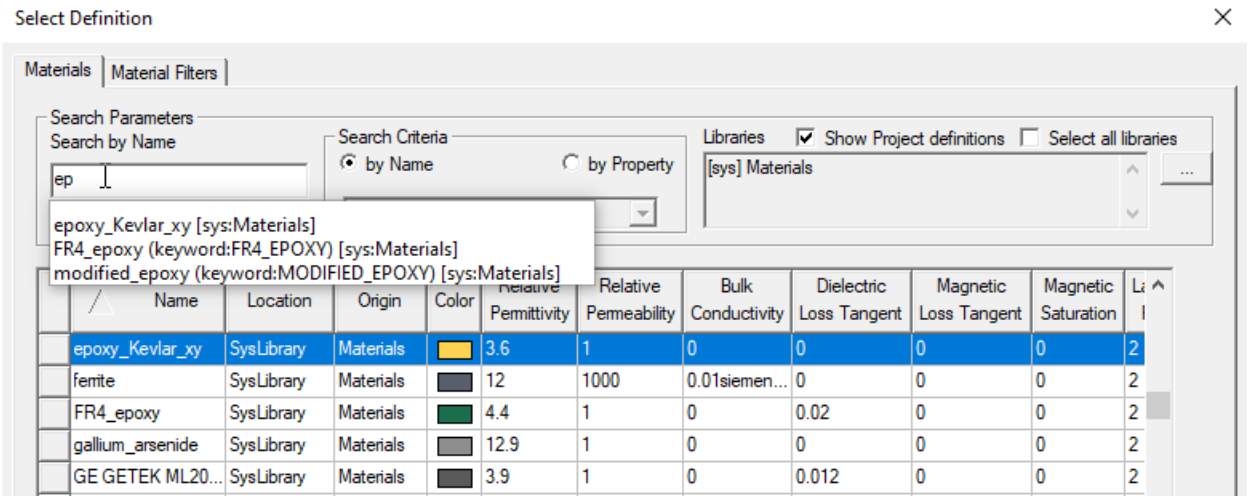
   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Search Criteria** section, ensure that **by Name** is selected, and in Libraries, specify the Libraries that you want to search. Only loaded libraries participate in text and keyword matching.
3. In the **Search by Name** text box, type a portion of the desired material name. The search text is case-insensitive.



A drop down under the Search Parameters field will show the a list of the materials from selected libraries matching any part of the current text, and, if the Granta Materials library is selected, matching keywords in the material definitions. The first row containing the material name most similar to the characters you typed will be selected.

If the selected material is not the one you are searching for, do one of the following:

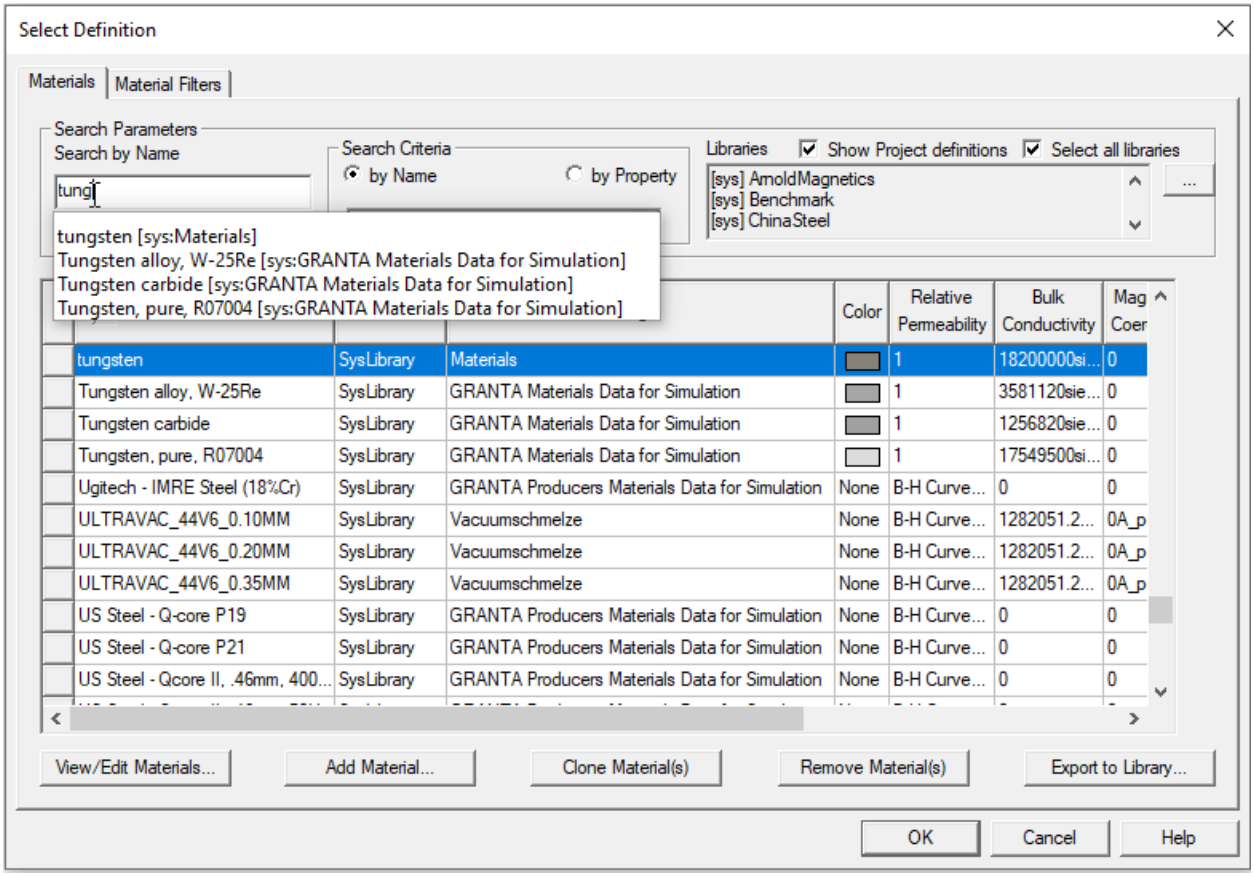
- Use the keyboard’s arrow keys to select the material above or below the currently selected row.
- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type different characters in the **Search by Name** text box.

.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design type as well as those in the project's local materials library. The Granta Materials Data for Simulation library now includes keywords that are used in the auto-complete matching. For example, entering "glass" will generate an auto-complete list of all materials with a name containing "glass" or with a keyword that contains "glass", all case-insensitive, for materials showing in the grid. The drop down of potential matches under the search field applies to all libraries. What shows in the grid is defined by which libraries are selected, what material filters are selected, project/design-specific validation, and whether project materials are selected for display.



Searching by Material Property

1. Access the **Select Definition** dialog box using one of the following methods:

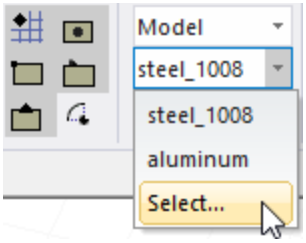
   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**
   - Click the **Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.

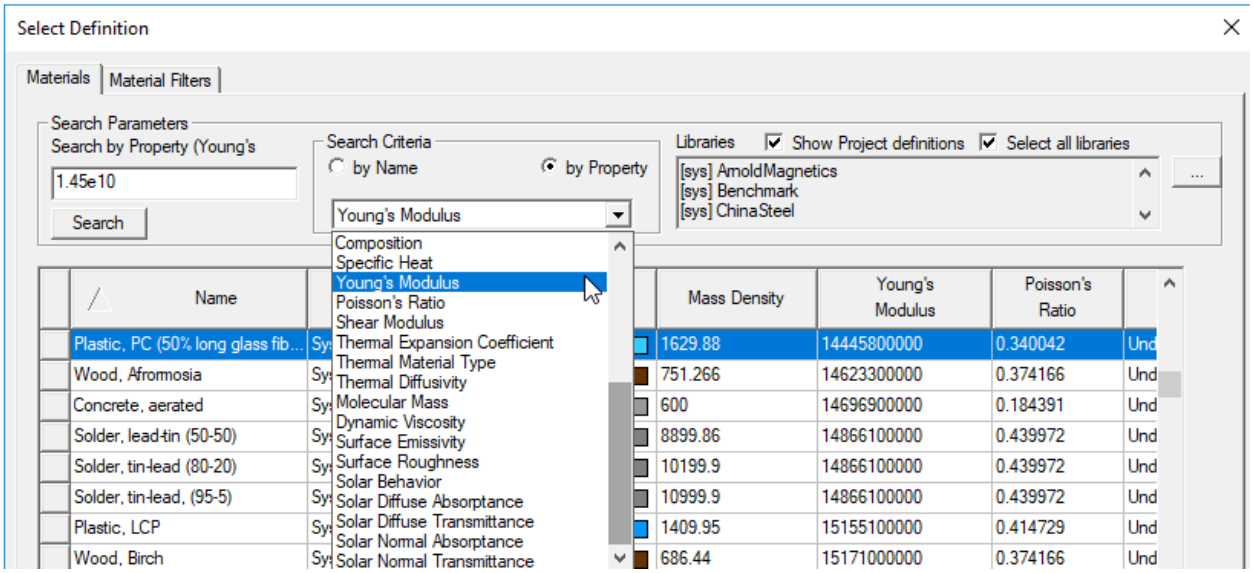
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With nothing selected (that is, to set the default material):

- On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:



2. In the **Search Criteria** section, select **by Property**.
3. Select a material property from the pull-down list:



> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed in the table, though all properties are available in this drop-down menu. To view the complete table of properties, see Filtering Materials.

4. In the **Search Parameters** area, type a numerical value in the **Search by Property** text box and then click **Search**.

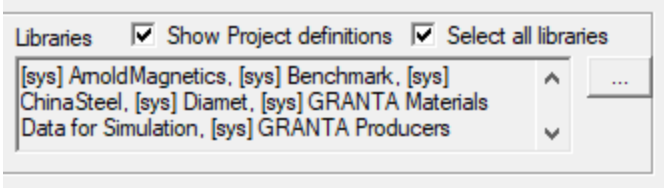
   The materials are sorted according to the property you selected. Additionally, the material with the property value closest to the one you typed, but without exceeding it, is selected.

   If the selected material is not the one you are searching for, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the keyboard's arrow keys to select the material above or below the currently selected row.
- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type a different numerical in the **Search by Property** text box and click **Search** again.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current design type as well as those in the project's local materials library.



Adding New Materials

You can add a new material to a project or global user-defined material library. To make the new project material available to all projects, you must export the material to a global user-defined material library.

Materials are added using the **View/Edit Material** dialog box, which can be opened from either the **Select Definition** window or the **Edit Libraries** window.

To open the **Select Definition** window:

- Click **Modeler** > **Assign Material**.

  The **Select Definition** window appears.

To open the **Edit Libraries** window:

- Click **Tools** > **Edit Libraries** > **Materials**. In the project tree, you can also right-click **Materials**, and select **Edit Library**.

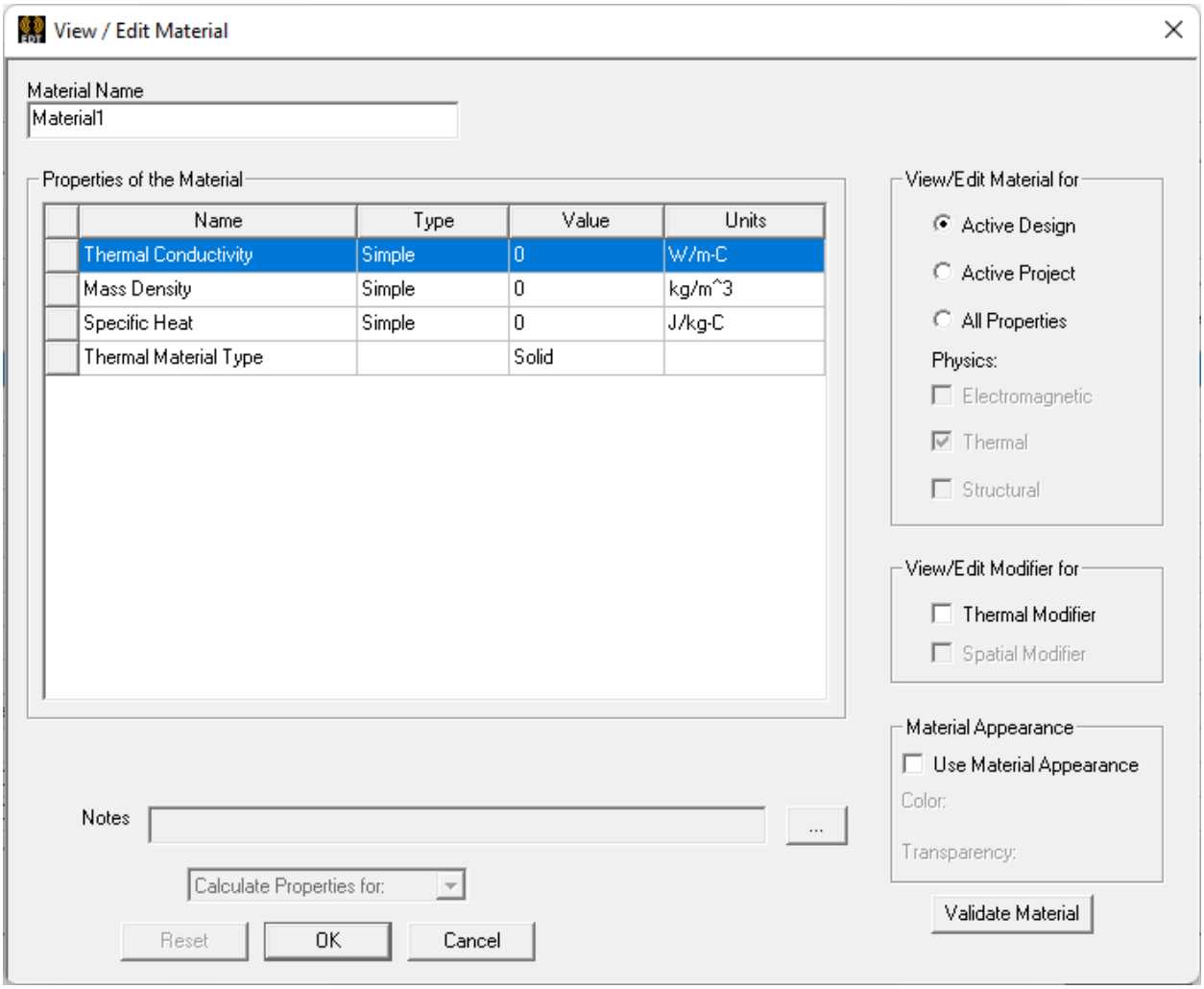
  The **Edit Libraries** window appears.

To add a new material:

1. From either the **Select Definition** window or the **Edit Libraries** window, click **Add Material**.

   The **View/Edit Material** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(The preceding image is based on the *Thermal* solution type.)

By default, only properties commonly used by the selected product are displayed. To view the complete table of properties, see Filtering Materials.

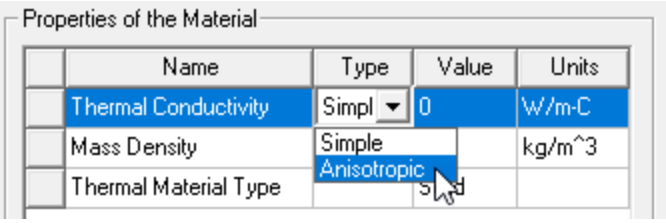
2. Type a name for the material in the **Material Name** text box, or accept the default.
3. Use the radio buttons in the **View/Edit Material for** section to specify whether the new materials apply to Active Design, Active Project, or All Properties. When **All Properties** is selected, **Physics** classification options are enabled to show or hide properties based on simulation type (Electromagnetic, Thermal, or Structural).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If a material is edited in a design type for which the **Physics** type has not been set (e.g., an HFSS design but Electromagnetic physics type was not set), the **Physics** type will be automatically set in the material.

You can also enable the View/ Edit Modifier check box for Thermal Modifier. Checking this box causes the Thermal Column to display at the right side of the Properties of the Material table. Selecting **Edit** rather than None causes display of the Edit Thermal Modifier dialog.

4. Enter a **Material Name**, or accept the default.
5. For each available material property, select a **Type** from the drop-down menu. Only applicable types appear for each material property. Some properties only use the Simple type, while others offer four or more types.



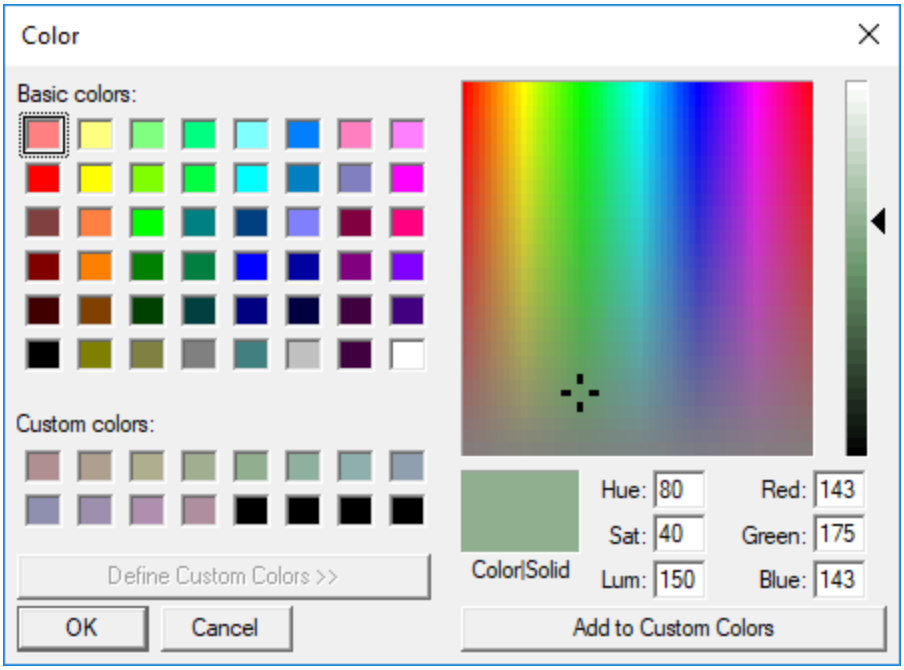
(The preceding image is based on the *Thermal* solution type.)

6. Enter a **Value** for the following material properties:
   - Thermal Conductivity (Thermal solutions only)
   - Mass Density
   - Young's Modulus (Modal solutions only)
   - Poisson's Ratio (Modal solutions only)

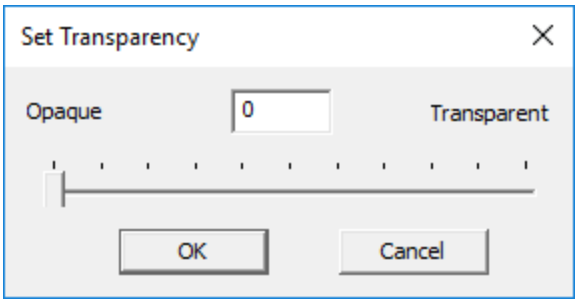
> **Note:**
>
> You may enter a variable name or mathematical expression in the **Value** box.

7. If one or more of the material properties are dependent on frequency, click **Set Frequency Dependency**, and then follow the directions for defining frequency dependent materials.
8. To modify the units for a material property, double-click the **Units** box and select a new unit system.
9. For Material Appearance, you can check the box to enable the fields for you to specify a color and transparency. Clicking the color bar opens a color selection window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

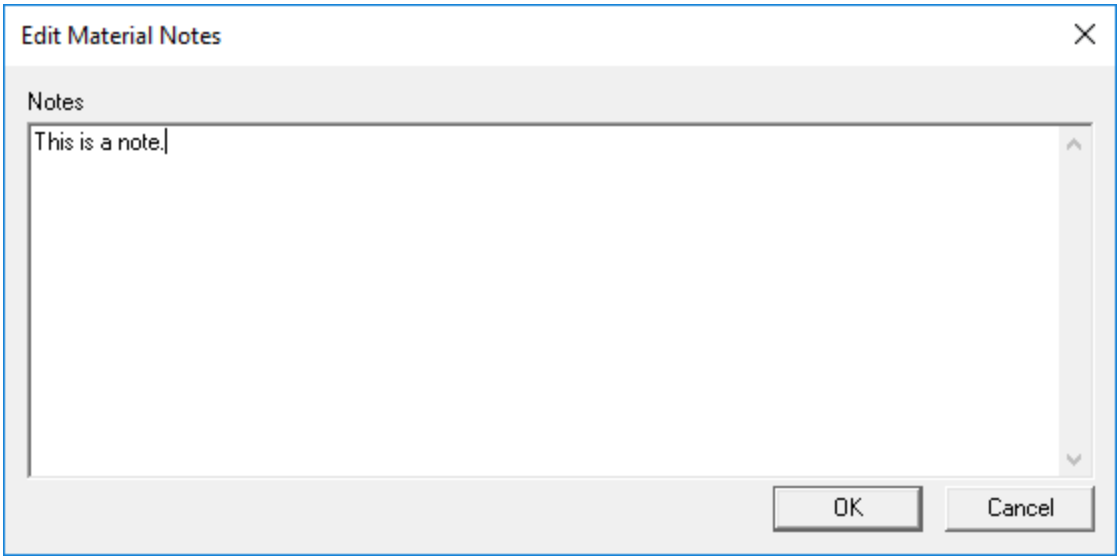


10. Clicking the Transparency box opens a Transparency dialog box with a text field and slider bar for selection.



11. Click **OK**.
12. To modify the units for a material property, double-click the **Units** box and select a new unit system.
13. Click **OK**.
14. If you want to add descriptive notes for the new material, click the ellipsis button [...] next to the **Notes** field. This opens a dialog box in which you can enter text.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



15. Click **OK** to add the Notes.
16. Click **OK** on the **View/Edit Materials** dialog to add the new material to the material library.

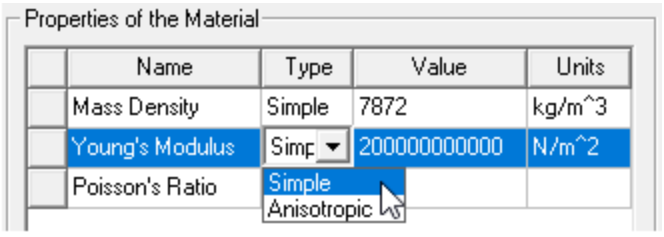
Assigning Material Property Types

Material properties can be assigned using the **View/Edit Material** dialog box, using one of the following material property types, based on the applicability to the material property:

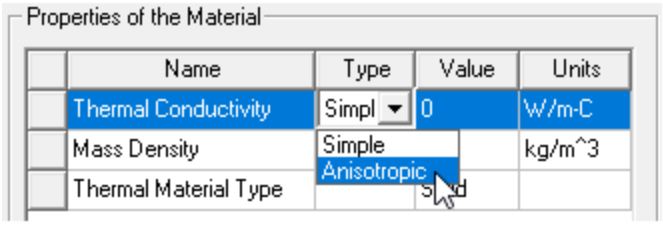
Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction. This models only the diagonal.
Tensor	A full 3x3 matrix general tensor, which may or may not be symmetrical.
Nonlinear	A BH Curve.

Select a material property type for each property from the **Type** drop-down menu. Some properties only use the Simple type

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(Modal Solutions)



(Thermal Solutions)

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic Young's Modulus and Poisson's Ratio or Thermal Conductivity. Each diagonal represents a tensor of your model along an axis.

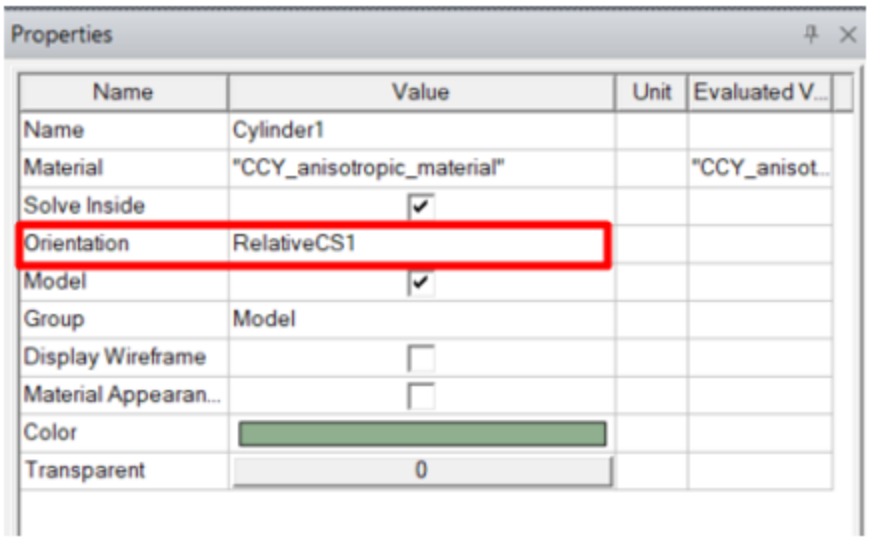
These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

Defining Anisotropic Tensors

If a material property is anisotropic, its characteristics are defined by its anisotropy tensor. Each diagonal represents a tensor of your model along an axis. These tensors are relative to the coordinate system specified as the object's *Orientation* property (when supported). Otherwise, the tensors conform to the global Cartesian coordinate system. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently. Spherical and Cylindrical tensors must be converted to Cartesian coordinates, as described below. In such cases you must specify the coordinate system carefully.

> **Note:** Spatial-dependent material properties are calculated according to coordinates in the 'Orientation CS'.This CS should be set carefully.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

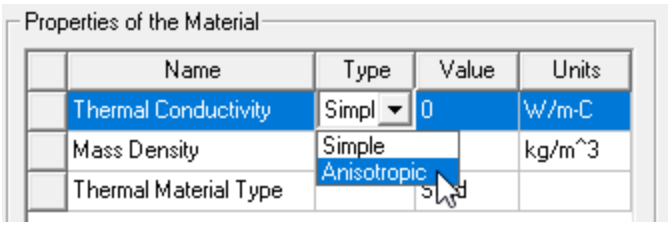


> **Important:**
>
> Anisotropic materials are supported in 2D models but are not available for Q3D.

Assigning Anisotropic Tensors

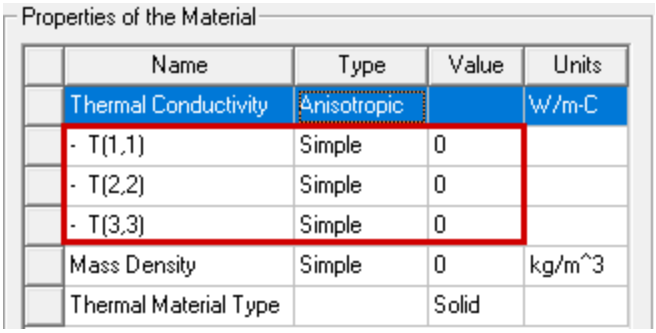
To assign anisotropic tensors to Young's Modulus and Poisson's Ratio or Thermal Conductivity:

1. From the **View/Edit Material** window, use the **Type** drop-down menu to select **Anisotropic**.



   Three rows, labeled **T(1,1), T(2,2),** and **T(3,3)** appear, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(The preceding images are based on the *Thermal* solution type.)

2. For each of the new anisotropic property rows, use the **Type** drop-down menu to select either **Simple** or **Nonlinear** (when supported). This setting determines the type of **Value** you can enter.
3. In the **Value** field, enter the Young's Modulus and Poisson's Ratio or Thermal Conductivity. along each axis of the material's tensor.
    - For **Young's Modulus** – Enter the modulus along the direction of each tensor axis. Tensors T(1,1), T(2,2), and T(3,3) correspond to the global Cartesian X, Y, and Z axes, respectively.
    - For **Poisson's Ratio** – This property quantifies how a strain in one direction produces a strain along an axis 90 degrees from the first axis. Enter the ratio for each axis pair. Tensors T(1,2), T(1,3), and T(2,3) correspond to the global Cartesian axis pairs XY, XZ, and YZ, respectively.
    - For **Thermal Conductivity** – Enter the thermal conductivity along the direction of each tensor axis. Tensors T(1,1), T(2,2), and T(3,3) correspond to the global Cartesian X, Y, and Z axes, respectively.
4. Click **OK** to save the values and return to the **Select Definition** window.

Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS

Mechanical accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Cylindrical and Cartesian bases are related as

$$\begin{bmatrix} \hat{e}_x \\ \hat{e}_y \\ \hat{e}_z \end{bmatrix} = \begin{bmatrix} cos\varphi & -sin\varphi & 0 \\ sin\varphi & cos\varphi & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \hat{e}_\rho \\ \hat{e}_\varphi \\ \hat{e}_z \end{bmatrix} = \boldsymbol{M}_{Cyl \to Cart} \begin{bmatrix} \hat{e}_\rho \\ \hat{e}_\varphi \\ \hat{e}_z \end{bmatrix}$$

where M is a transformation matrix.

Tensors, describing material properties, could be transformed accordingly as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Cyl\to Cart}\bar{\bar{A}}_{\rho\varphi z}\boldsymbol{M}^{T}_{Cyl\to Cart} = \begin{bmatrix} cos\varphi & -sin\varphi & 0 \\ sin\varphi & cos\varphi & 0 \\ 0 & 0 & 1 \end{bmatrix}\bar{\bar{A}}_{\rho\varphi z}\begin{bmatrix} cos\varphi & sin\varphi & 0 \\ -sin\varphi & cos\varphi & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

If we assume only the diagonal components as non-zeros for the tensor in the Cylindrical basis

$$\bar{\bar{A}}_{\rho\varphi z} = \begin{bmatrix} A_{\rho\rho} & 0 & 0 \\ 0 & A_{\varphi\varphi} & 0 \\ 0 & 0 & A_{zz} \end{bmatrix}$$

the resulting tensor in the Cartesian basis could be derived as

$$\bar{\bar{A}}_{xyz} = \begin{bmatrix} A_{\rho\rho}cos^2\varphi + A_{\varphi\varphi}sin^2\varphi & \left(A_{\rho\rho} - A_{\varphi\varphi}\right)sin\varphi cos\varphi & 0 \\ \left(A_{\rho\rho} - A_{\varphi\varphi}\right)sin\varphi cos\varphi & A_{\rho\rho}sin^2\varphi + A_{\varphi\varphi}cos^2\varphi & 0 \\ 0 & 0 & A_{zz} \end{bmatrix}$$

We should define and using X, Y and Z:

$$\rho = \sqrt{X^2 + Y^2}$$

$$\varphi = \begin{cases} \text{atan}\left(\frac{Y}{X}\right) + \begin{cases} 180°\,\text{sign}(Y) & if \quad X < 0 \\ 0 \quad if\ X \geq 0 \end{cases} & if \quad \rho > 0 \\ \square \\ 0 \quad if \quad \rho = 0 \end{cases} \qquad -180° \leq \varphi \leq 180°$$

Related Mechanical Project variables:

$rho	sqrt(X^2+Y^2)
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

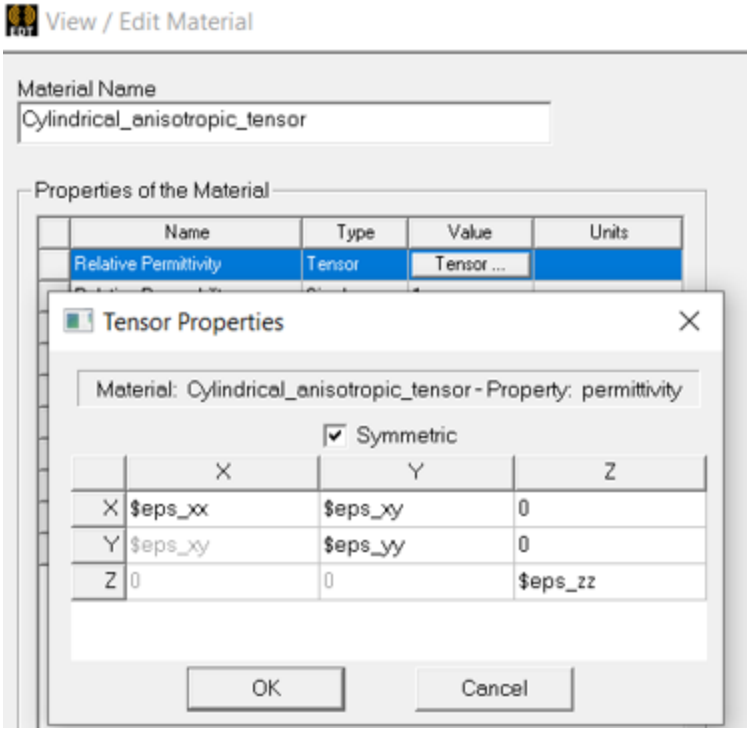
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The material characteristics close to Z-axis could be slightly incorrect because of uncertainty of in the case of =0.

First you create all required Project variables.

Name	Value	Unit	Evaluated V...
$eps_rr	4		4
$eps_pp	8		8
$eps_zz	12		12
$rho	sqrt(X^2+Y^2)		*****
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)		*****
$eps_xx	$eps_rr*cos($phi)^2+$eps_pp*sin($phi)^2		*****
$eps_xy	(eps_rr-eps_pp)*sin($phi)*cos($phi)		*****
$eps_yy	$eps_rr*sin($phi)^2+$eps_pp*cos($phi)^2		*****

You can then use the variables to create the material.

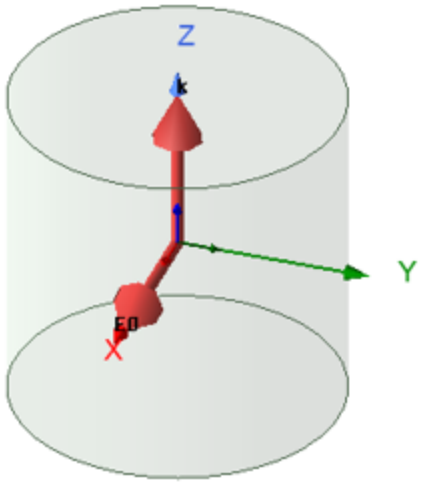


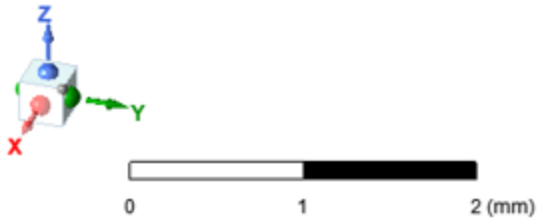
For the test case for cylindrical anisotropic material definition, using converted tensors based on variables:

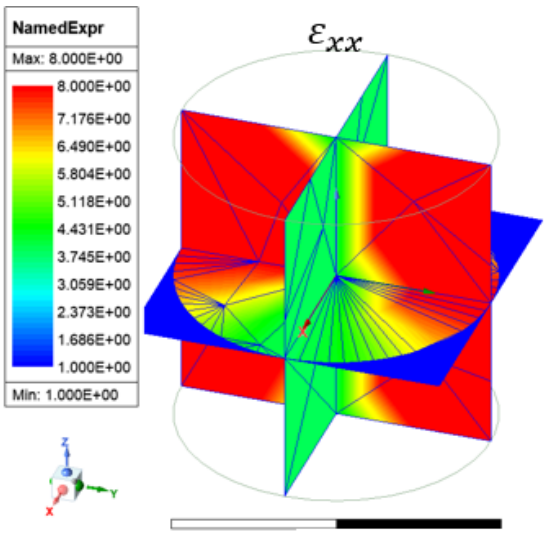
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

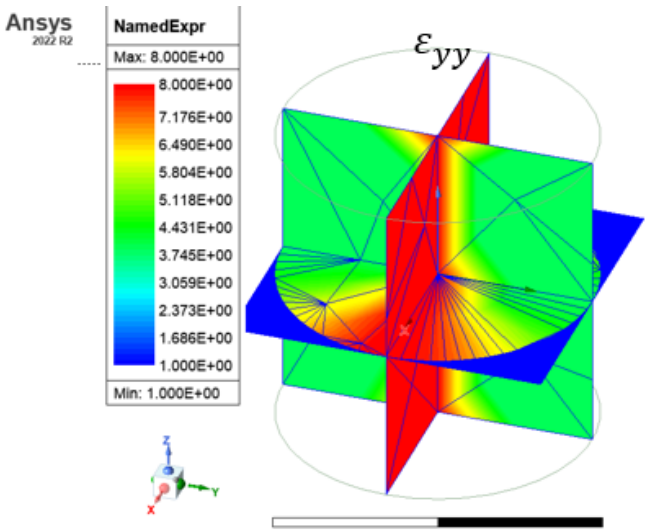
$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\varphi\varphi} = 8 \;\; \varepsilon_{zz} = 12$

For this problem type, consider that scattering on a cylinder where diameter and height are equal to 0.2 wavelength and a plane wave excitation along Z-axis.

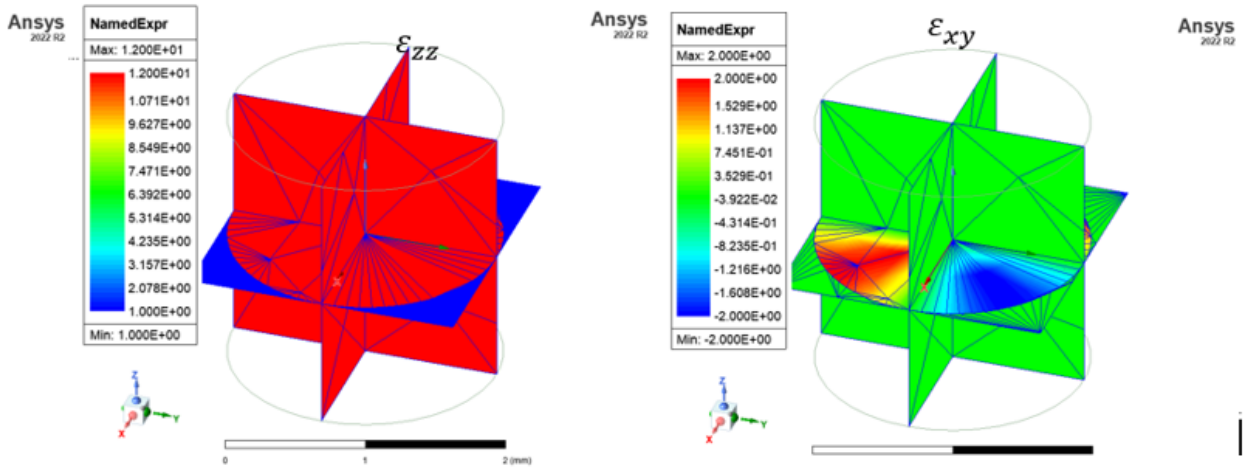








Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS

Mechanical accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Spherical and Cartesian bases are related as

$$\begin{bmatrix}\hat{e}_x\\ \hat{e}_y\\ \hat{e}_z\end{bmatrix} = \begin{bmatrix}sin\theta cos\varphi & cos\theta sin\varphi & -sin\varphi\\ sin\theta sin\varphi & cos\theta cos\varphi & cos\varphi\\ cos\theta & -sin\theta & 1\end{bmatrix}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix} = \boldsymbol{M}_{Sph\to Cart}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix}$$

where **M** is a transformation matrix

Tensors, describing material properties, could be transformed accordingly as

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Sph\to Cart}\bar{\bar{A}}_{\rho\theta\varphi}\boldsymbol{M}^T_{Sph\to Cart}$$

We should define ρ,θ , and φ using X, Y and Z

$$\rho = \sqrt{X^2+Y^2+Z^2} \qquad \rho_{XY} = \sqrt{X^2+Y^2}$$

$$\varphi = \begin{cases}\text{atan}\left(\frac{Y}{X}\right) + \begin{cases}180°\,\text{sign}(Y) & if\ \ X<0\\ 0 & if\ X\geq 0\end{cases} & if \quad \rho_{XY}>0\\ \\ 0 & if \quad \rho_{XY}=0\end{cases}$$

$$\theta = \begin{cases}\text{acos}\left(\frac{Z}{\rho}\right) & if \quad \rho>0\\ \\ 0 & if \quad \rho=0\end{cases}$$

$$0 \leq \theta \leq 180°$$
$$-180° \leq \varphi \leq 180°$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related Mechanical Project variables

$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

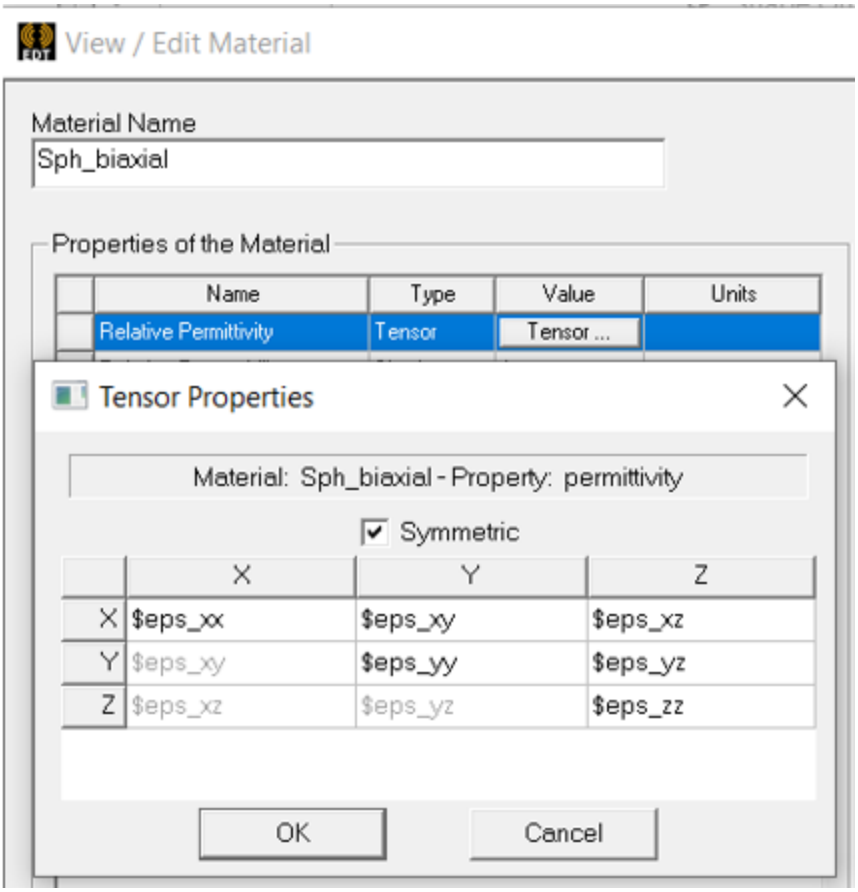
Note: the material characteristics close to the Z-axis could be slightly incorrect because of uncertainty of φ in the case of ρ_{xy}=0.

First you must create the necessary project variables.

Name	Value
$eps_rr	4
$eps_tt	8
$eps_pp	12
$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)
$eps_xx	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*cos($phi)^2+$eps_pp*sin($phi)^2
$eps_xy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2-$eps_pp)*sin($phi)*cos($phi)
$eps_xz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*cos($phi)
$eps_yy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*sin($phi)^2+$eps_pp*cos($phi)^2
$eps_yz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*sin($phi)
$eps_zz	$eps_rr*cos($theta)^2+$eps_tt*sin($theta)^2

You can then use the variables to create the material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



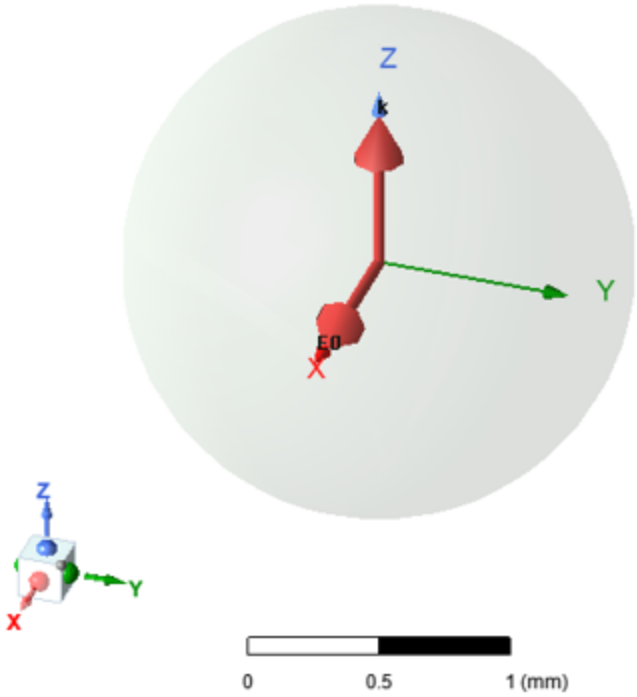
For the test case:

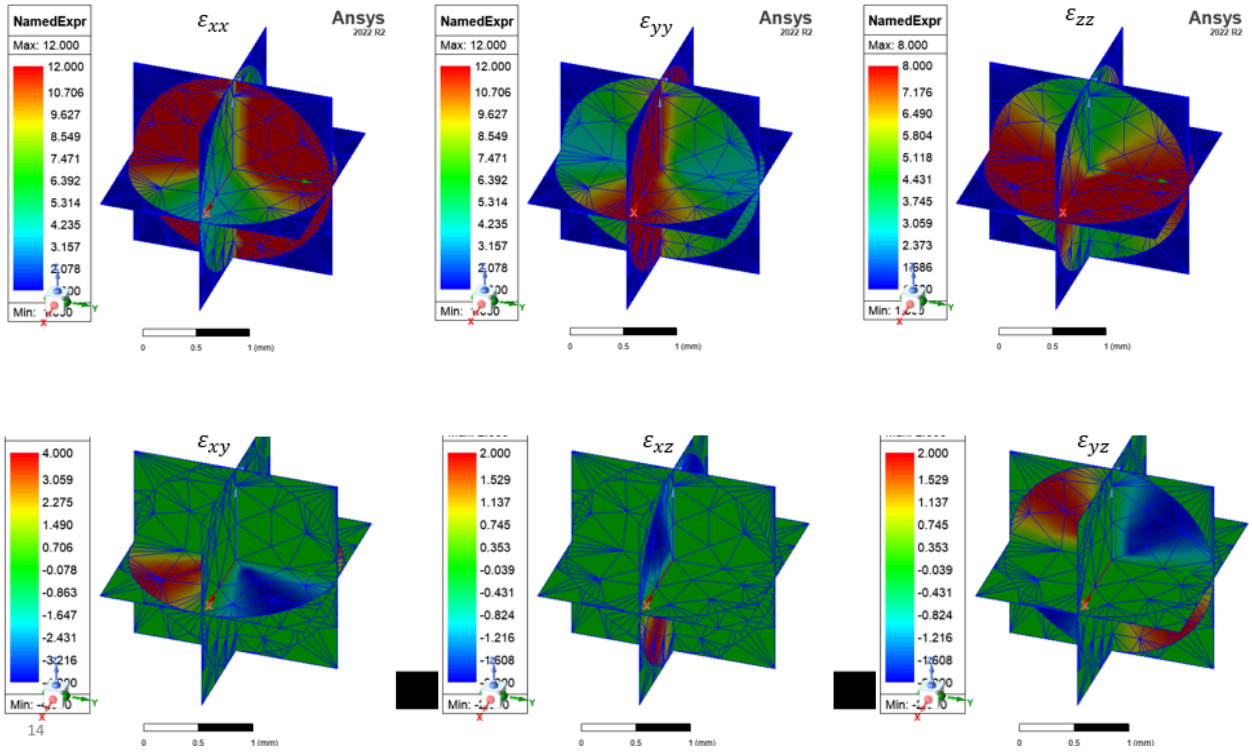
$$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\theta\theta} = 8 \;\; \varepsilon_{\varphi\varphi} = 12$$

Problem type: scattering on a sphere(diameter is equal to 0.2 wavelength).

Plane wave excitation along Z-axis

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Variable Material Properties

When defining or modifying a material's properties, each material property value in the **View/Edit Material** dialog box can be assigned a project variable. Simply type the project variable's name in the appropriate **Value** box. Project variables are used for material properties because materials are stored at the project level.

For example, define a project variable with the name **MyYoungsModulus** and define its value as **4e10**. To assign this property value to a material, type **$MyYoungsModulus** in the **Young's Modulus** box for the material. Be sure to include the prefix **$** before the project variable name, which notifies the software that the variable is a project variable.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

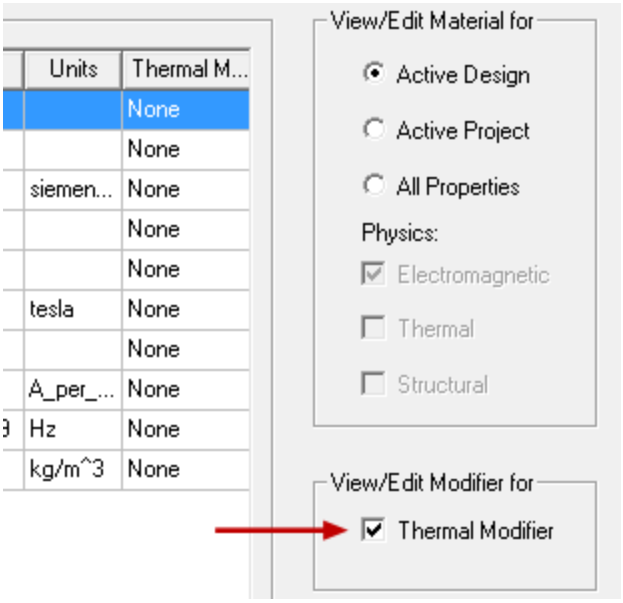
Specifying Thermal Modifiers

Thermal modifiers are multiplicative in nature and assigning them introduces spatial dependence. Because you cannot have spatial dependence and causality, Ansys Electronics Desktop ignores Auto Causal Materials options when a thermal modifier exists.

To specify thermal modifiers for a material:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

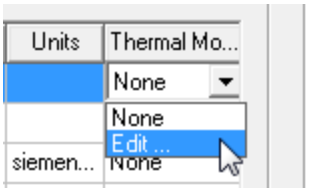
1. In the **View/Edit Material** dialog box, you must enable the **Thermal Modifier** check box:



This option causes the properties table to expand to include a Thermal Modifier column.

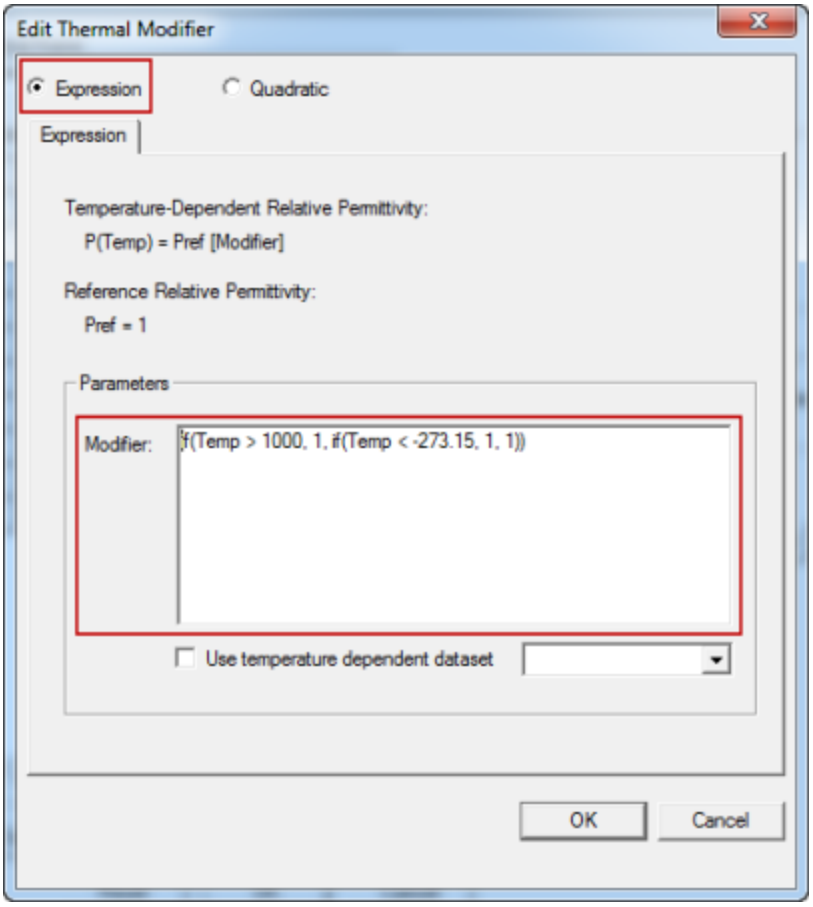
By default, the **Thermal Modifier** option is set to **None** for all material properties.

2. Click a **Thermal Modifier** cell and select **Edit...** from the drop-down menu:



The **Edit Thermal Modifier** dialog box appears with the **Expression** option selected.

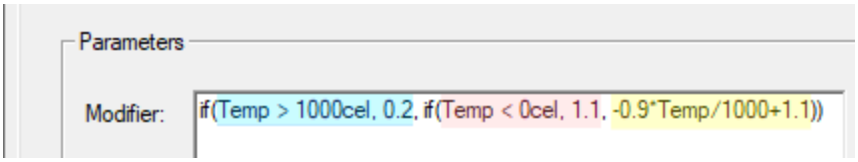
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With this option selected, the **Modifier** text box is displayed under **Parameters**. This text box contains a conditional expression template, which you can use as a starting point for creating your own expression. The property for which the multiplier is being defined and its nominal value are indicated above the **Parameters** section.

- With **Expression** selected, you can write an equation for a thermal modifier in the **Modifier** text box.

  The following example defines a multiplier that varies linearly from 1.1 (at 0° C) to 0.2 (at 1000° C). The multiplier is constant at 1.1 for temperatures below 0° C and constant at 0.2 for temperature above 1000° C:

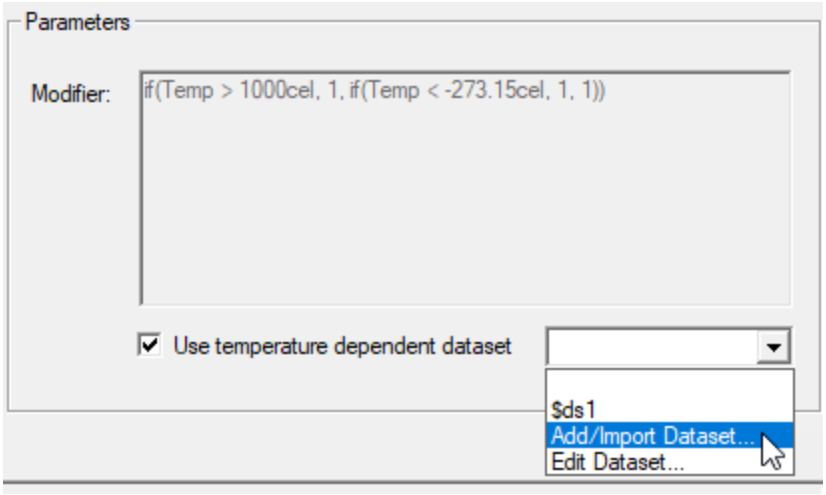


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, the linear equation **-0.9 * Temp / 1000 + 1.1** (highlighted in yellow) defines the multiplier for the range 0° C ≤ Temp ≤ 1000° C.

The default unit of temperature in thermal modifier expressions is Celsius (cel). However, you can override the units by appending the appropriate abbreviation to the numbers in the expression (mkel, ckel, dkel, kel, cel, rank, fah). Model temperatures, material properties, and thermal modifiers need not have consistent units; values are converted automatically as needed.

- Check **Use temperature dependent data set** to disable the Modifier text box. You can then use the drop-down menu at the bottom-right corner of the dialog box to select an existing project dataset or to create a new one by choosing **Add/Import Dataset**.

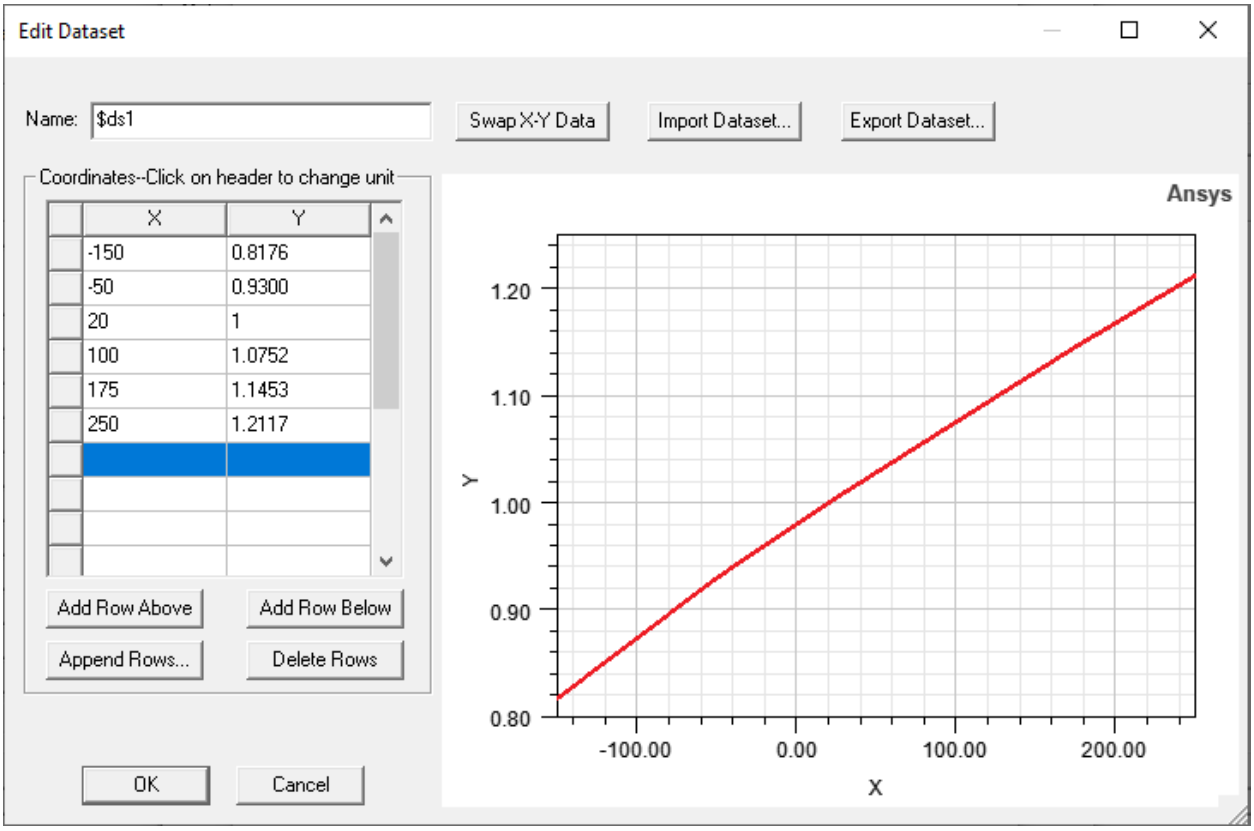


This command lets you define the thermal modifier dataset (a set of X and Y values). Alternatively, if a suitable project dataset already exists, it will be available for selection from the same drop-down menu.

When creating a new dataset, the **X** values are temperatures, and the **Y** values are the corresponding thermal modifiers. The value of the nominal material property is multiplied by this thermal modifier to determine the resultant property at a given temperature. The modifier is interpolated linearly between specified data points.

The following example is a dataset defining thermal multipliers for the thermal expansion coefficient (TEC) of an aluminum-beryllium alloy. The temperature range is -150 to 250° C, and the nominal value (corresponding to 20° C) is 1.37e-5/°C:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Based on these multipliers, the resultant TEC at -150° C is approximately 1.12e-5 (= 1.37e-5 * 0.8176), and the TEC at 250° C is 1.66e-5 (= 1.37e-5 * 1.2117).

In addition to filling the table of values manually, you can also **Import** an existing tab-delimited file to create the dataset.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

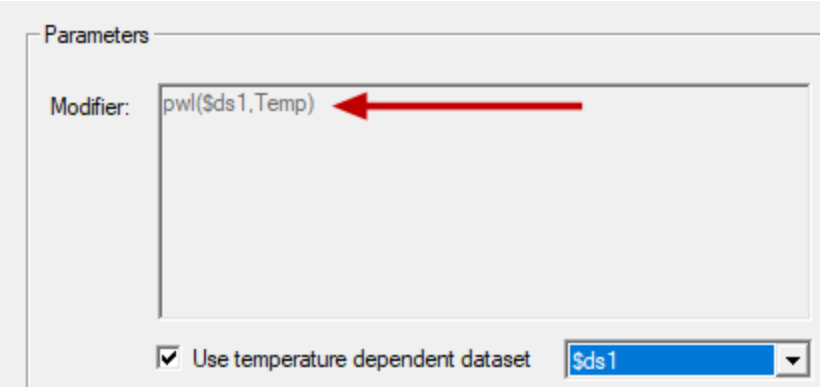
Important:

When you add a dataset from the **Edit Thermal Modifier** dialog box, you cannot specify the temperature units. The numbers in the **X** column will be interpreted as degrees Celcius and cannot be changed. Also, you cannot append a unit abbreviation (such as "fah" or "cel") to the numbers. You must enter temperature values in degrees Celcius for correct results. The values in the **Y** column are multipliers and therefore dimensionless.

Alternatively, you can create a dataset based on any desired temperature units before defining your temperature-dependent material properties. To do so, click **Project> Datasets** from the menu bar and then choose **Add**. If your project includes a design type that supports 3D datasets, a submenu appears offering a choice of a 1D or 3D dataset. If you see this choice, click **1D**. In either case, the **Add Dataset** dialog box appears.

For the dataset to be used later as a thermal modifier, you must click the **X** column header and choose **Temperature**. Then, click the **X** column header again and choose the desired temperature units. Finally, populate the table with your temperature and multiplier values and then click **OK** and **Done**. The dataset will now be available for selection from the drop-down menu in the **Edit Thermal Modifier** dialog box.

After choosing or creating a dataset, it appears in the **Modifier** field of the **Edit Thermal Modifier** dialog box, as shown in the following image:



The syntax of the dataset expression is explained as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **pwl:** Type of dataset – Piecewise linear
- **$ds1:** Name of an existing dataset – "$" indicates a *project* dataset, which is a requirement for thermal modifiers
- **Temp:** Type of X value – Temperatures

  The Y values in this case are simple multipliers and therefore unitless.

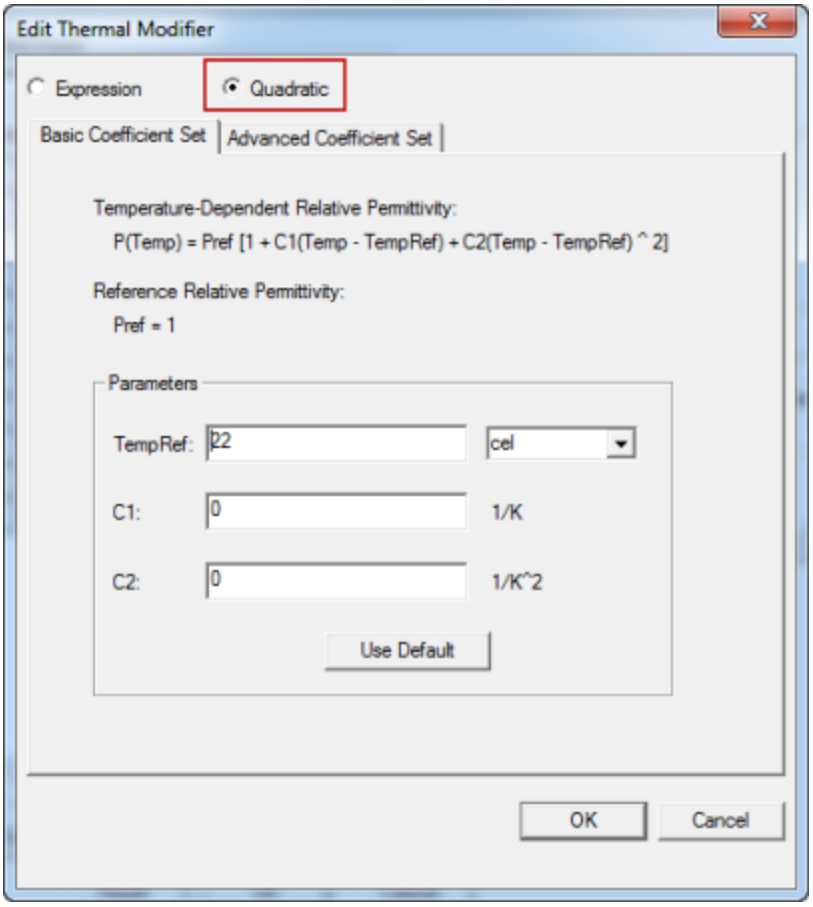
> **Note:**
>
> To edit an existing dataset, select **Edit Dataset** from the drop-down menu in the **Edit Thermal Modifier** dialog box. The **Datasets** dialog box appears. Select the dataset to modify and click **Edit**.

Modifier Expressions for Mechanical Designs:

- Temperature-dependent material properties are supported for Thermal and Structural solutions. They do not apply to Modal solutions.
- If your design will experience temperatures below 0° C, you must define the thermal modifiers for all applicable properties using manually defined dataset tables. General expressions are automatically converted to tables for the Mechanical solver, and these automatically generated tables only contain temperature datapoints in the range of 0–1000° C.
- For Structural solutions, if you define a thermal modifier for **Young's Modulus (E)**, you must also define a thermal modifier for **Poisson's Ratio (v)** and vice versa. Either *both* of these properties must be temperature-dependent or *neither* of them.
- If you define the thermal modifiers for **E** and **v** using dataset tables, as opposed to general expressions (linear, quadratic, conditional, or so on), you must use the exact same temperature values for both datasets. Otherwise, you will encounter material validation errors.

3. Alternatively, select the **Quadratic** option to display the tabs for **Basic Coefficient Set** and **Advanced Coefficient Set** (as shown below):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

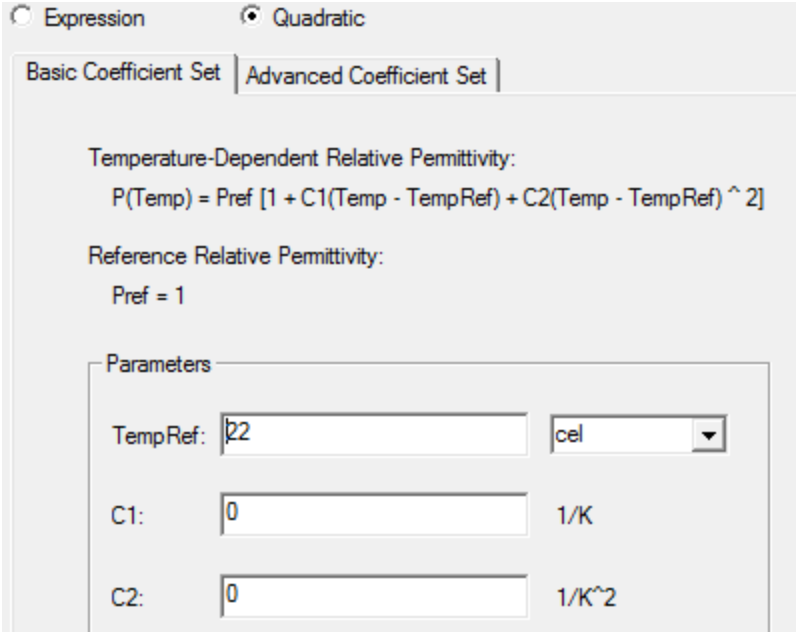


- Under the **Basic Coefficient Set** tab, you can edit fields for the TempRef and units, and fields for C1 and C2 for the following equation:

  P(Temp) = Pref[1+ C1(Temp - TempRef) + C2(Temp - TempRef)^2]

  where TempRef is 22 cel by default and where the Pref is defined as the reference relative permittivity.

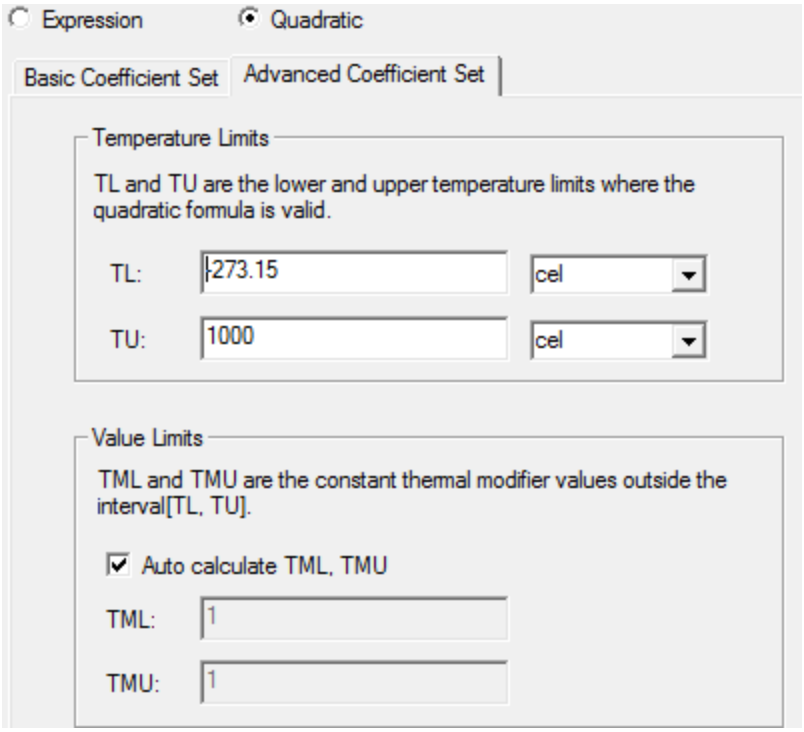
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> The coefficients, C1 and C2, should be negative to yield physical results.

- Under the **Advanced Coefficient Set** tab, you can edit fields for lower and upper temperature limits (TL and TU, respectively) and select their units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also edit the constant value limit for the thermal modifier values outside the limits. By default, these are automatically calculated. Uncheck the Auto Calculate TML and TMU to specify new values for thermal modifier lower (TML) and thermal modifier upper (TMU).

For Mechanical solutions: The restrictions regarding quadratic expressions are the same as those that apply to general expressions and datasets. See the preceding Modifier Expressions for Mechanical Designs bullet list at the end of step 2.

4. Click **OK** to accept your edits and return to the View/Edit materials dialog box.

Defining Material Properties as Expressions

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window can be assigned a mathematical expression. Simply type the expression in the appropriate **Value** box. Expressions typically contain intrinsic functions, such as sin(x), and arithmetic operators, such as +, -, *, and /, but do not include project variables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Defining Functional Material Properties

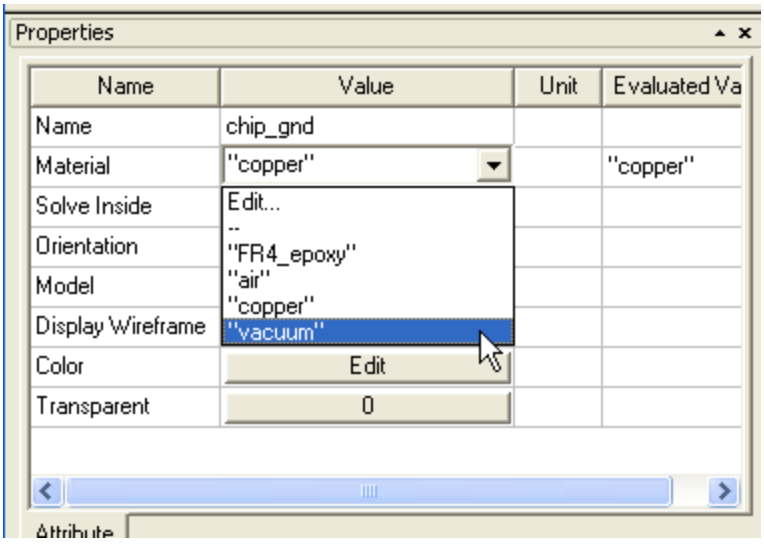
Any material property that can be specified by entering a constant can also be specified using a mathematical function. This is useful when you are defining a material property whose value is given by a mathematical relationship — for instance, one relating it to frequency or another property's value. When defining or modifying a material's properties, simply type the name of the function in the appropriate **Value** box.

Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Assigning Materials from the Object Properties Window

The Properties dialog for each object includes a materials property. If you click on the current material property you see a drop down list that includes an *Edit* command and a list of materials in the current project. You can select from the list of current materials to assign the selected material to that object.

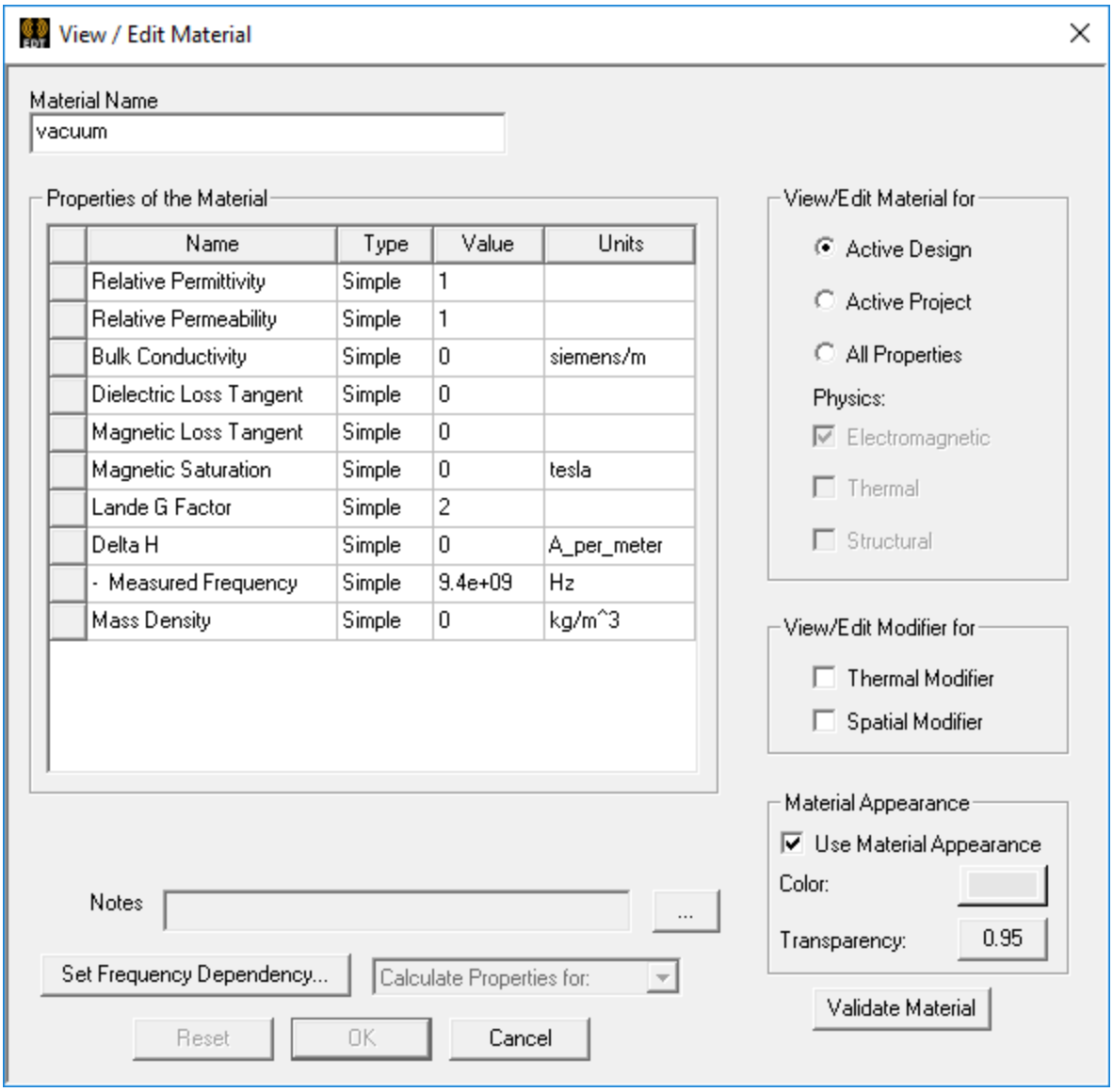


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing and Modifying Material Attributes

1. From the **Select Definition** window, select a material to view or modify, and click **View/Edit Materials**.

   The **View/Edit Material** window appears. The material name and its property values are listed.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed.
>
> To view the complete table of properties, see Filtering Materials.

2. Under **View/Edit Material for**, select:
   - **Active Design** to display properties used in the active design.
   - **Active Project** to display properties used in the active project.
   - **All Properties** to display all properties available. This enlarges the table of properties to show all properties possible. You can use the scroll bars or size the dialog to see all properties. When **All Properties** is selected, the following **Physics** classification options are enabled to show or hide properties based on simulation type:
     - **Electromagnetic**
     - **Thermal**
     - **Structural**

   > **Note:**
   >
   > If a material is edited in a design type for which the **Physics** type has not been set (e.g., an HFSS design but Electromagnetic physics type was not set), the **Physics** type will be automatically set in the material.

3. You can modify the material as follows:
   a. Provide a new name for the material in the **Material Name** text box.
   b. Under **Type**, specify whether a material property is **Simple** or **Anisotropic** as required for that property.

      For **Simple**, you provide a value or variable.

      For **Anisotropic**, you provide tensor values.

   c. Provide new material property values in the **Value** boxes.
   d. Change the units for a material property.
   e. Enable the **Use Material Appearance** check box to specify a color and transparency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Materials stored in the global material library cannot be modified.

4. If you want to add descriptive notes for the new material, click the ellipsis button [...] to the right of the **Notes** field. This opens a dialog box in which you can enter text. Pressing **Enter** or clicking **OK** saves the note. To enter multiple lines of notes, use **Ctrl+Enter** at the start of each new line.
5. Click **OK** on the **View/Edit Materials** dialog to add the new material to the material library.
6. Click **OK** to save the changes and return to the **Select Definition** window.

> **Warning:**
>
> If you modify a material that is assigned in the active project after generating a solution, the solution will be invalid.

Validating Materials

The Ansys Electronics Desktop can validate a material's property parameters for an Ansys EM software product. For example, the software will check if the range of values specified for each material property is reasonable.

If a material's property parameters are invalid, an error message will appear in the lower-right corner of the **View/Edit Material** window. If the parameters are valid, a green check mark will appear there.

To validate the material attributes listed in the **View/Edit Material** window:

- Select **Active Design**, **Active Project**, or **All Properties**, and then click **Validate Material**.

Copying Materials

1. In the **Select Definition** window, select the material you want to copy, and then click **Clone Material** or right-click the selected material and select **Clone** from the shortcut menu.
2. To modify the material's attributes, follow the directions for modifying materials.
3. Click **OK** to save the copy in the active project's material library.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Removing Materials

1. In the **Select Definition** window, select one or more materials you want to remove from the active project's material library.
2. Click the **Remove Material (s)** button or right-click the selected material and click **Remove** on the short-cut menu.

The material is deleted from the project material library.

> **Note:**
>
> The following materials cannot be deleted:
>
> - Materials stored in Ansoft's global material library.
> - Materials that have been assigned to objects in the active project.

In a project library, you may want to use the **Tools** > **Project Tools** > **Remove Unused Definitions** command to remove selected materials definitions that your project does not require.

Exporting Materials to a Library

1. In the **Select Definition** window, select the material you want to export.
2. Click the **Export Material to Library** button, or right-click the selected material and click **Export** from the short cut menu.

   The **Export to material library** file browser appears.
3. Click **PersonalLib** to export the material to a local project directory, accessible only to the user that created it.

   Click **UserLib** to export the material to a library that is shared by more than one user, usually in a central location.
4. Type the library's file name and then click **Save**.

Sorting Materials

You can change the order of the materials listed in the **Select Definition** window. You can sort the list of materials by name, library location, color, or material property value. To change the order of the listed materials:

- Click the column heading by which you want to order the materials.

If the arrow in the column heading points up, the material data will be listed in ascending order (1 to 9, A to Z) based on the values in the column you chose. If you want the material data to be

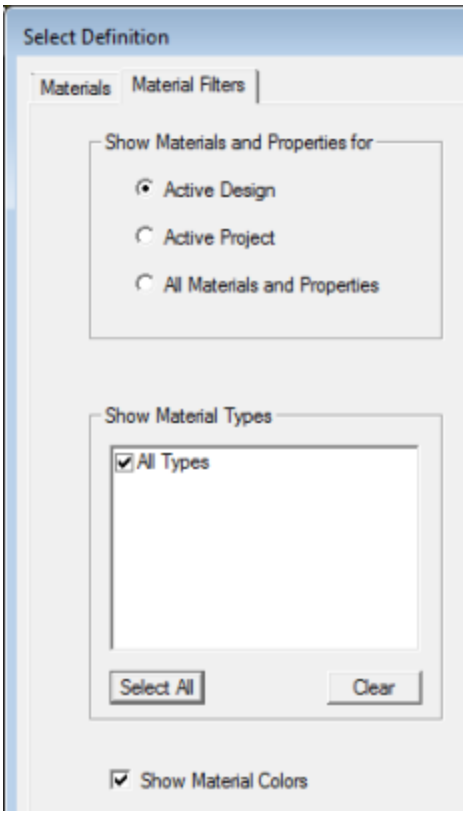
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

listed in descending order (9 to 1, Z to A), click the column heading again. The arrow will point down.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Filtering Materials

If you want to remove certain materials or material properties from the list in the **Select Definition** window, use the filter options under the **Material Filters** tab. You can filter out materials based on the product or library with which they are associated. You can also filter out material properties and types of material properties. And you can remove the filtering in order to see all available material properties.



To filter materials or material properties listed in the **Select Definition** window, using the choices in the **Materials** tab:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The text field under Libraries lists the libraries for the project. Selecting the listed library highlights it and cause the table to display the materials in that library.
2. Above the Libraries area, you can check or uncheck boxes to show or hide Project Definitions and All Libraries.

- With both unchecked, nothing appears in the materials table. With both checked, the table shows all materials and highlights those used in the project.
- With only Project Definitions checked, the materials table shows only the materials used in the project.
- With All Libraries checked, the table displays all materials, but may not show all available properties.

To filter out or show additional materials/properties in the **Materials** tab:

1. Click the **Material Filters** tab.
2. Under **Show Materials and Properties for**, select one of the radio buttons:

- **Active Design** to display materials/properties used in the active design.
- **Active Project** to display materials/properties used by the active project.
- **All Materials and Properties** to display all materials and properties available. Selecting this enlarges the table of materials shown under the **Materials** tab to show all materials possible. You can use the scroll bars or size the dialog to see all materials.

3. Under **Show Material Types**, select materials types to display on the **Materials** tab. Click **Select All** to select all of the types listed. Click **Clear** to clear all selections.
4. If you check Show Material colors, the Materials tab of the **Select Definition** window includes a color for material colors. See Viewing and Modifying Material Attributes.
5. Click the **Materials** tab to save your selections.

Click **Cancel** to revert back to the last saved selections.

Working with Material Libraries

There are three different materials libraries in Mechanical: a *system library*, a *user library*, and a *personal library*.

The library files that ship with Electronics Desktop are stored under the **`syslib`** directory. These libraries are intended to be read-only and should not be modified. They are available for any material assignment in any project.

In addition to the system libraries, Electronics Desktop recognizes two user-configurable library structures, called the *User Library* and the *Personal Library*. These are used to add user (or company)-defined materials. Customarily, **`userlib`** is a network repository for proprietary or corporate definitions available to all seats in an enterprise, while **`personalLib`** contains project and design-specific libraries as needed by individual designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A root library directory is set up at installation. If none is specified, the default is the root Electronics Desktop directory.

Materials from all libraries are available for use in projects.

Editing Libraries

There are two different methods of editing libraries:

- Right-click **Materials** in the project window to display the **Edit Libraries** shortcut menu. Clicking displays the **Edit Libraries** window.

  Editing definitions from the project window does not modify the configured libraries for any particular design, since this is editing in general.

- Using **Tools** > **Edit Libraries** > **Materials** from the menu bar takes the current design into account and adds any new libraries to the configured list for the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - Drawing a Model

After you insert a design into the current project, you can draw a model of the electromagnetic structure. The general strategy is to build the model as a collection of 3D objects. You can assign any single material to each 3D object.

You can create 3D objects by using the modeler's **Draw** commands or you can draw 1D and 2D objects, and then manipulate them to create 3D objects. Objects are drawn in the **3D Modeler** window. You can also import objects from other systems and use the 3D component libraries

To open a new **3D Modeler** window, do one of the following:

- Insert a new design into the current project.
- Double-click an Ansys Electronics Desktop design that uses the 3D Modeler (HFSS, Q3D, Maxwell (2D and 3D), or Icepak) in the project tree.

If a **3D Modeler** window for an existing design is not open, do one of the following:

- Click **[solverName]** > **3D Model Editor**.
- Right-click the design name in the project tree, and then click **3D Model Editor** on the shortcut menu.

The model you draw is saved with the current project when you click **File** > **Save**.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

When working with multiple projects, or when a project has multiple designs, you may have multiple **Modeler** windows available. To switch to the modeler window associated with a specific design:

1. In the Project Manager window, select the **Design** of interest.
2. Click **[solverName]** > **3D Model Editor** to focus the modeling window on the selected design.

If the menu command is unavailable, then the selected design is already in the modeler window.

Setting Units of Measurement for the Model

You can specify the units of measurement for drawing geometric models. After the units of measurement have been specified, they are assigned to the objects in the **3D Modeler** window.

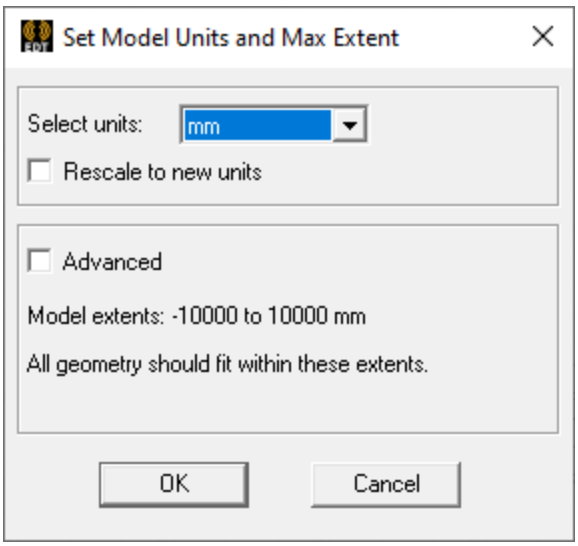
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. Click **Modeler** > **Units**.

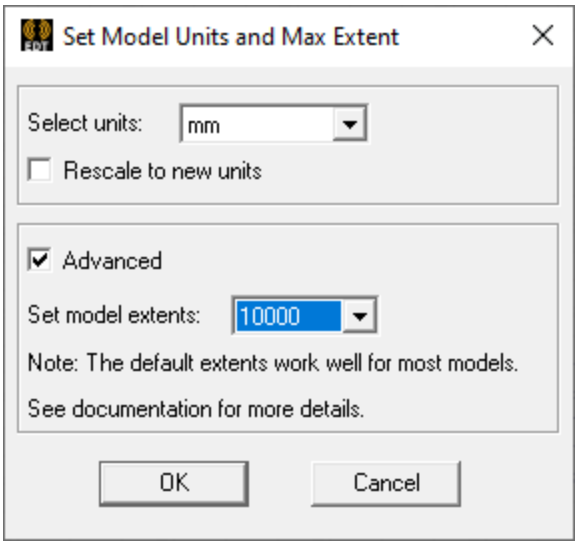
   The **Set Model Units** dialog box appears.



2. Select the new units for the model from the **Select units** drop-down menu.
3. Specify how the change in units affects the model:
   - Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
   - Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. While most models should fit the default, you can check Advanced to enable Max model extent (E) to specify model extents in the 3D modeler.



.

Parasolid kernel has a strict size limit of +/- 500 units. As many AEDT models go beyond this limit in the desired units, we have implemented internal scaling to allow creation of models with a larger extent. With our default model extent of 10,000, all geometry will be internally scaled down by a factor of 100 to fit within parasolid size limit. We recommend using the default extent of 10,000 as much as possible. Modeling operations will output an error if model goes beyond 50,000 units.

A second extent option of 100 is available in the advanced options.

If model extent is set to 100, we do not scale geometry internally. Modeling operations will output an error if model goes beyond +/- 500 units. Extent of 100 should be only used in rare situations where it is determined that scaling is causing a failure, and there is no workaround.

5. Click **OK** to apply the new units to the model.

Drawing Objects

You can draw one-, two-, or three-dimensional objects using the **Draw** commands. You can alter objects individually or together to create the geometry of your structure. In the **Tools** > **Modeler Options,** **Drawing** tab, you can set a default to either draw objects directly with the mouse or by invoking a **Properties** dialog in which you can enter the values for the object dimensions. The **Dialog** mode drawing feature works with the equation based curve, equation based surface, and all two and three dimensional objects. You can toggle to **Point** mode via the **F3** function key and to **Dialog** mode via the **F4** function key. When you use the **Dialog** mode for drawing objects the Edit property of new primitives setting is ignored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One-dimensional (1D) objects in the modeler include straight line, arc line, center-point arc, and spline segments, or a combination of these - called polylines. One-dimensional objects are open objects; their boundaries do not enclose a region, unless you connect their endpoints. They have length, but no surface or volume. Generally they are used as temporary objects from which to create 2D objects.

Two-dimensional (2D) objects in the modeler include objects such as equation based surfaces, rectangles, ellipses, circles, and regular polygons. Two-dimensional objects are closed sheet objects; their boundaries enclose a region. You can create 2D sheet objects by covering the enclosed region. In many applications (FSS, antennas) it is essential to calculate net power flow through a surface.

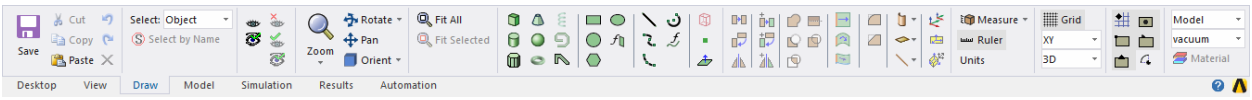
You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as line or rectangular. If you then assign it either a height or a width, the polyline becomes a sheet object.

By default, the history tree organizes sheet objects according to their boundary assignments. To change this, select the **Sheets** icon, and right-click to display the **Group Sheets by Assignment** check box. Within the calculator sheet objects are listed under surface.

Three-dimensional (3D) objects in the modeler include objects such as boxes, cylinders, regular polyhedra, cones, spheres, torii, and helices. These objects have boundaries that enclose a region with volume.

You can create 3D objects by manipulating 2D objects along a plane or by using the appropriate **Draw** commands. You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as circle rectangular. If you then assign it an appropriate diameter or both height or a width, the polyline becomes a 3D object.

You can access the **Draw** commands via icons on the **Draw** tab ribbon:



By default, the history tree organizes 3D objects by material. To change this, select the **Objects** icon, and right-click to display the shortcut menu to see **History Tree Layout commands**. You can also use Group Commands for Modeler Objects.

While you draw objects you can also:

- Select Movement Mode as 3D, In Plane, Out of Plane, Along X, Y or Z axis.
- Select Grid Plane as XY, YZ, or XZ.
- Set the Drawing Plane
- Select the Coordinate System and choose Cartesian, Cylindrical or Spherical coordinates.
- Change Snap Settings

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set Reference Point for the movement mode
- Adjust the View

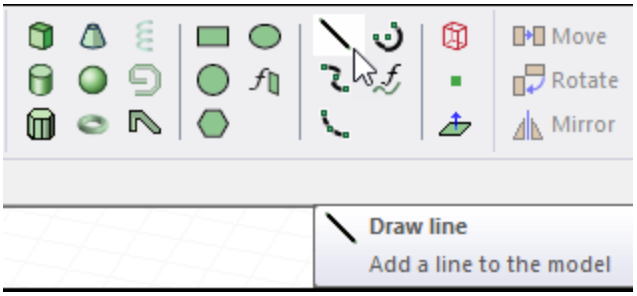
After you draw an object in the **3D Modeler** window, you can modify the object's properties, such as its position, dimensions, or color, in the **Properties** dialog box. Most model object properties can be assigned as Design variables when can then be manipulated during the solve to test their effect on the solution. For non-model objects, you can use Post Processing variables (default and Design variables,.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

Drawing a Line Segment

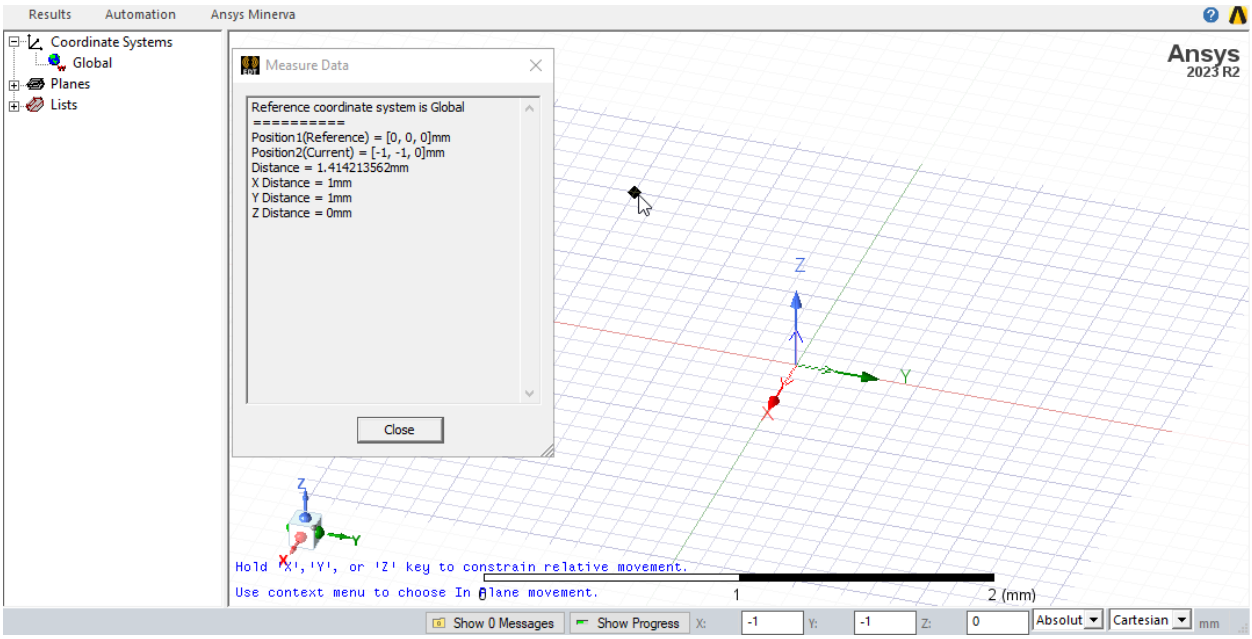
To create an object with one or more straight line segments, use the **Draw** > **Line** command.

1. From the menu bar, click **Draw**> **Line** or, on the **Draw** ribbon tab, click the **Draw line** icon:



2. Select the first point of the line in one of the following ways:
   - Click the point in the Modeler window, checking the Measure Data dialogue for the precise coordinates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

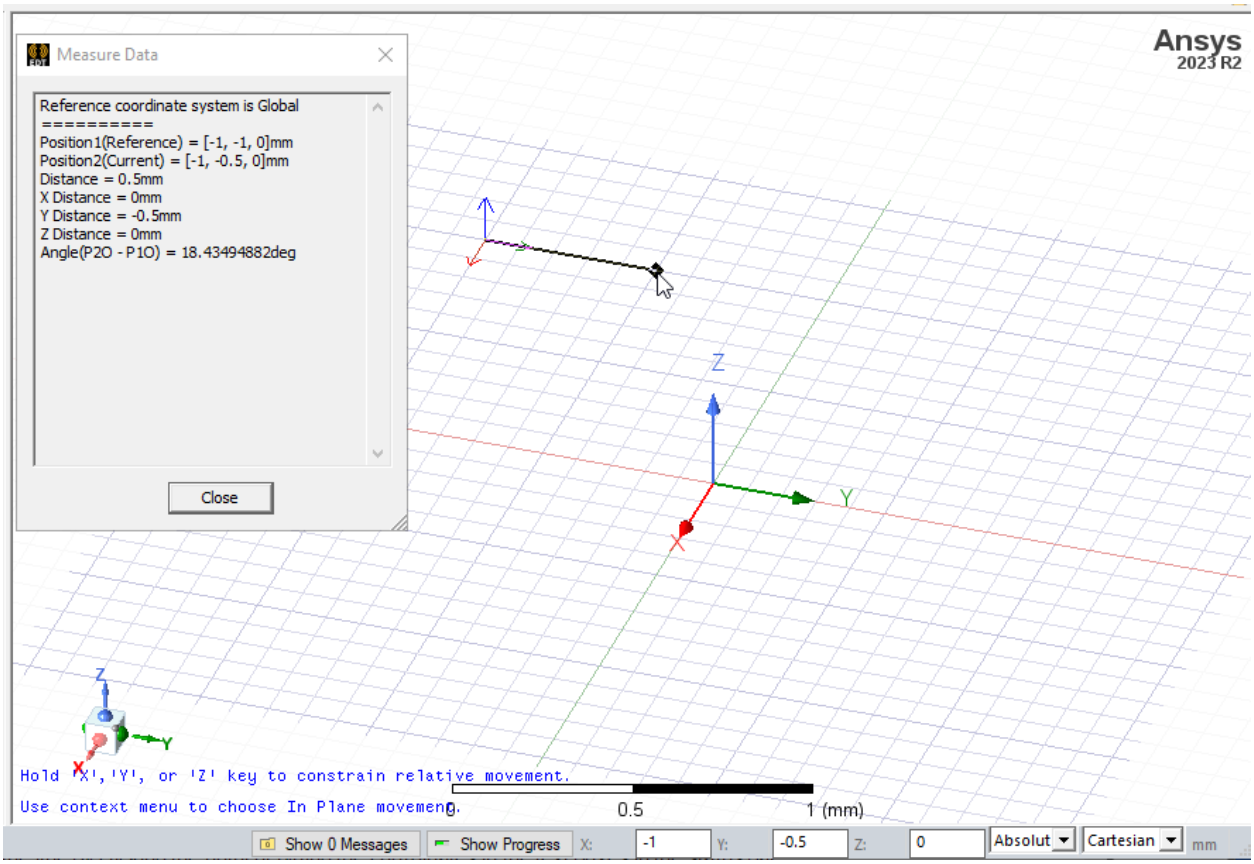


- You can accept the first point or change it by editing its coordinates in the X, Y, and Z boxes in the status bar. Note that the Status bar includes selections for Absolute or Relative Coordinates, as well as for Cartesian, Cylindrical and Spherical Coordinates.

To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

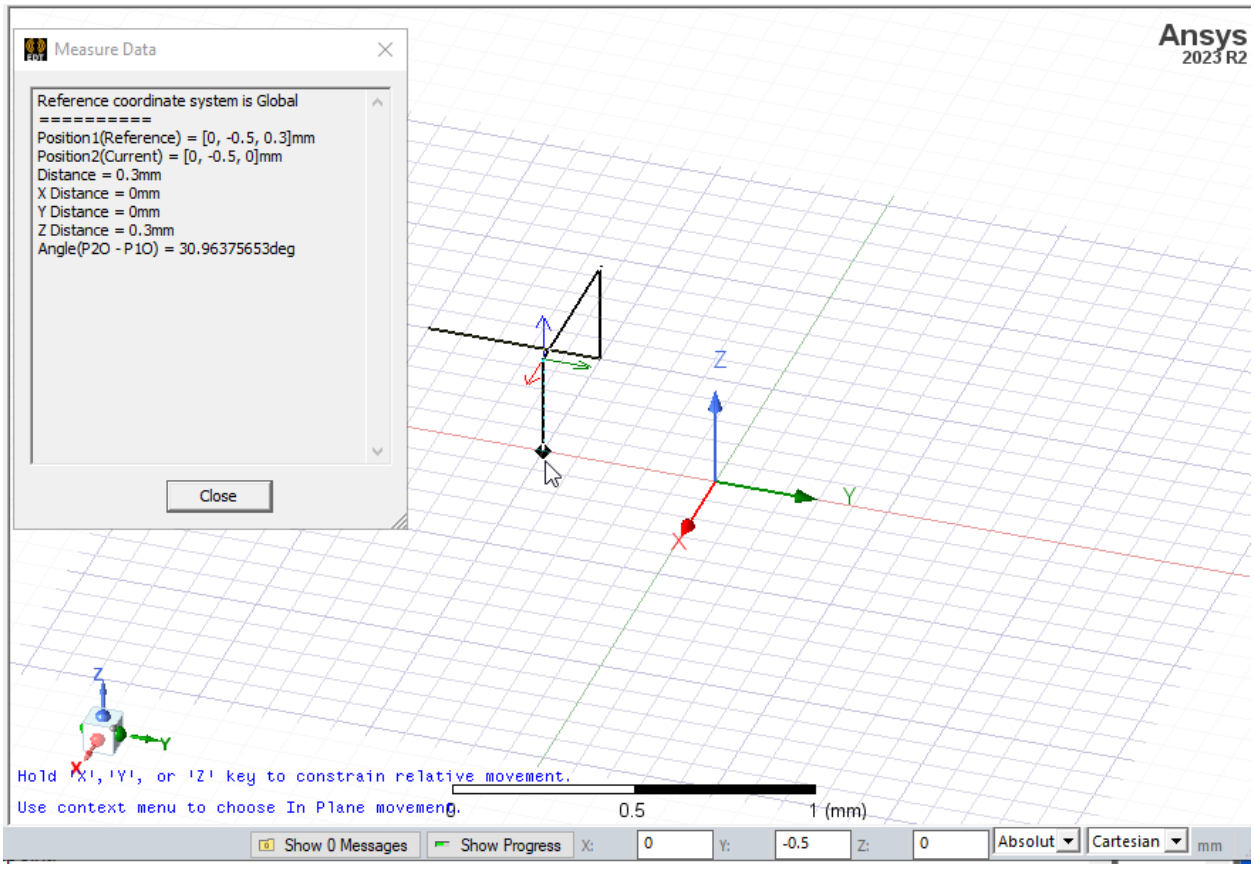
3. Select the next point of the line by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The endpoint serves as the start point for a subsequent line segment. You can specify the plane through which the line passes by selecting the X, Y, or Z keys to constrain movement. For example, the following figure shows a line with five points that includes X, Y, and Z movements.

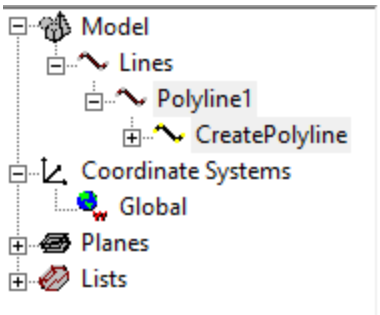
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

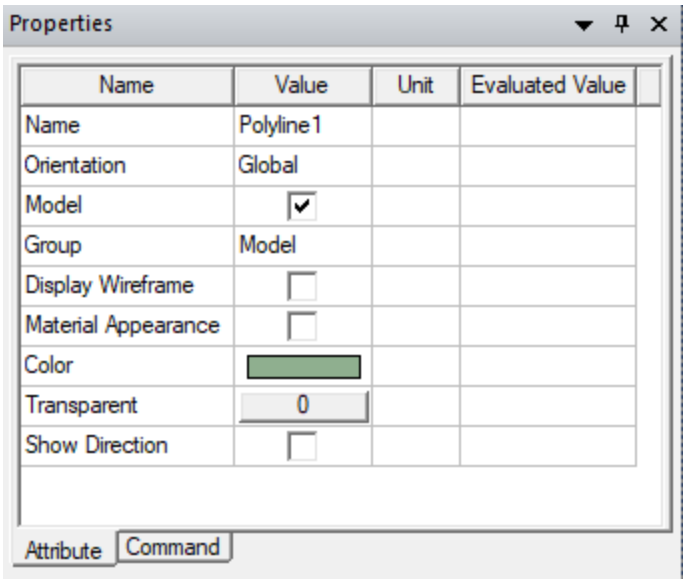
4. Complete the line in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the context (right-click) menu.
   - Press **Enter**.

   The new line appears selected in the Modeler window and shown in the History tree.

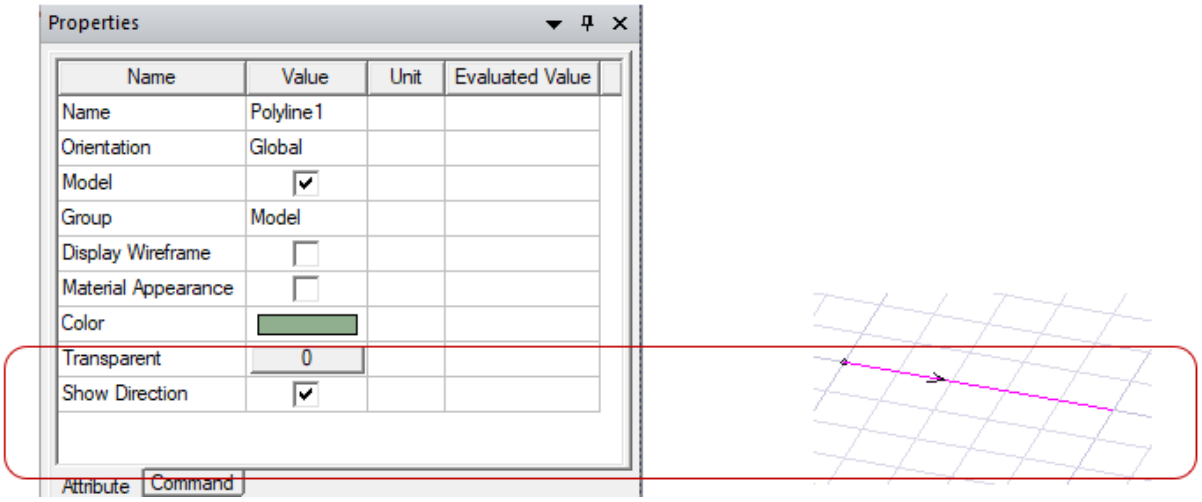


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the object's properties. You can also do this through the docked Properties window.

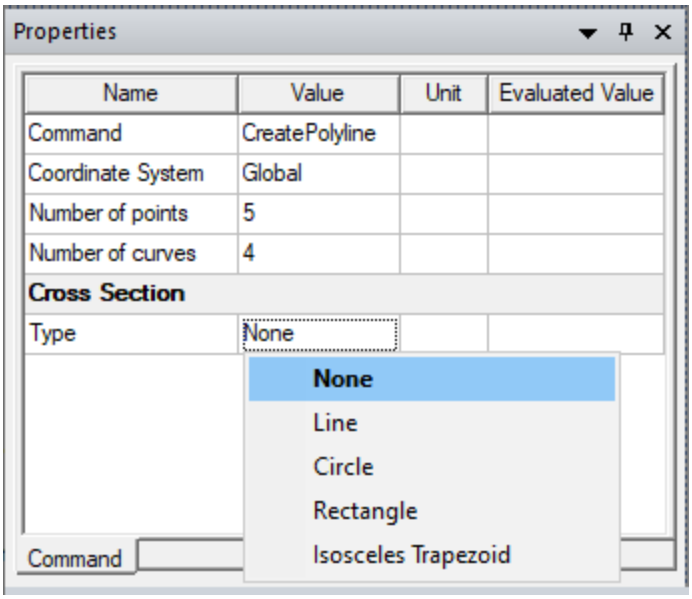


Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line (see Modal or Thermal example).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The Command tab **Properties** includes the Coordinate System that applies to the line, the number of points and curves, as well as the ability to assign a specific Cross Section to the line.



6. If you created the line with the Modeler option for editing properties of new primitives, click **OK** to close the **Properties** dialog box.

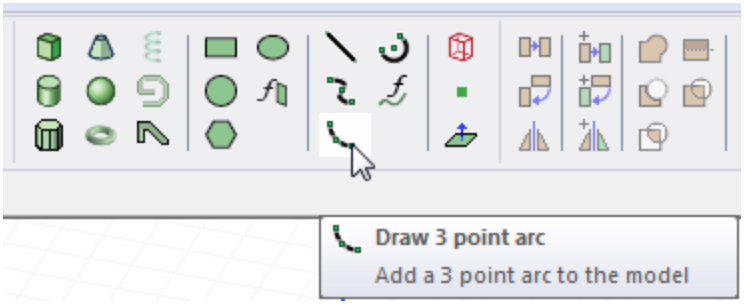
> **Note:** While drawing a polyline, you can switch between straight line, arc line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Three-Point Arc Line

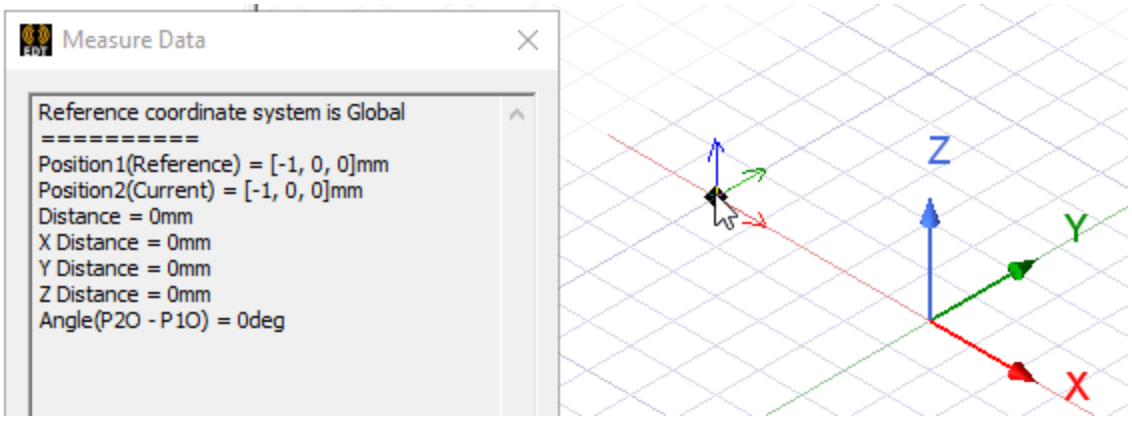
In the modeler, a three-point arc line segment is an arced line defined by three points on its curve. Use the **Draw** > **Arc** > **3 Point** command to create a polyline object with one or more arc line segments. Before you draw a three-point line arc, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Arc**> **3 Point** or, on the **Draw** ribbon tab, click the **Draw 3 point arc** icon:

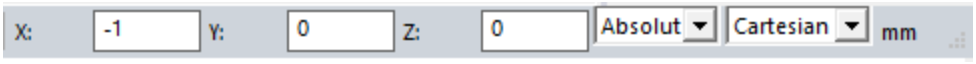
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



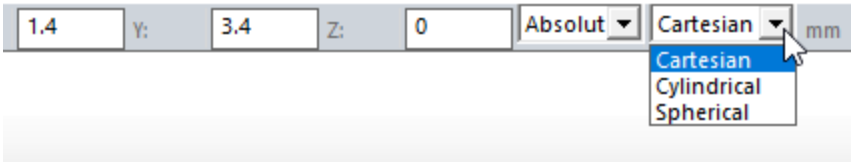
2. Select the start point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes.



     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the line in Cartesian, Cylindrical, or Spherical coordinates.



3. Select the midpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

4. Select the endpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   The endpoint serves as the start point for a subsequent arc line segment.

5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line (see Modal or Thermal example).

6. Click **OK**.

   Based on the three points you specified, the modeler calculates the center point and radius of the arc and draws an arced line through the three points.

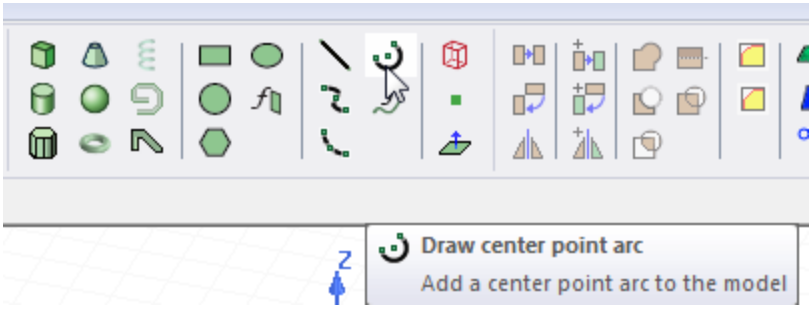
> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Center-Point Arc Line

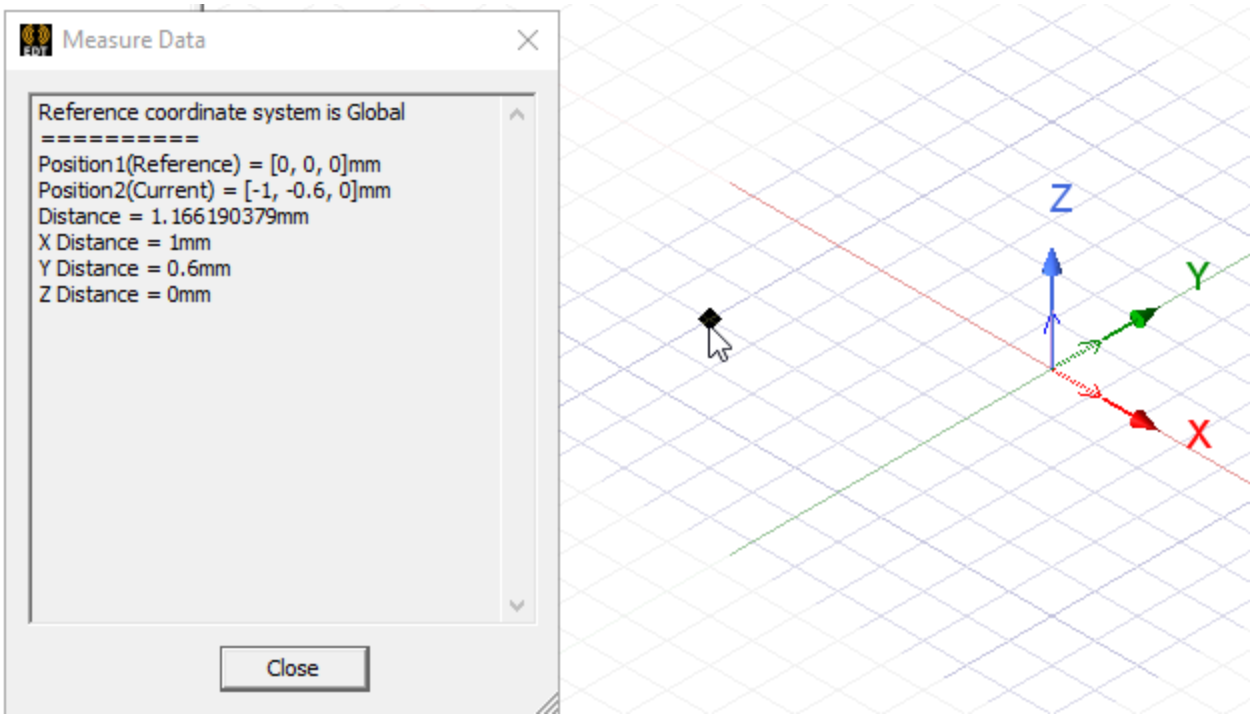
In the modeler, a center-point arc line segment is an arced line defined by a center point, start point and angle. Use the **Draw** > **Arc** > **Center Point** command to create a polyline object with one or more center-point arc line segments. Before you draw a center-pint arc line, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Arc**> **Center Point** or, on the **Draw** ribbon tab, click the **Draw center point arc** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



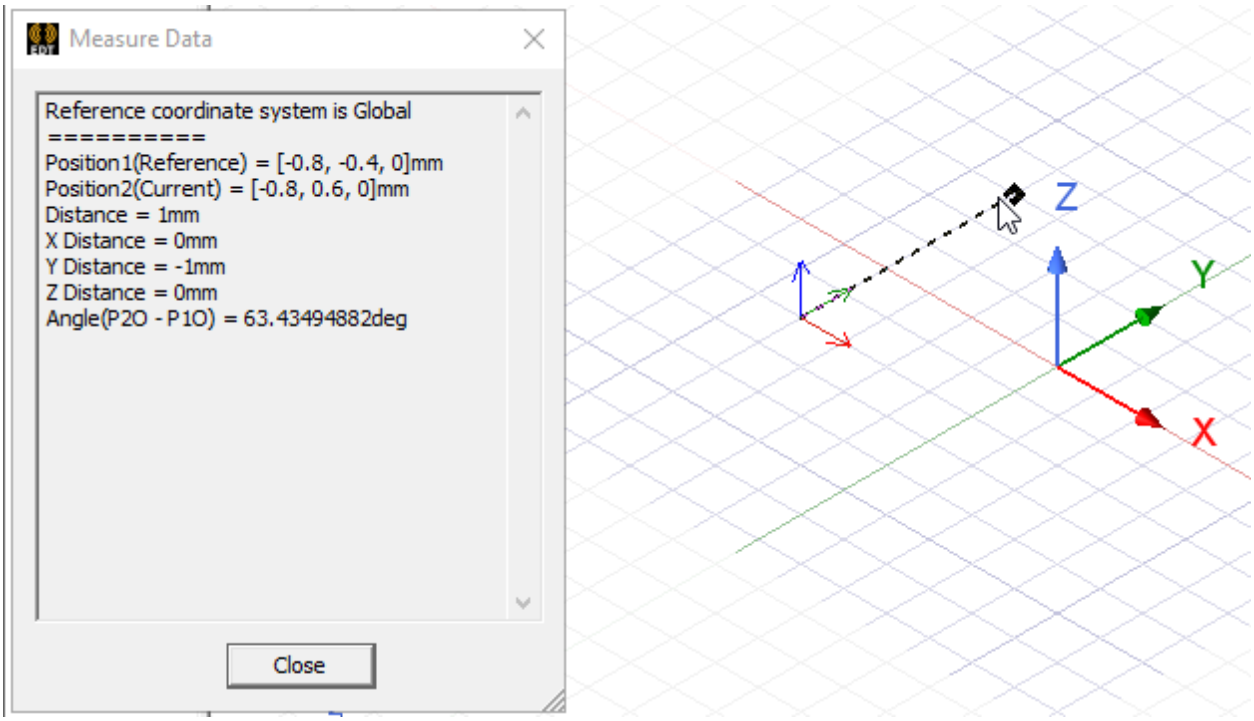
2. Select the center point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

3. Select the start point, or radius, of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

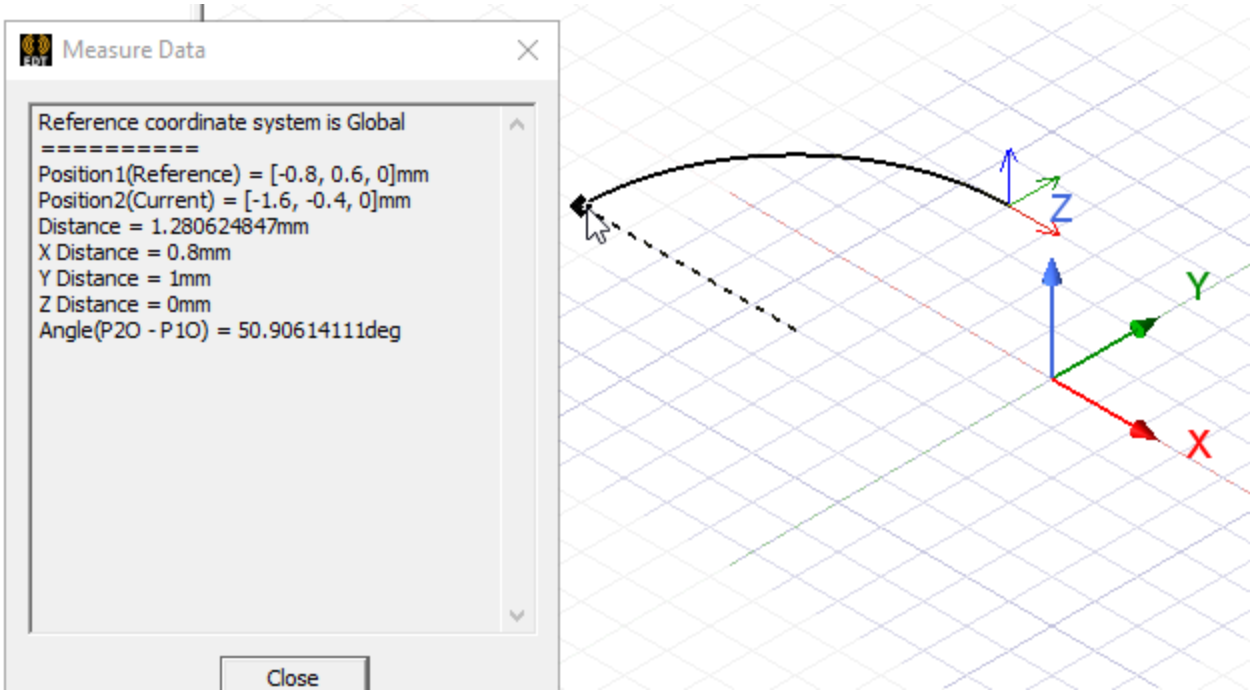


To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

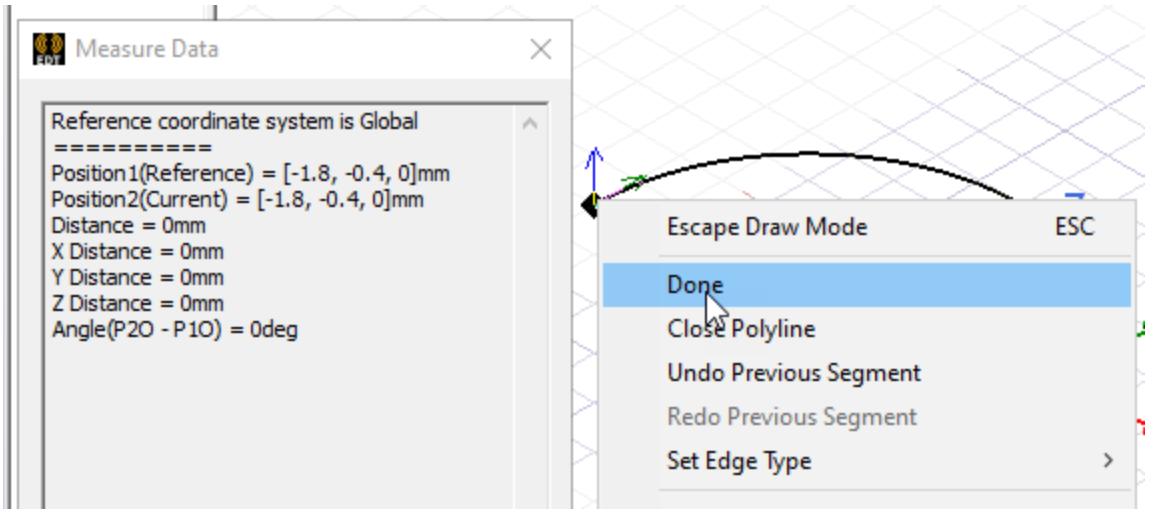
To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Sweep the angle, or endpoint, of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.



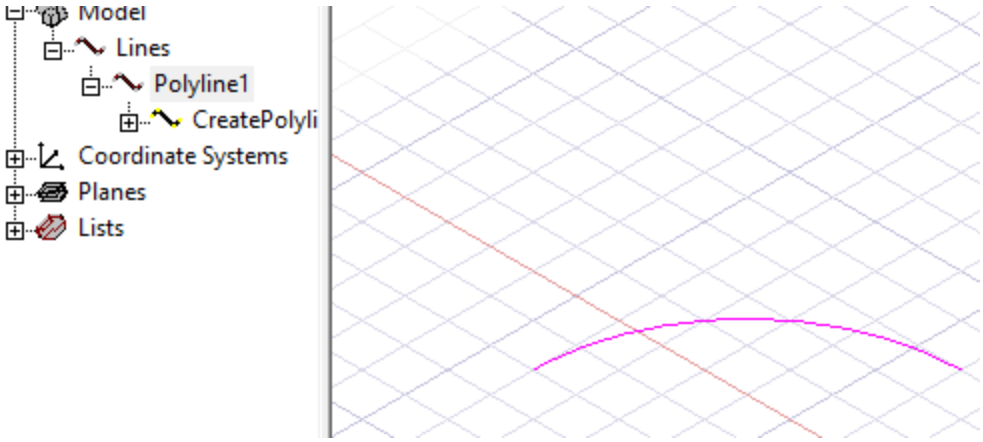
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes. The new polyline

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appears in the modeler window and in the History tree.



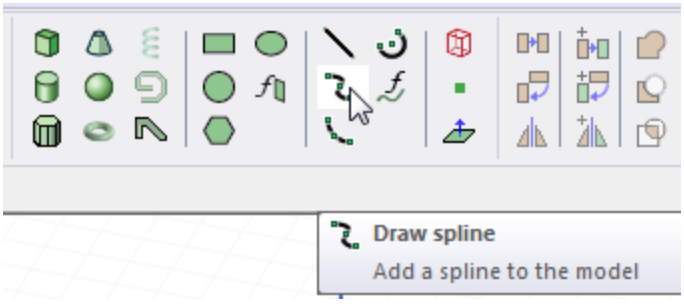
6. Click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Spline

A spline is a curved line defined by three points. The modeler uses a natural spline type: a piece wise cubic spline with an end condition that has a derivative of zero. Use the **Draw** > **Spline** command to create a polyline object with one or more spline segments. Before you draw a spline, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

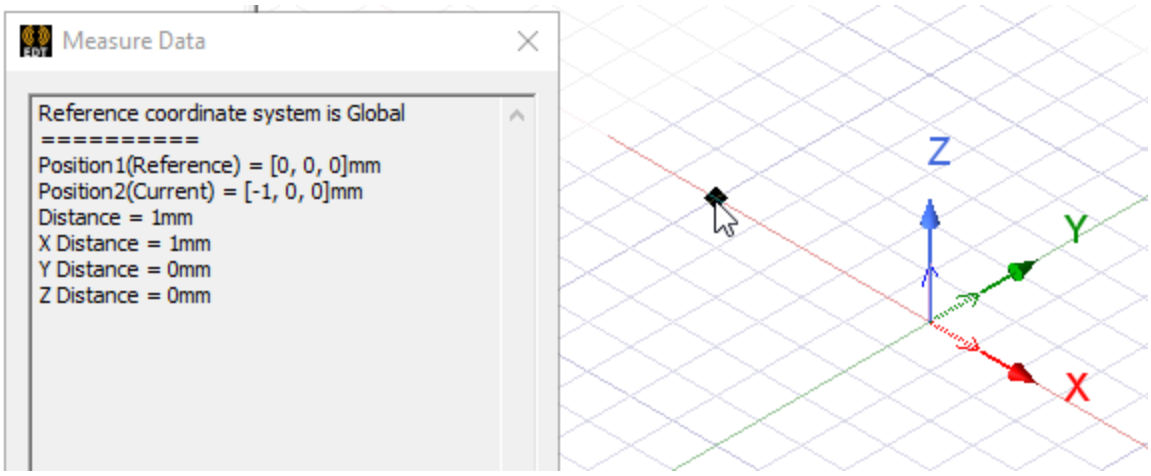
1. From the menu bar, click **Draw**> **Spline** or, on the **Draw** ribbon tab, click the **Draw spline** icon:



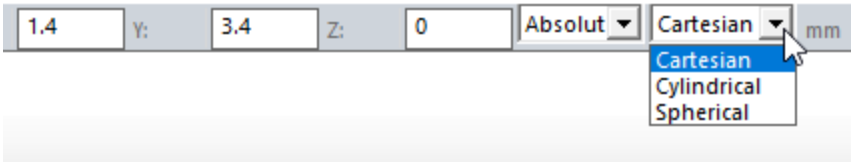
2. Select the spline's start point in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



- Type the point's coordinates in the text boxes in the Status bar, and then press **Enter**.
  The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the spline in Cartesian, Cylindrical, or Spherical coordinates.

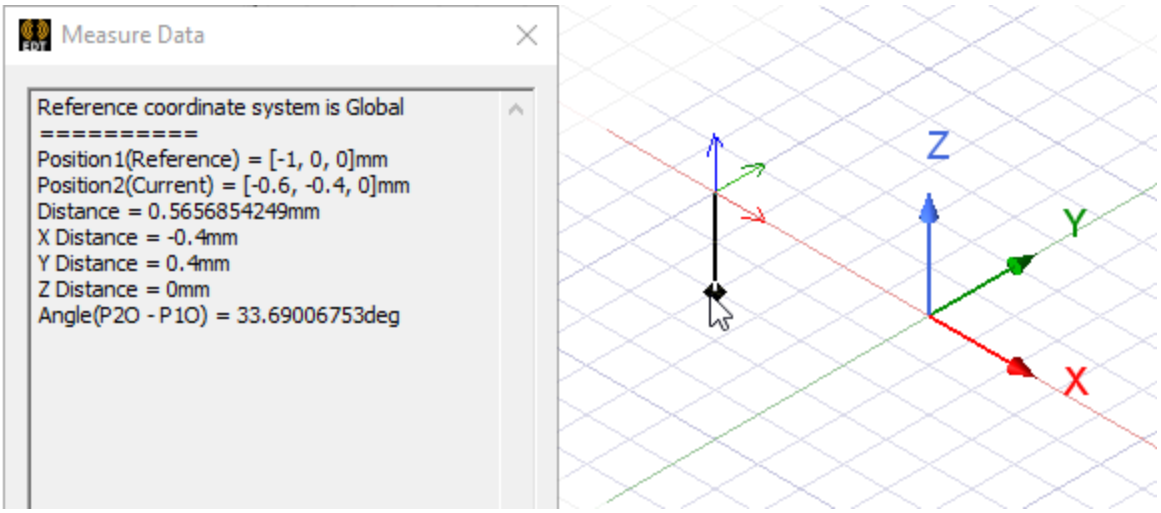


To delete the last point entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

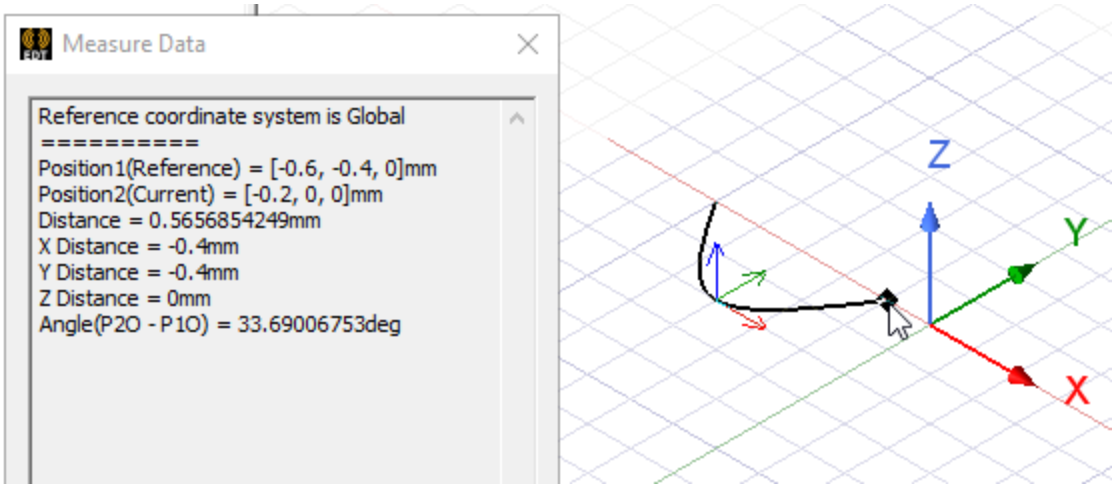
To delete all selected points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the midpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.



4. Select the endpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.

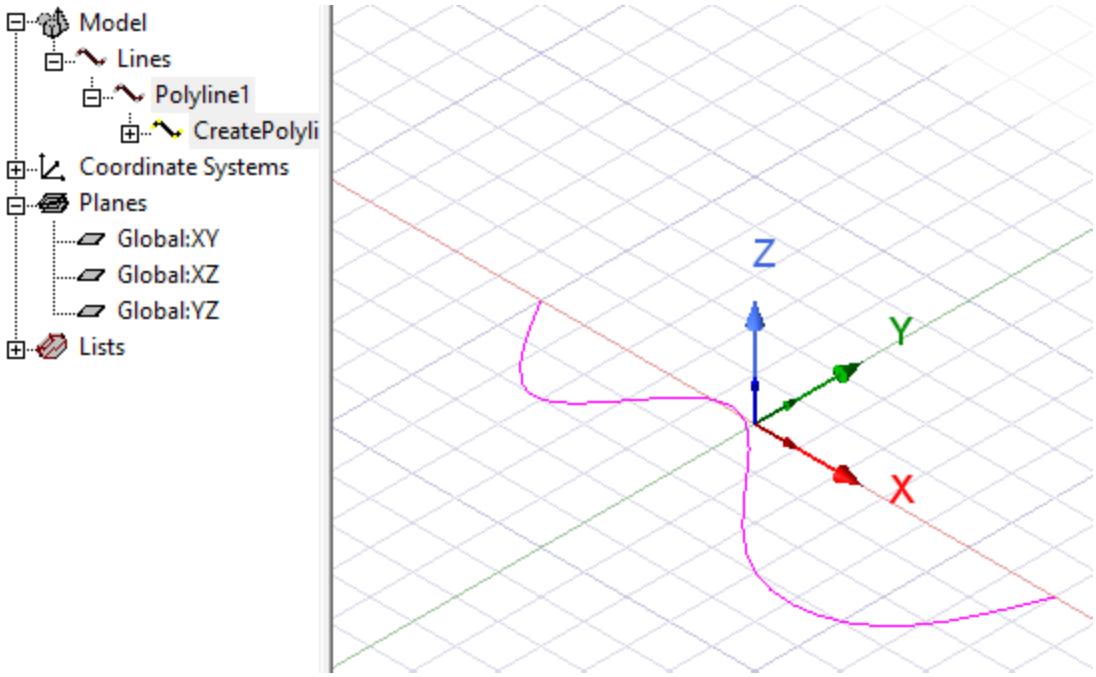


The endpoint serves as the start point for a subsequent spline segment. You can complete the spline at this point, or add additional segments.

5. Complete the spline in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press **Enter**.
  The spline appears in the Modeler window and the History tree.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to **Show Direction** as arrows. The **Show Direction** property is most helpful to unambiguously show the line start orientation when plotting fields along a line (see Modal or Thermal example).

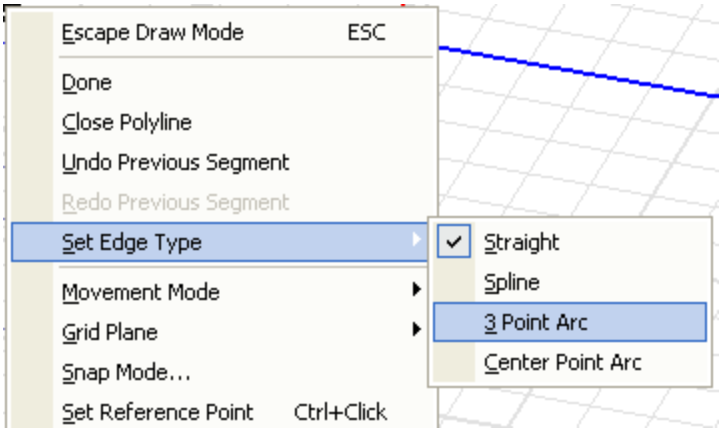
6. If you have edited Properties, click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between spline, straight line, or arc line segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Polyline

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. The endpoint of one segment is the start point for the next segment. Use the shortcut menu's **Set Edge Type** commands to switch between straight line, arc line, or spline segments while drawing a polyline.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

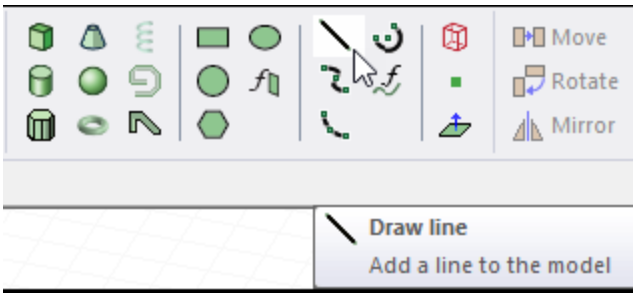


Before drawing a polyline, determine how closed polylines will be handled in the 3D Modeler options, as follows:

Using the menu bar, click **Tools** > **Options** > **General Options**. In the *Options* dialog box that appears, choose **3D Modeler** > **Operation** from the tree on the left side. Then, in the **Polyline Creation** section of the operation options, select or clear the **Automatically cover closed polylines** check box.

If checked, closed polylines become sheet objects and are listed under *Sheets* in the History Tree. If unchecked, closed polylines are listed under *Lines* in the History Tree.

1. From the menu bar, click **Draw**> **Line** or, on the **Draw** ribbon tab, click the **Draw line** icon:



2. Right-click in the **3D Modeler** window to access the shortcut menu, and then point to **Set Edge Type**.
3. Click **Straight, Spline, 3 Point Arc,** or **Center Point Arc**, depending on which type of polyline segment you want to draw.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.

5. Repeat steps 2 and 3 for each segment of the polyline object. The endpoint of the previous segment serves as the start point for the next segment.

   The shortcut menu lets you do the following for each segment:

   Undo Previous Segment or **Redo Previous Segment**.

6. Complete the polyline in one of the following ways:
   - Double-click the endpoint of the final segment.
   - Click **Done** on the shortcut menu.

   > **Note:**
   >
   > To connect the polyline's start and endpoints, click **Close Polyline** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation plotting fields along a line (see Modal or Thermal example). Notice that by going to the History Tree and selecting Create Polyline for that object, you can assign a cross section and dimensions to a polyline.

7. Click **OK**.

   If you select a polyline in the History Tree, you can use the Measure mode to see the total length.

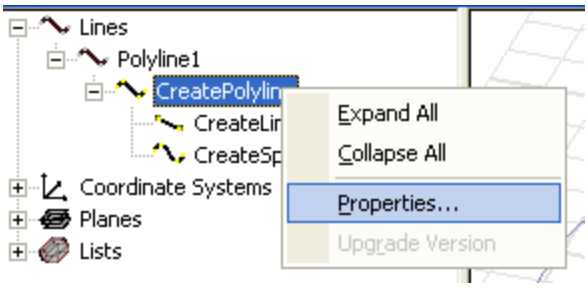
Assigning a Cross Section and Dimensions to a Polyline

By viewing the properties of a polyline from the History Tree, you can assign either a Line, Circle, Rectangle or Isosceles Trapezoid cross section to a polyline. This assignment enables editable dimension properties of width for a line, diameter for a circle, and height and width for a rectangle or trapezoid. To assign a cross section to a polyline:

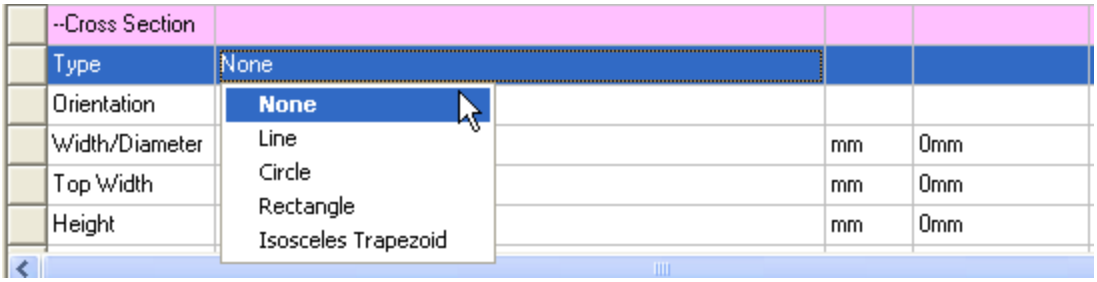
1. In the History Tree of the Modeler window, select the **CreatePolyline** entry for the polyline that you want to give a cross section.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This action displays the polyline properties in the docked **Properties** window (if you have it displayed). Alternatively, right-click the **CreatePolyline** entry and choose **Properties...** from the shortcut menu to display the **Properties** dialog box.



2. In the **Cross Section** section of the properties, in the second column of the **Type** row, click **None** to display the cross section choices.



3. Choose a **Line, Circle, Rectangle,** or **Isosceles Trapezoid** cross section.
   - Selecting **Line** causes the Cross Section area of the polyline properties to display editable fields for **Orientation** and **Width**.
   - Selecting **Circle** causes the Cross Section area of the polyline properties to display an editable field for **Diameter**.
   - Selecting **Rectangle** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width**, and **Height**.
   - Selecting **Isosceles Trapezoid** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width/Diameter**, **Top Width**, and **Height**.
4. If you select **Line**, **Rectangle,** or **Isosceles Trapezoid** you can edit the **Orientation** as **Auto, X, Y,** or **Z**. This setting specifies the direction in which the **Height** dimension extends.
5. Specify the dimensions and select the units for the Cross section.

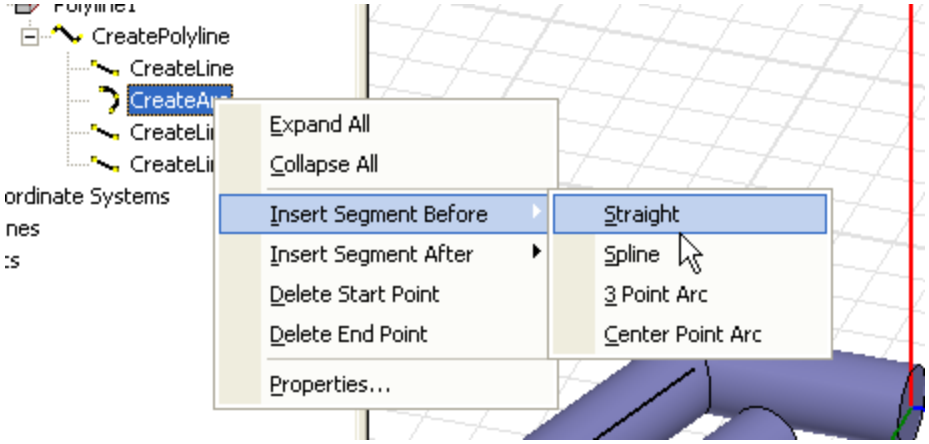
   Type a value in the dimension field(s) and select units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dimensions must be reasonable relative to the specified shape and orientation of the polyline. If the polyline cannot be extended into current Orientation for the given dimension(s), you will receive a warning. If you receive a warning, check the Orientation, dimension and units.

When the modeler can extend the dimensions legally, it displays the modified object, and lists it in the History Tree as either a Sheet object (Line or one dimensional Rectangle cross section) or as a Solid object (Circle or two dimensional Rectangle cross section).

6. You can modify the new polyline either by editing the properties, or by using the History Tree to select one of the segments that make up the polyline and right-clicking to display the shortcut menu showing commands to **Insert**, **Delete**, or display the segment **Properties**.



A segment that you select in the History Tree is indicated in the Modeler window by black highlighting. If you insert a new segment, it adopts the dimensions you specified for the polyline object.

Inserting Line Segments

You can insert line segments of various kinds for existing line, arc, or spline objects.

1. Select a line segment in the History Tree (not the modeler window). The individual segments entries are **CreateLine**, **CreateArc**, **CreateAngularArc**, or **CreateSpline**.

   This action highlights the object and enables the **Insert Line Segment** subcommands in the **Draw** menu.

2. Using the menu bar, go to the **Draw** > **Line Segment** submenu and then choose either the **Insert Segment Before** or **Insert Segment After** submenu.

   Alternatively, right-click a polyline segment in the History Tree and, from the shortcut menu, choose either the **Insert Segment Before** or **Insert Segment After** submenu

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use this submenu to specify the kind of segment to add. The inserted segment can be: **Straight, Spline. 3 Point Arc,** or **Center Point Arc**.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.
   - If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.

   Note:

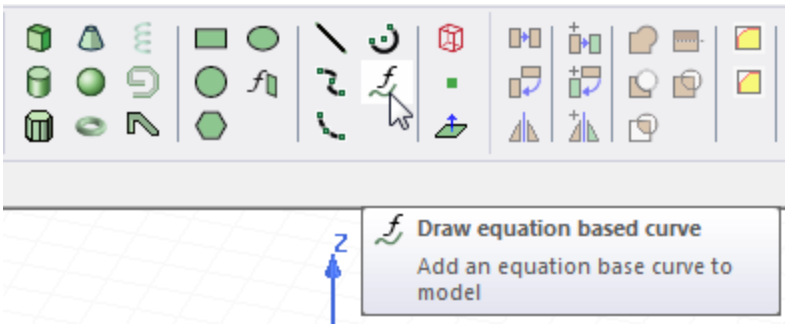
   When inserting straight segments or arcs, the insertion operation will terminate as soon as you complete the definition of a single new segment. For splines, you can insert a spline with any number of defining points, and the insertion operation terminates as soon as you terminate the spline.

5. Repeat steps 1 through 4 for each segment of the polyline before or after which you want to insert a new segment. The endpoint of the previous segment serves as the start point for the next segment.

Drawing an Equation-Based Curve

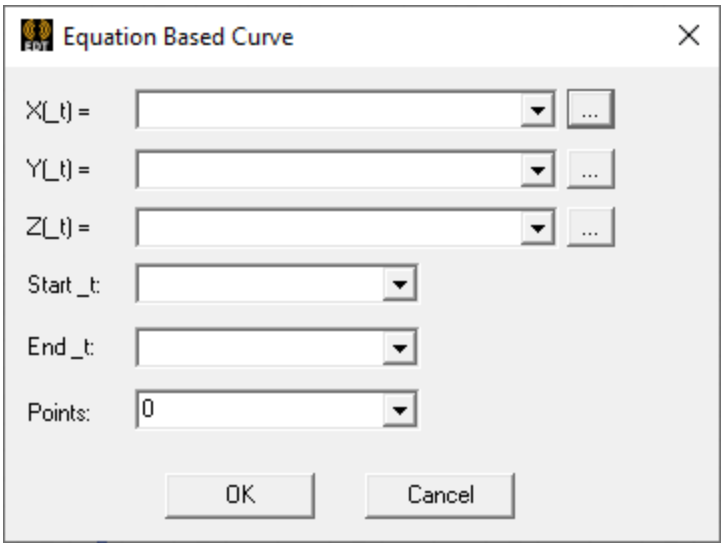
Any line that can be described by an equation in three dimensions can be drawn. Before you draw am equation-based curve, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Equation Based Curve** or, on the **Draw** ribbon tab, click the **Draw equation based curve** icon:



   The *Equation Based Curve* dialog box opens:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

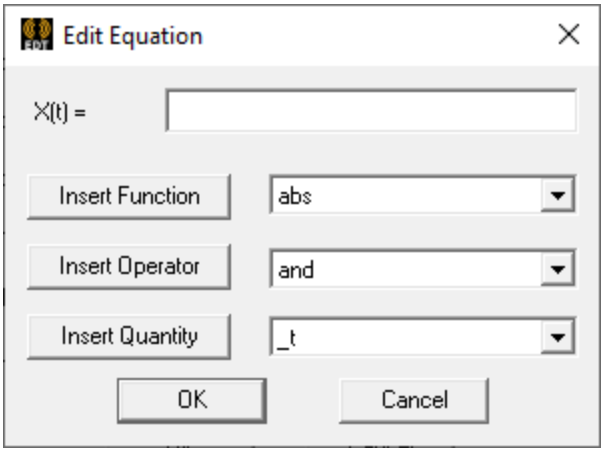


2. Type equations for **X(_t)**, **Y(_t)**, and **Z(_t)**

   You can also define an equation by doing the following:

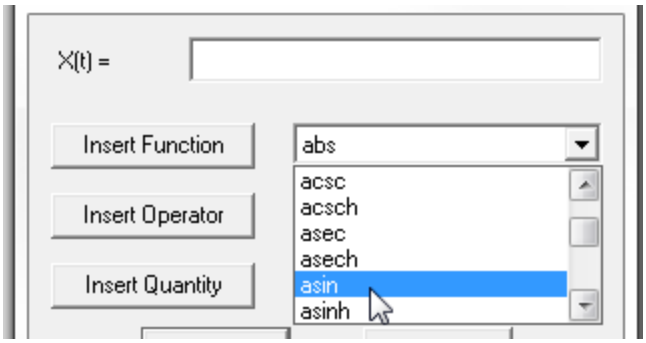
   a. Click an ellipsis button **[...]**.

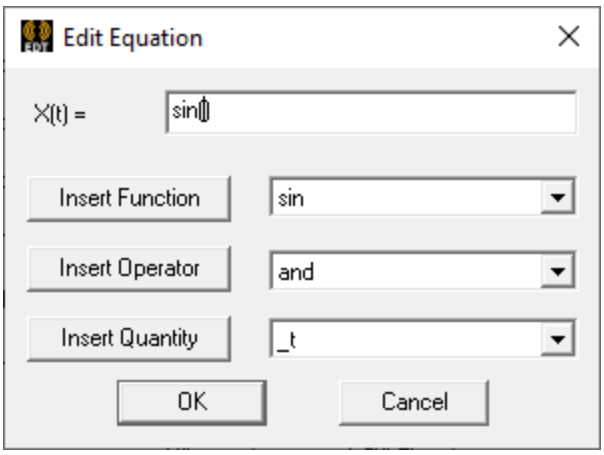
      The **Edit Equation** dialog box appears:



   b. Do one or more of the following to define the equation:
      - Type a numerical value, variable, or expression directly in the text box.
      - Build an equation for X(_t), Y(_t), or Z(_t) by specifying appropriate function, operator, and quantity to insert into the expression from the three drop-down menus:

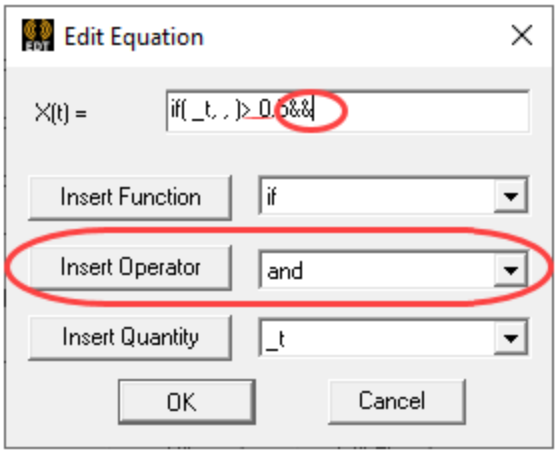
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

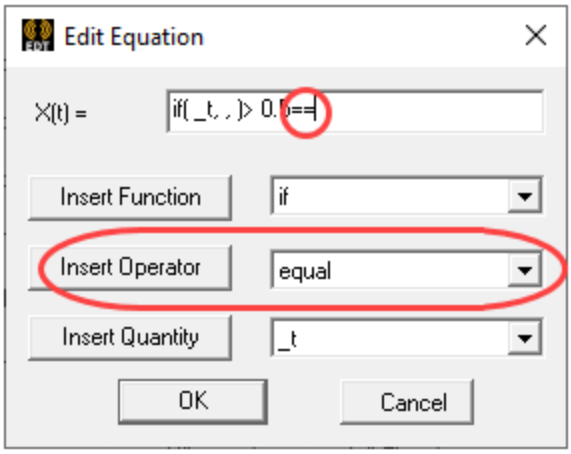




Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

- The **Insert Operator** button inserts corresponding operator:





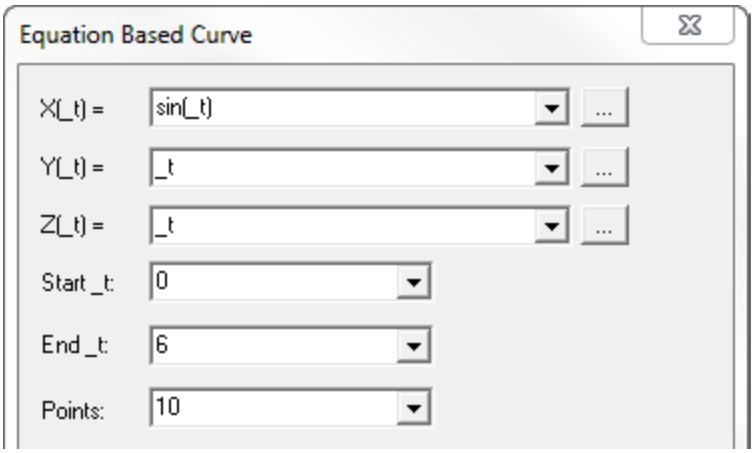
- Use a combination of the preceding two methods to build complex equations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Curve** dialog box.

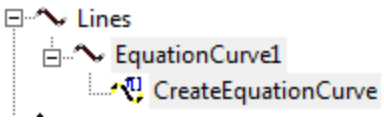
> **Note:**
>
> - The equations are taken to be meters. Use of units in equations is not properly supported.
> - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
> - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
> - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Select a start value from the **Start_t** drop-down menu.
4. Select an end value from the **End_t** drop-down menu.
5. Type in or select the number of points in the curve from the **Points** drop-down menu.



6. Click **OK** on dialog to close it and create the curve.

   The curve appears in the modeler window, and the History tree shows the curve object and the command.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes byediting the properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line (see Modal or Thermal example).

Functions, Operators and Quantities for the Edit Equation Dialog Box

This table lists the functions, operators and Quantities for the Edit Equation dialog box.

Item	Definition
Functions	
abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
acot	Inverse cotangent
acoth	Inverse hyperbolic cotangent
acsc	Inverse cosecant
acsch	Inverse hyperbolic cosecant
asec	Inverse secant
asech	Inverse hyperbolic secant
asin	Arc sine (the inverse function of sine)
asinh	Inverse hyperbolic sine
atan	Arc tangent (the inverse function of a tan)
atanh	Inverse hyperbolic tan
cos	Cosine
cosh	Hyperbolic cosine
cot	Cotangent
coth	Hyperbolic cotangent
csc	Cosecant
csch	Hyperbolic cosecant
ln	Natural logarithm.
log	Natural logarithm (same as ln).
sin	Sine.
sinh	Hyperbolic sine.
sqrt	Square root of the selected simulation quantity.
tan	Tangent.

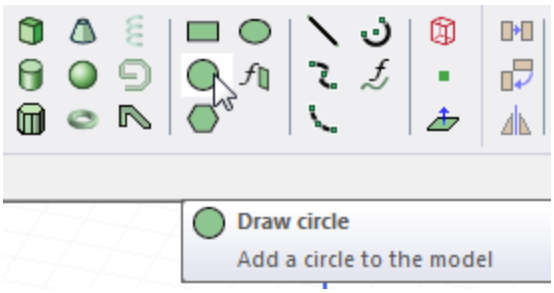
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Definition
tanh	Hyperbolic tangent.
Operators	
and	
cross	
division	
dot	
equal	
exp	
exponent	
greater_than	
greater_than_ or_equal	
if	if(, ,)
not	
not_equal	
or	
Quantities	
_t	For equation based lines
_u,_v	For equation based surfaces.
PI	

Drawing a Circle

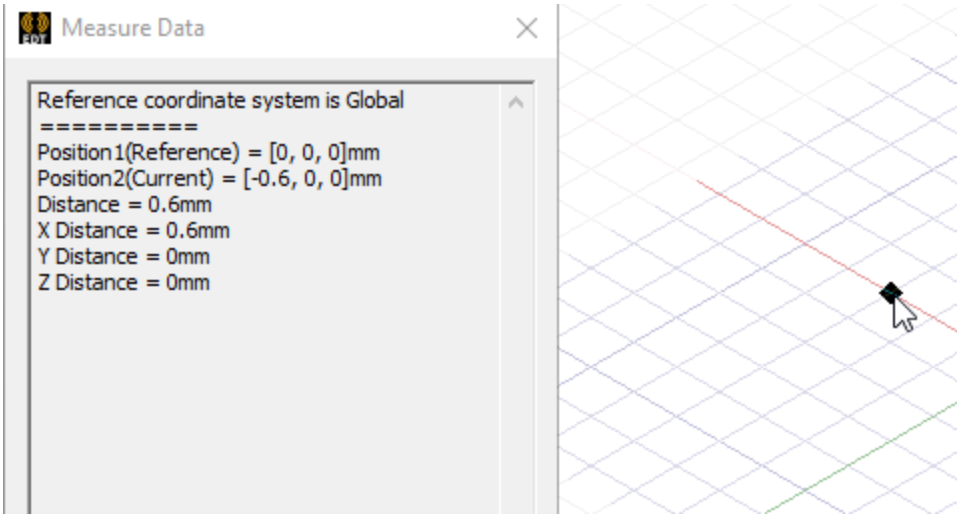
Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces in the modeler. Before you draw a circle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Circle** or on the **Draw** ribbon tab, click the **Draw circle** icon:



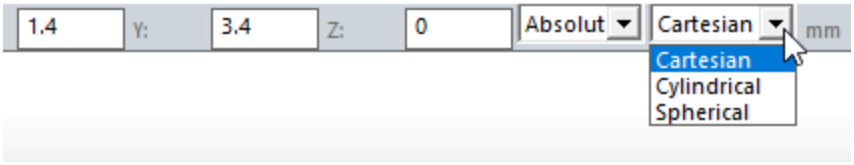
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the circle in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

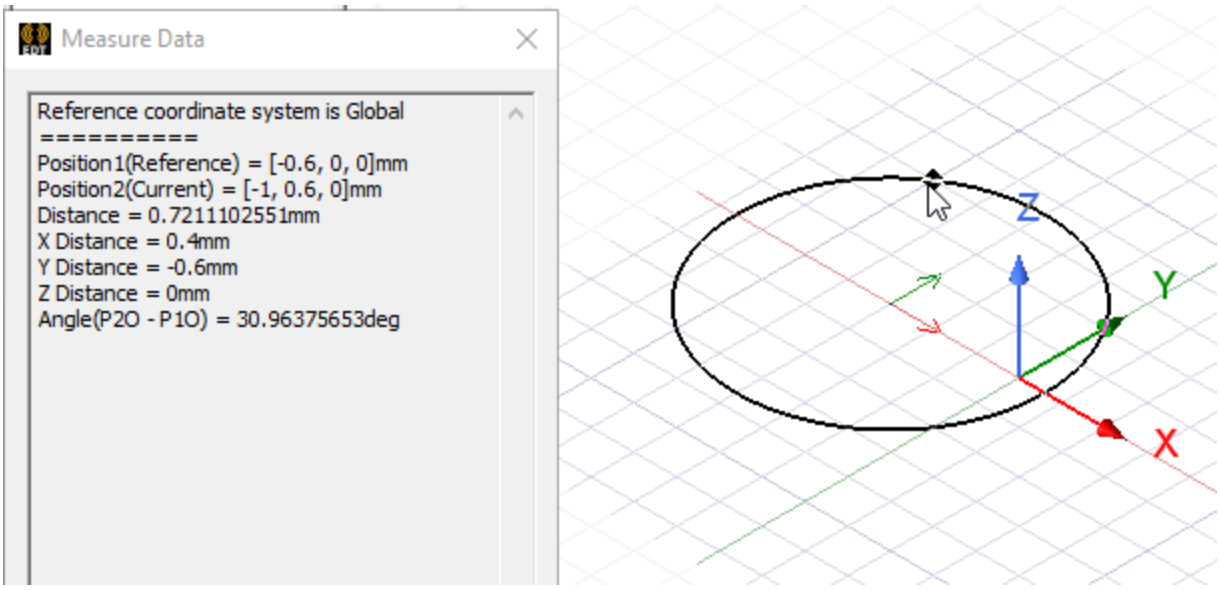
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the circle in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the radius by selecting a point on the circle's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

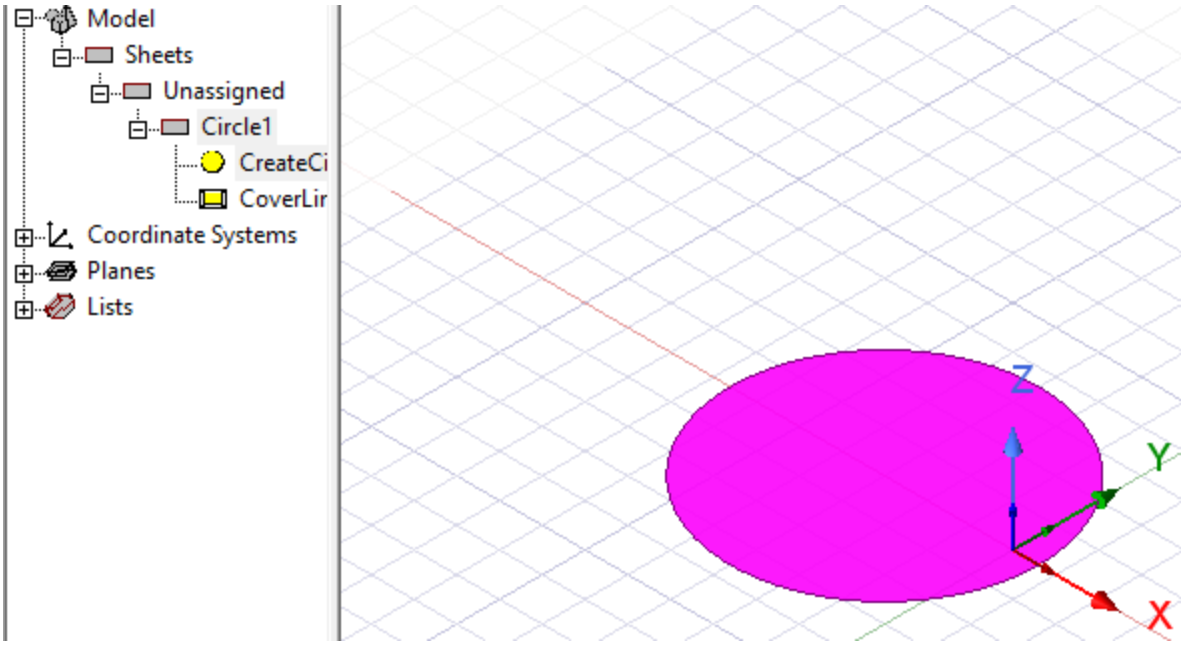
- Click the point.



- Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Double-click or right-click and select Done from the shortcut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the circle will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object. The new circle appears in the Modeler window and History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Modifying Surface Approximation Settings and related sections:

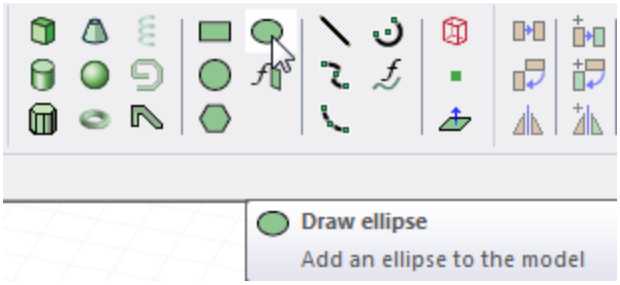
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing an Ellipse

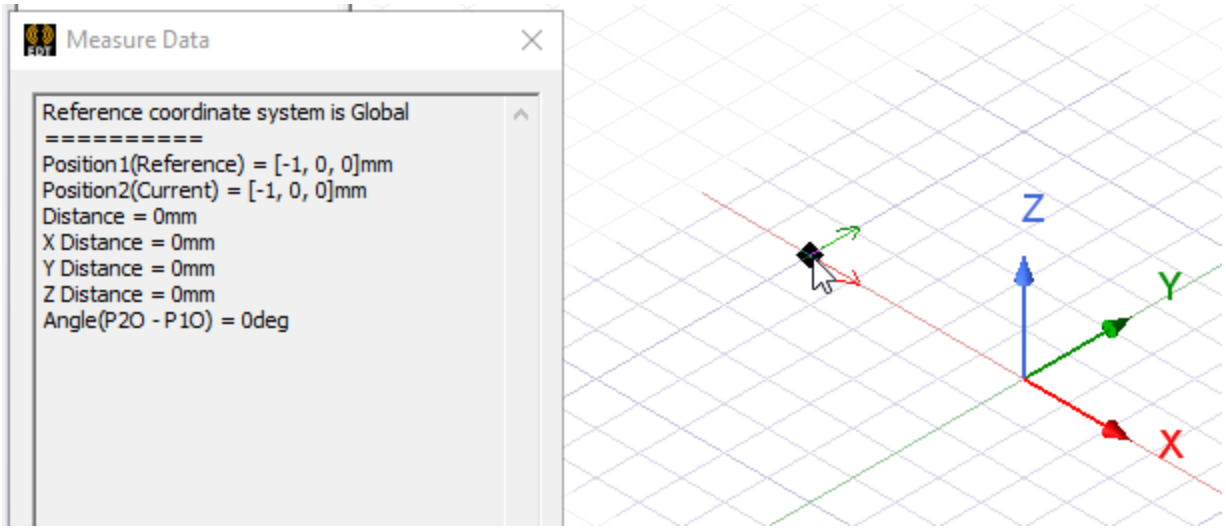
Draw an ellipse by specifying a center point, base radius, and secondary radius. Before you draw an ellipse, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw>** **Ellipse** or, on the **Draw** ribbon tab, click the **Draw ellipse** icon:

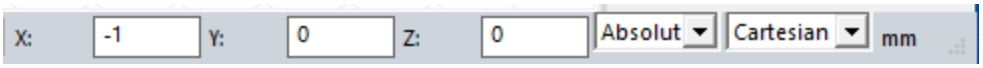
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



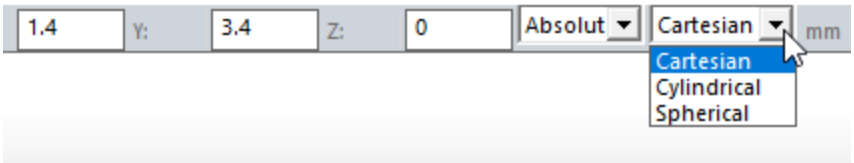
2. Select the center point of the ellipse in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.



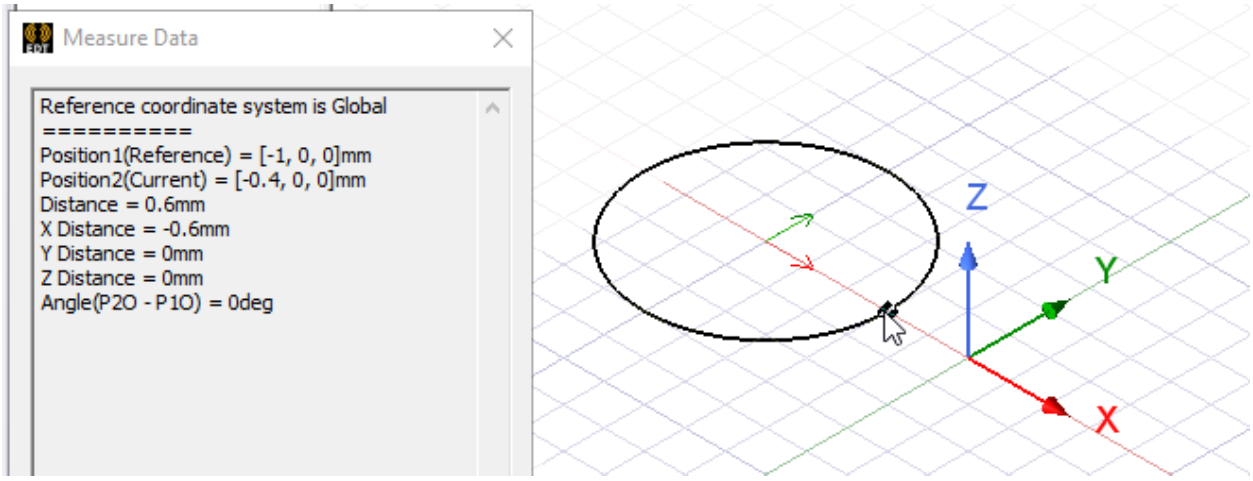
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the ellipse in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the base radius of the ellipse. If the current drawing plane is xy, then x is the base radius direction. If the drawing plane is yz, then y is the base radius direction. If the drawing plane is xz, then z is the base radius direction. Select the point in one of the following ways:

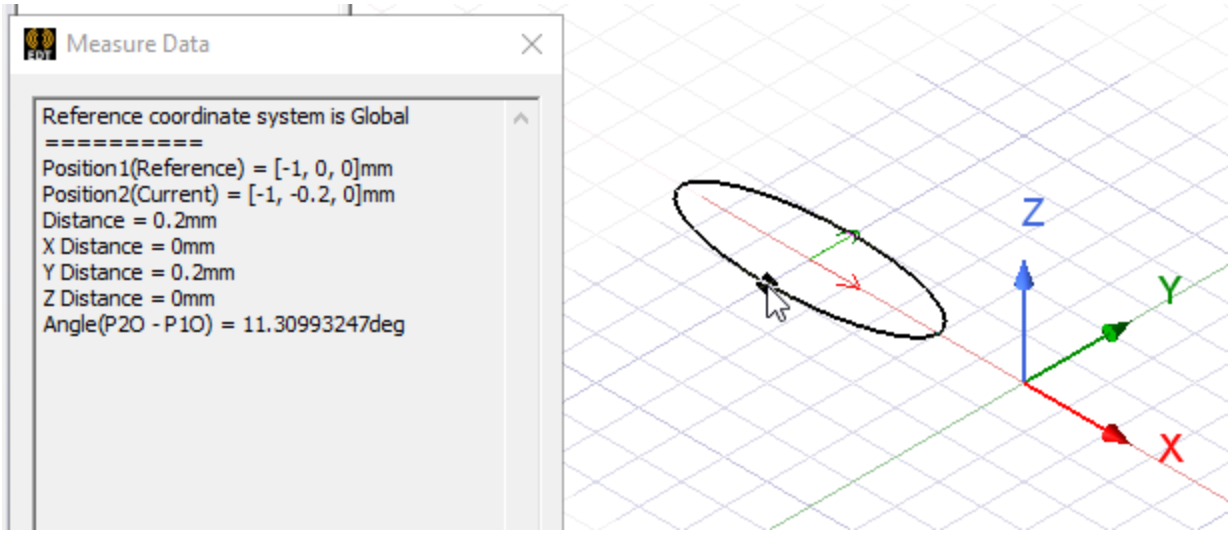
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point. Ansys Electronics Desktop constrains mouse movement to the base radius direction.



- Type the coordinates of a point relative to the center point in the **dX, dY,** or **dZ** box, where **d** is the distance from the previously selected point.

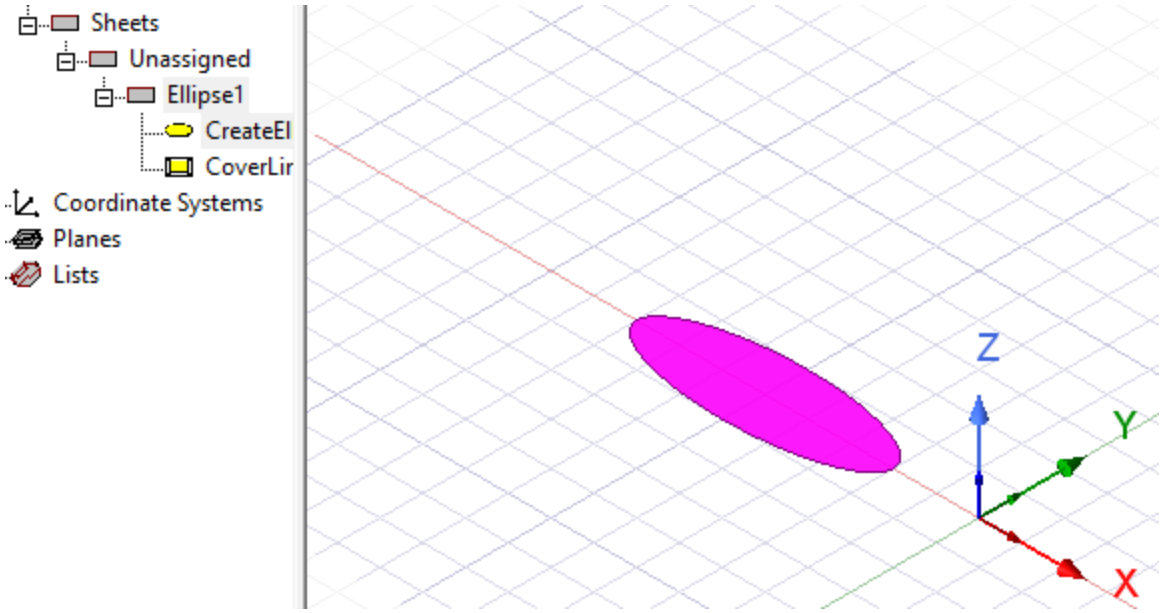
4. Specify the secondary radius of the ellipse. Select the point in one of the following ways:
   - Click the point. Ansys Electronics Desktop constrains mouse movement to a point on the plane orthogonal to the base radius direction.



   - Type the coordinates of a point relative to the center point in the **dXm dY,** or **dZ** box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double click, or right click and select **Done** from the short cut menu. The ellipse appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The **Ratio** value represents the aspect ratio of the secondary radius to the base radius.

   If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the ellipse will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

If the base radius is larger than the secondary radius, the ellipse's longer axis will lie along the default base radius direction. If the secondary radius is larger than the base radius, the ellipse's longer axis will lie perpendicular to the default base radius direction. To create an ellipse with an arbitrary orientation, rotate or move the ellipse after drawing it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

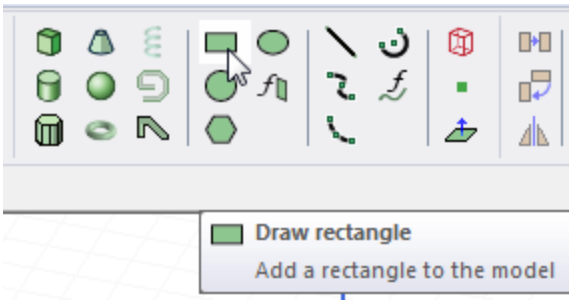
Modifying Surface Approximation Settings and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Rectangle

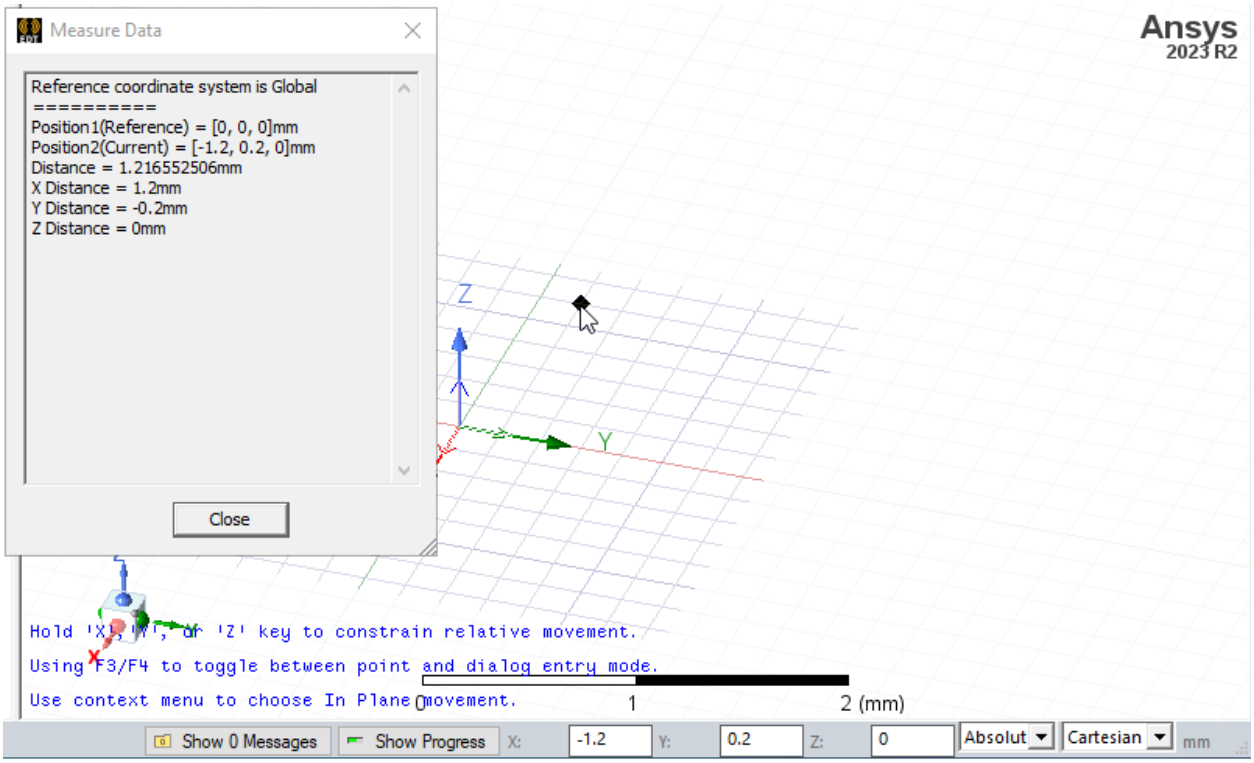
Draw a rectangle (or square) by selecting two diagonally opposite corners. Before you draw a rectangle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Rectangle** or, on the **Draw** ribbon tab, click the **Draw rectangle** icon:

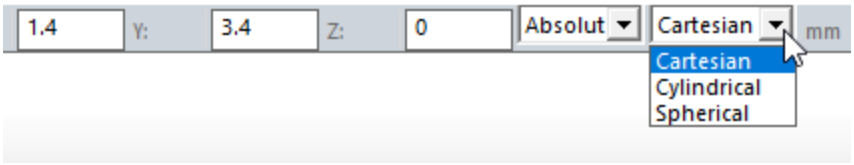


2. Select the first diagonal corner in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



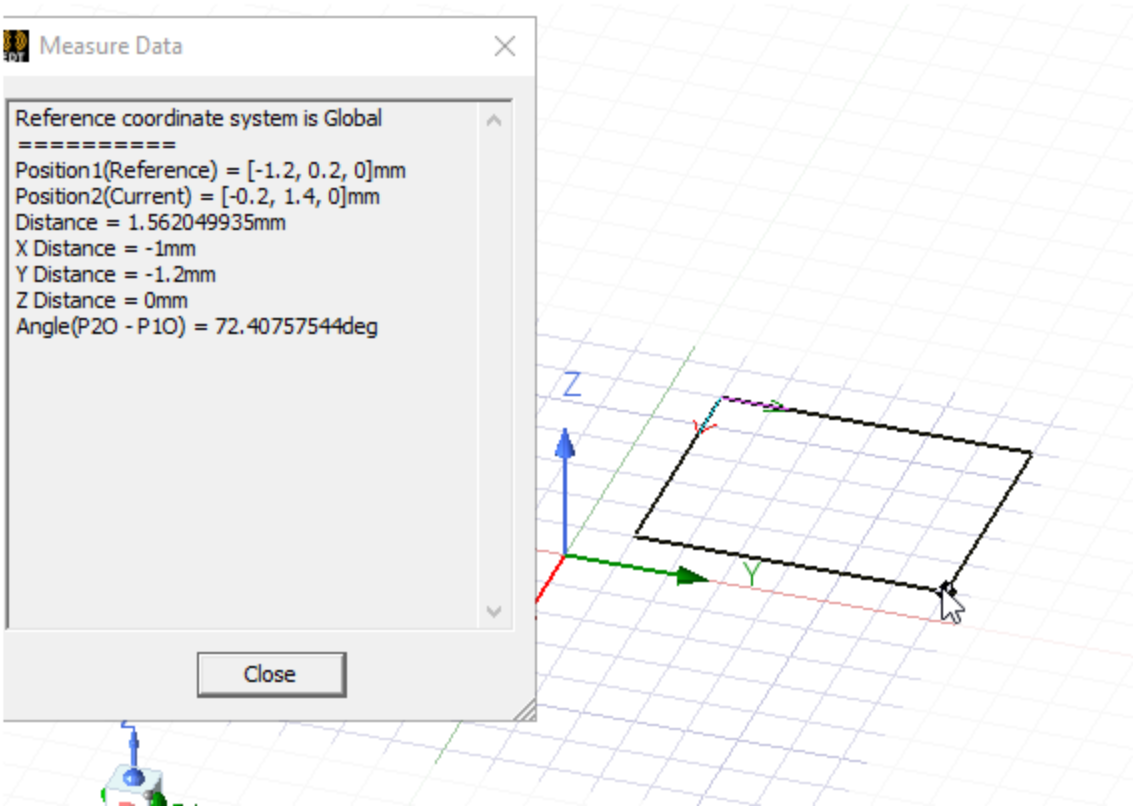
- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cylinder in Cartesian, Cylindrical, or Spherical coordinates.



To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the second corner of the rectangle in one of the following ways:
   - Click the point.

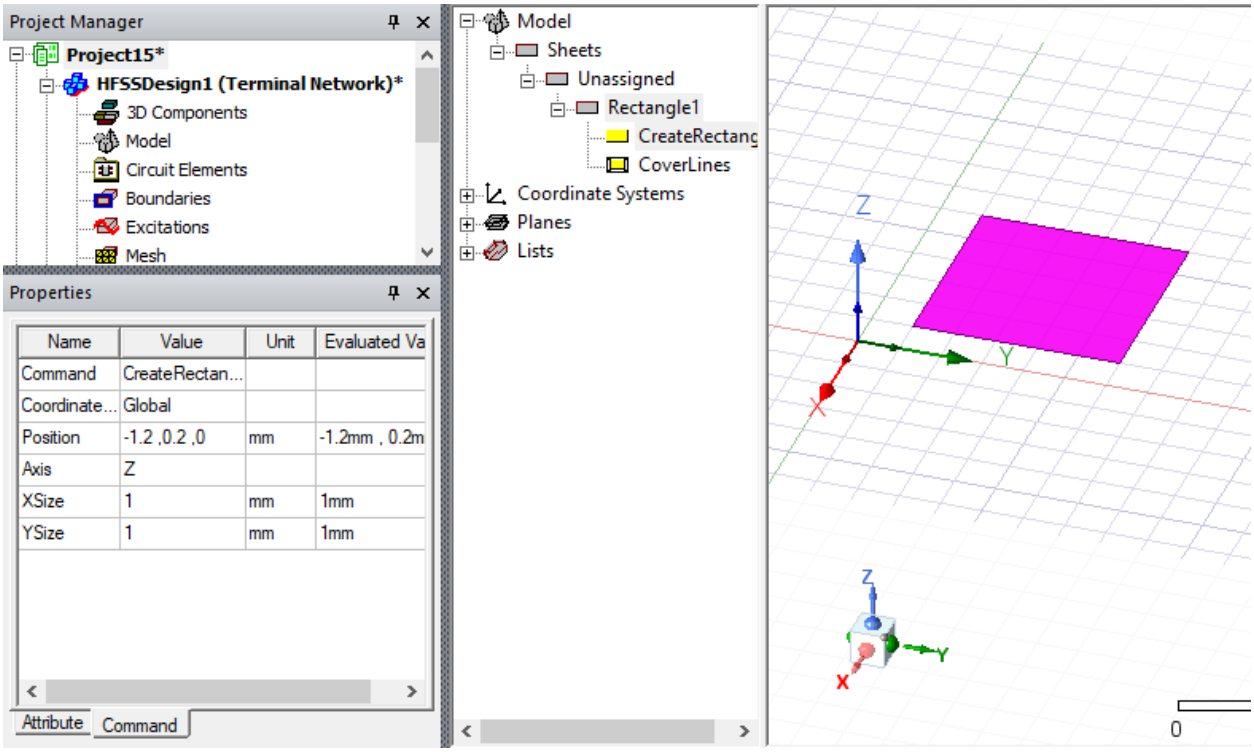


   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

4. Click **OK**. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the rectangle will be covered, resulting in a 2D sheet object, as shown in the History tree in the figure below. Otherwise it will be a closed 1D polyline object.

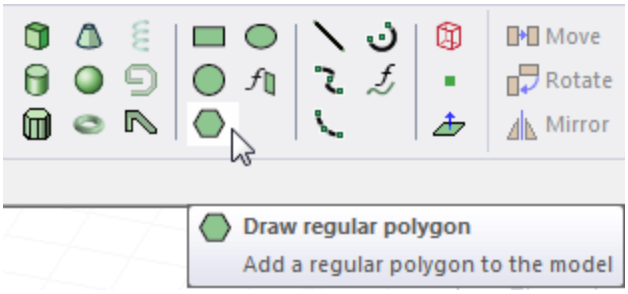
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing a Regular Polygon

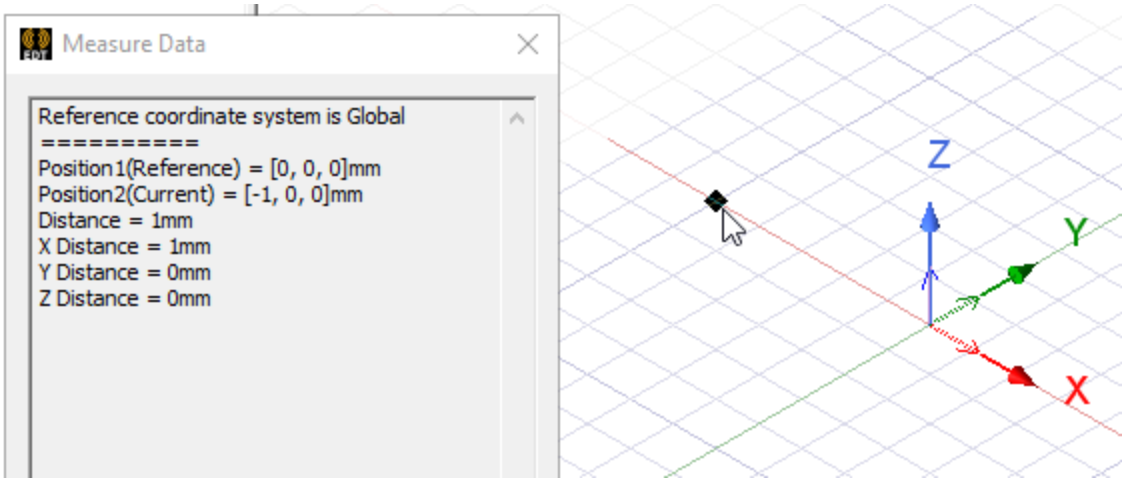
A regular polygon is a 2D object with three or more equal sides. Regular polygons are useful for drawing faceted 2D objects. Before you draw a regular polygon, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw>** **Regular Polygon** or, on the **Draw** ribbon tab, click the **Draw regular polygon** icon:



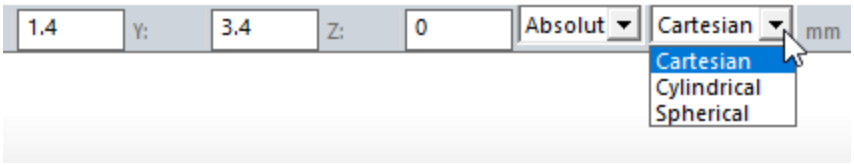
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the polygon in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

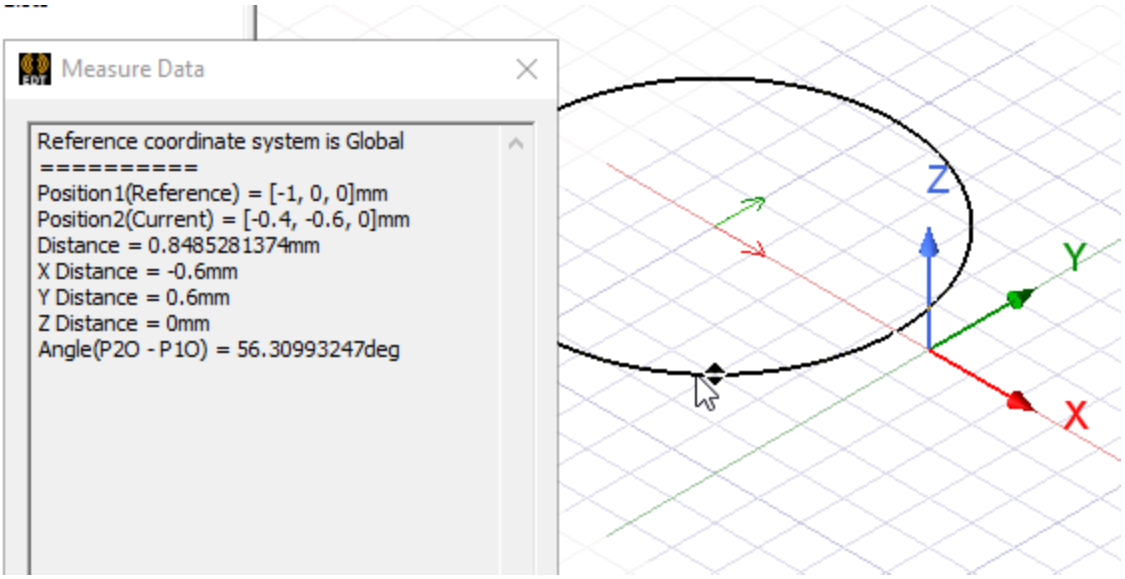
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the polygon's radius, the distance from the center point to one of the polygon's vertices, in one of the following ways:

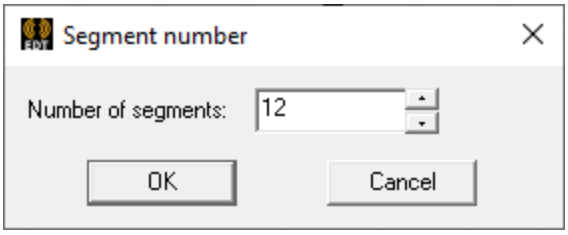
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



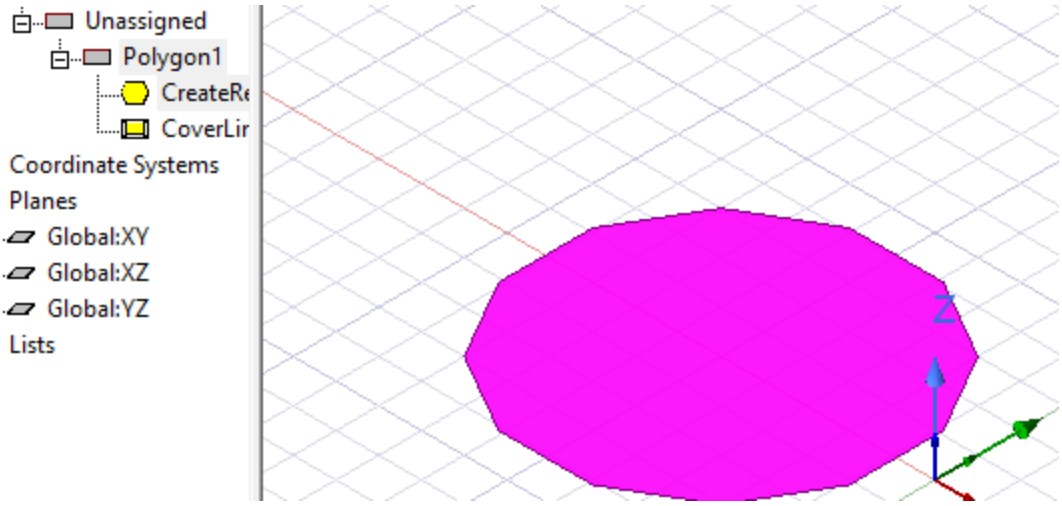
- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. In the **Segment number** dialog box, enter the **Number of segments** in the polygon, and then click **OK**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click OK. The polygon appears in the Modeler window and in the History tree.



5. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If you have edited the Properties dialog, click **OK**.

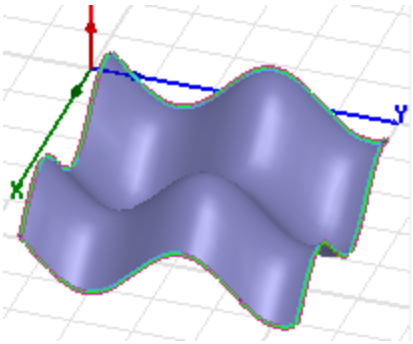
> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

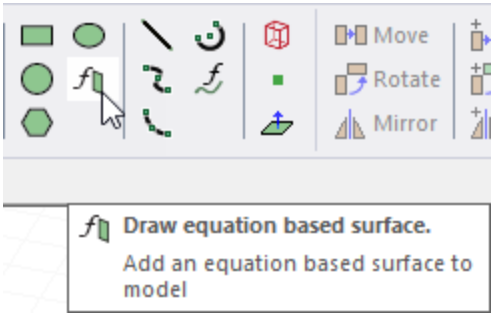
Drawing an Equation-Based Surface

Any surface that can be described by an equation in three dimensions can be drawn. Before you draw an equation-based surface, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

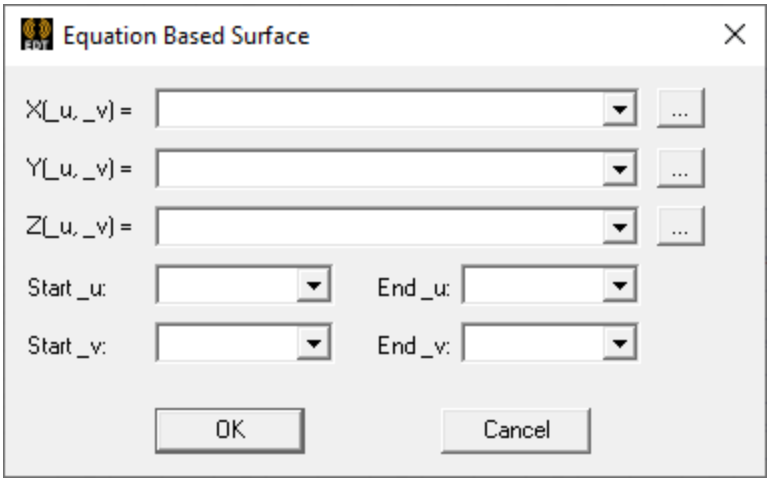
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. From the menu bar, click **Draw**> **Equation Based Surface** or, on the **Draw** ribbon tab, click the **Draw equation based surface** icon:



The **Equation Based Surface** dialog box opens.



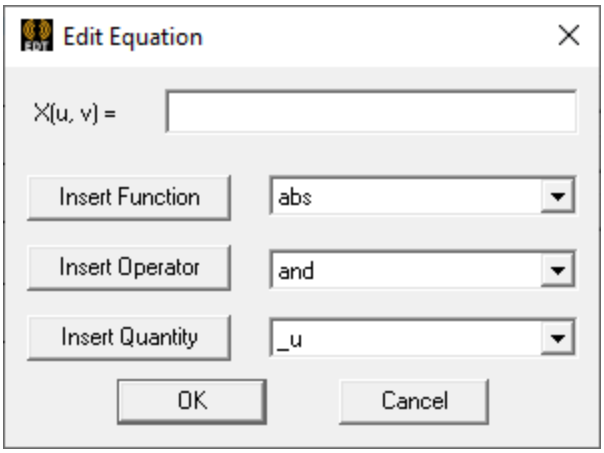
2. Type equations for **X(_u, _v)**, **Y(_u, _v)**, and **Z(_u, _v)**.

   You can also define an equation by doing the following:

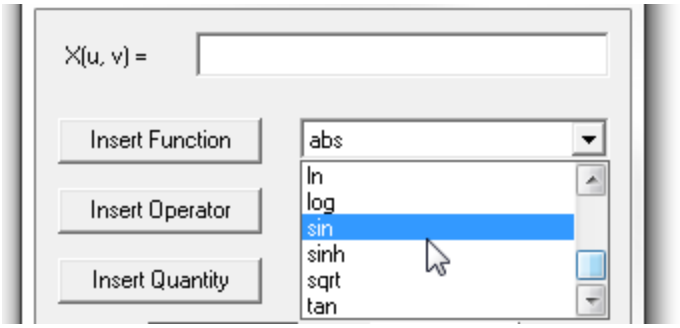
   a. Click the ellipsis button (**...**).

      The **Edit Equation** dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



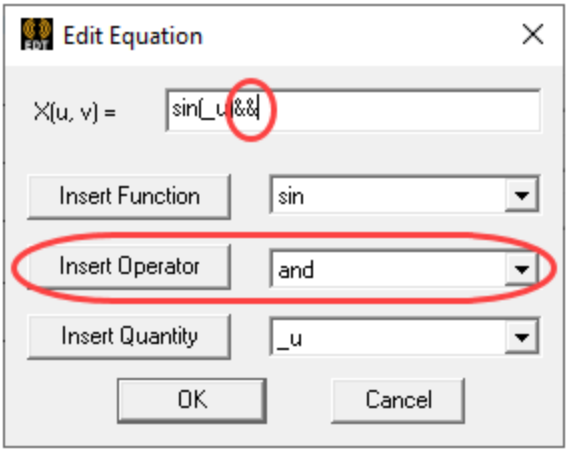
b. Do one or more of the following to define the equation:
   - Type a numerical value, variable, or expression directly in the text box
   - Build an equation for X(u, v), Y(u, v), or Z(u, v), by selecting the function, operator, and quantity to insert into the equation from the three drop-down menus:

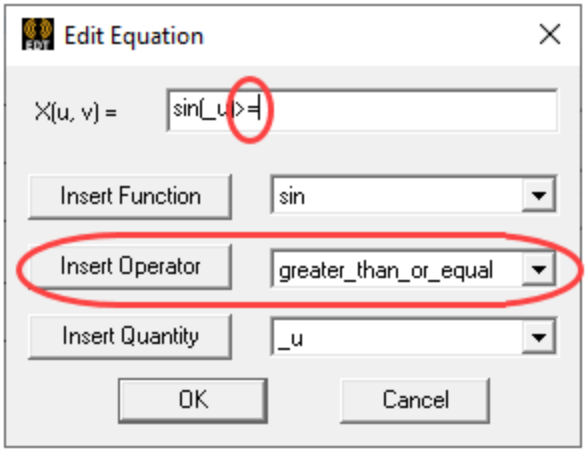


     Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

   - The **Insert Operator** button inserts corresponding operator:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



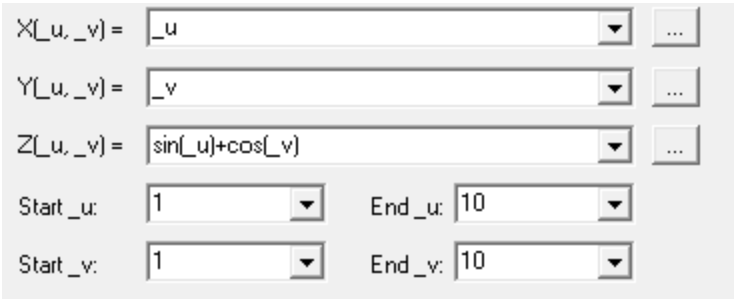


- You can combine the preceding two methods to build complex equations.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Surface** dialog box.
   - The equations are taken to be meters. Use of units in equations is not properly supported..
   - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
   - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
   - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Specify start values for the **Start_u** and **Start_v** fields.

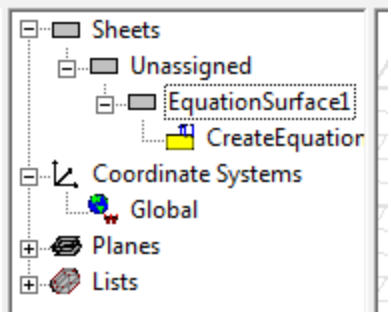
   Select end values for **End_u** and **End_v** fields.



4. Click **OK** on the **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The surface is drawn in the Modeler window. The History tree contains the Equation Surface object and the command properties.

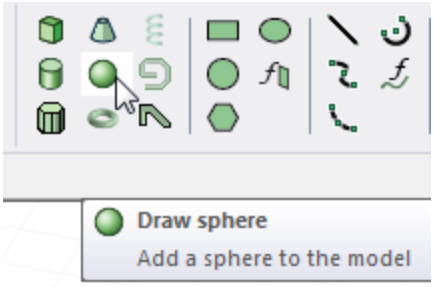


If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes.

Drawing a Sphere

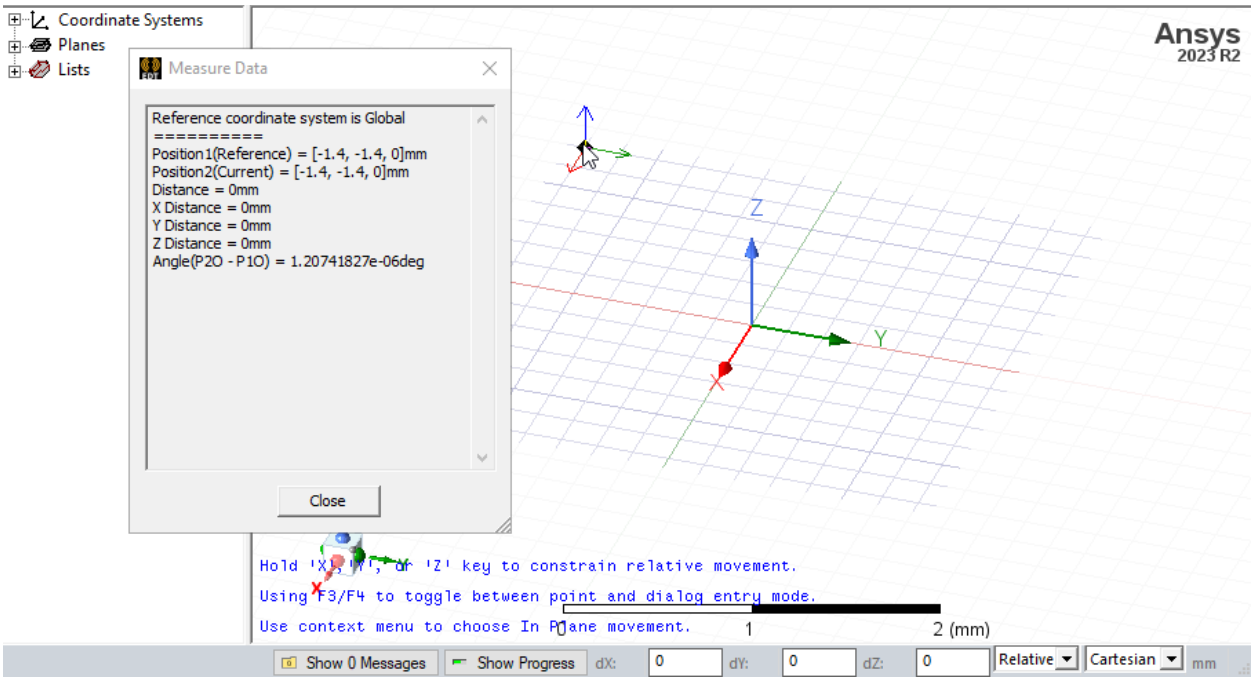
Draw a sphere, a 3D circle, by selecting a center point and a radius. Spheres are drawn as true surfaces in the modeler. Before you draw a sphere, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Sphere** or, on the **Draw** ribbon tab, click the **Draw sphere** icon:



2. Select the center point of the sphere in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

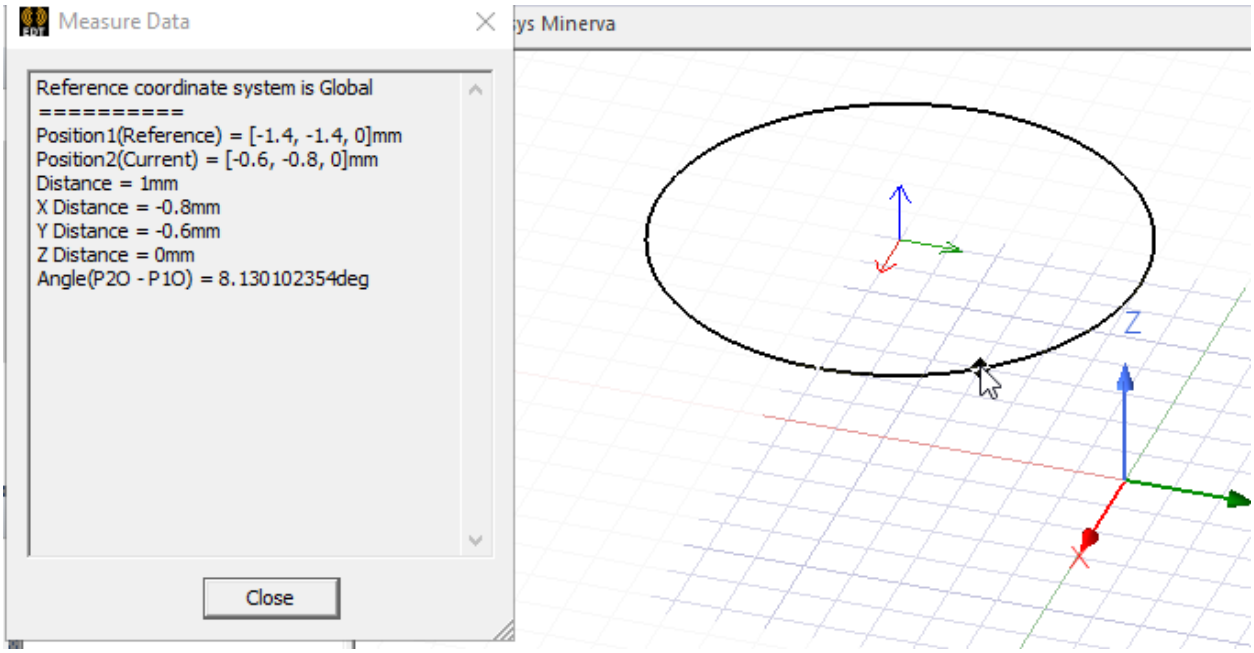


- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

3. Specify the radius by selecting a point on the sphere's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

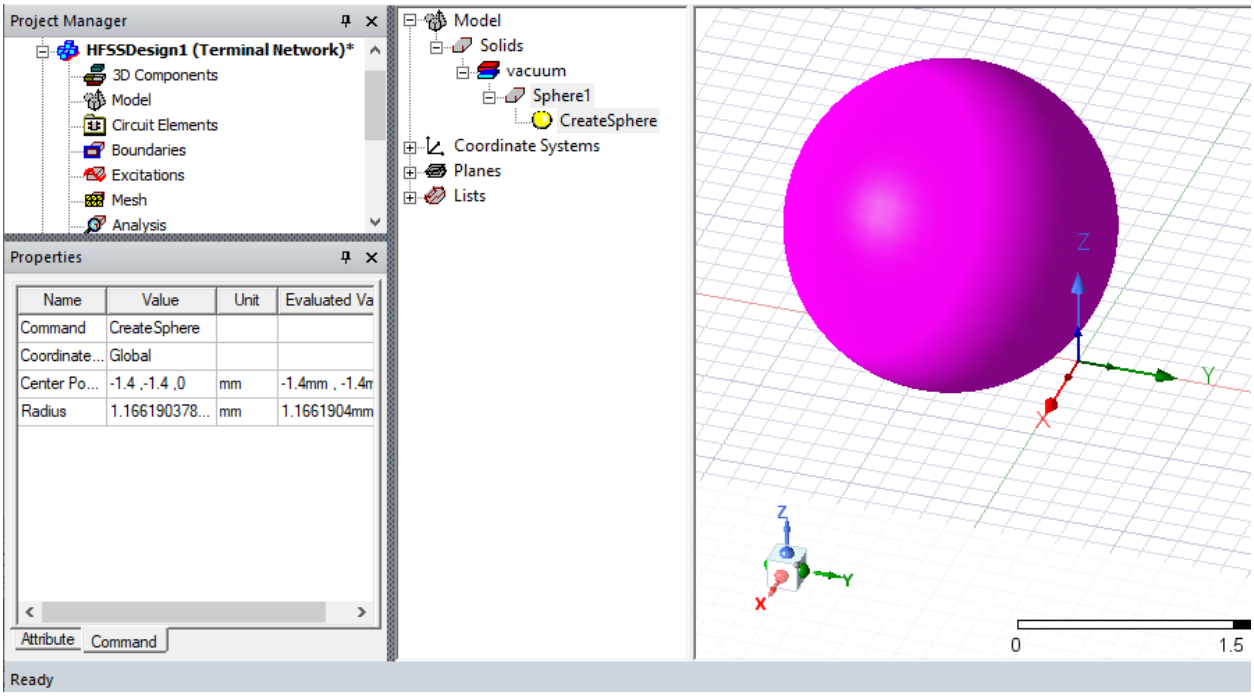
- Click the point



- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK**.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Modifying Surface Approximation Settings and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Related Topics

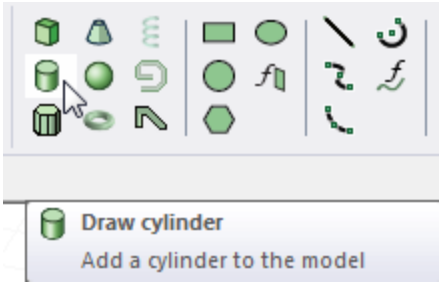
Modifying Objects

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

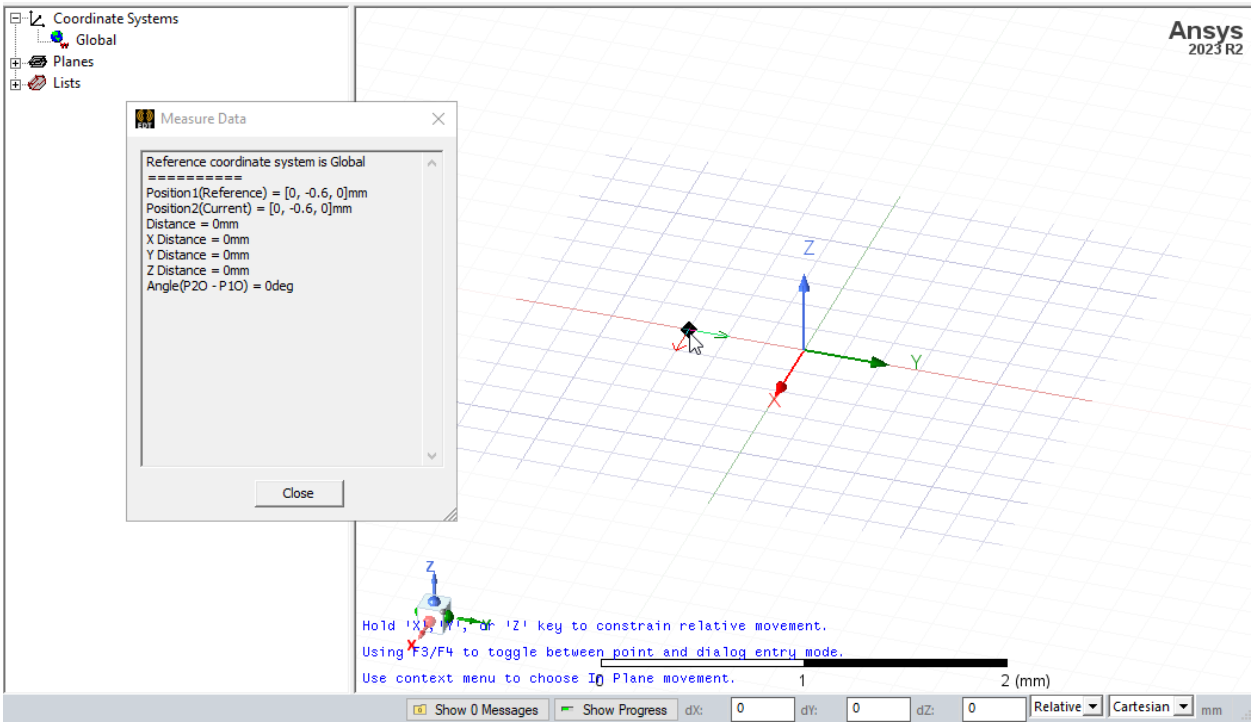
Drawing a Cylinder

Draw a cylinder by selecting a center point, radius, and height. Cylinders are drawn as true surfaces in the modeler. Before you draw a cylinder, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Cylinder** or, on the **Draw** ribbon tab, click the **Draw cylinder** icon:



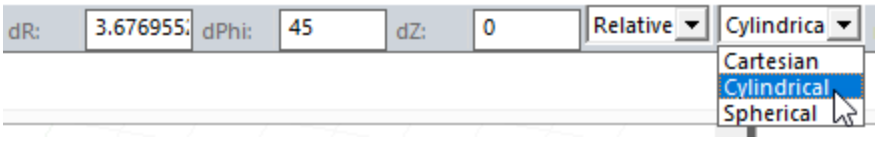
2. Select the center point of the cylinder's base circle in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.



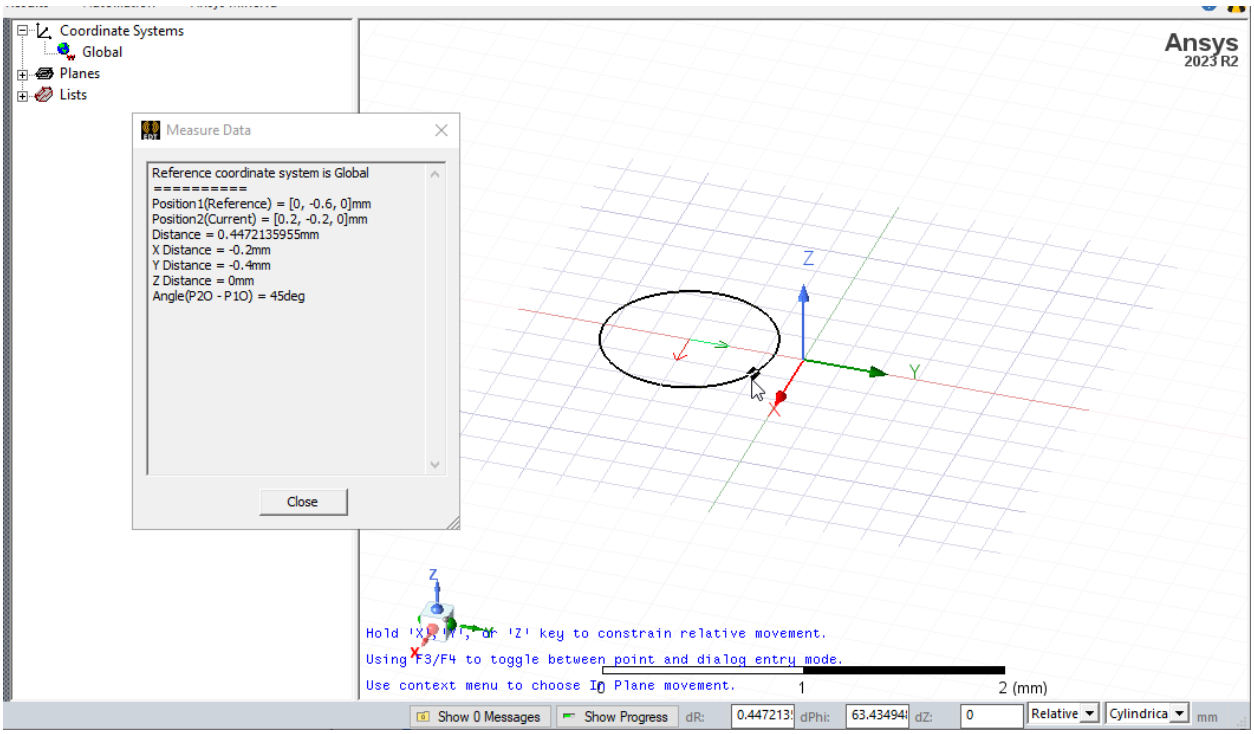
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cylinder in Cartesian, Cylindrical, or

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spherical coordinates.

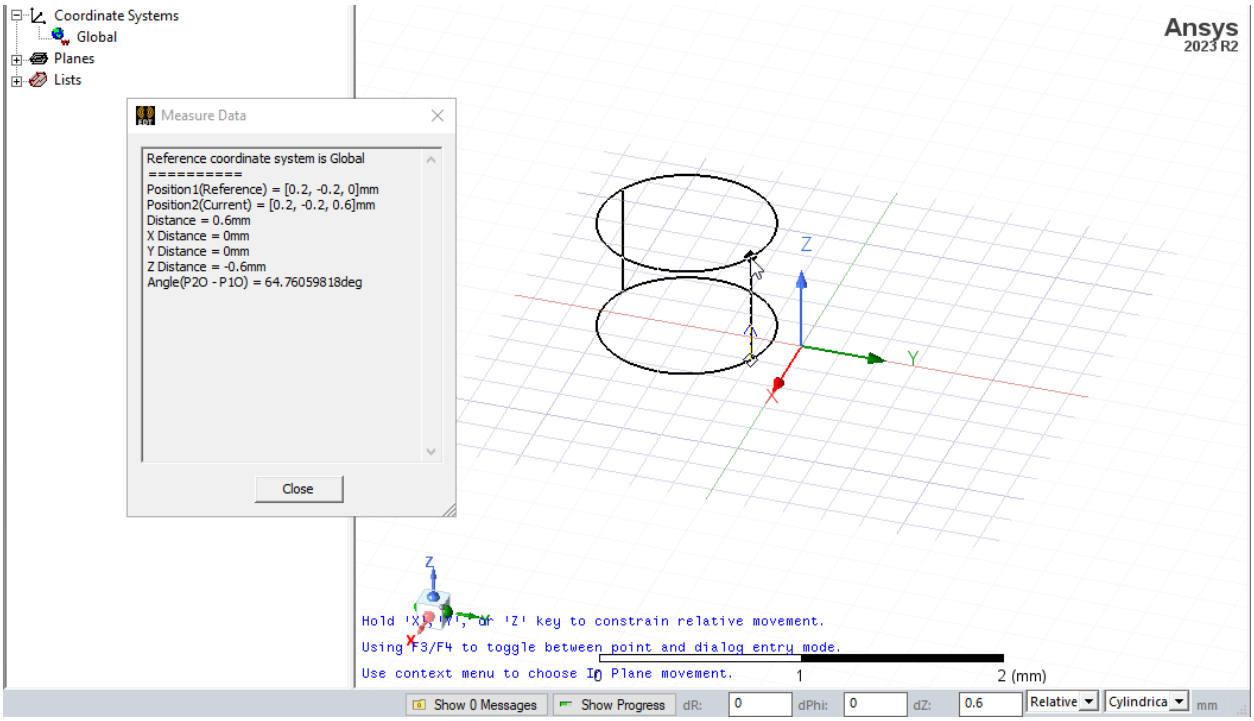


3. Specify the radius by selecting a point on the base circle's circumference in one of the following ways:
   - Click the point.



   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the cylinder's height by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



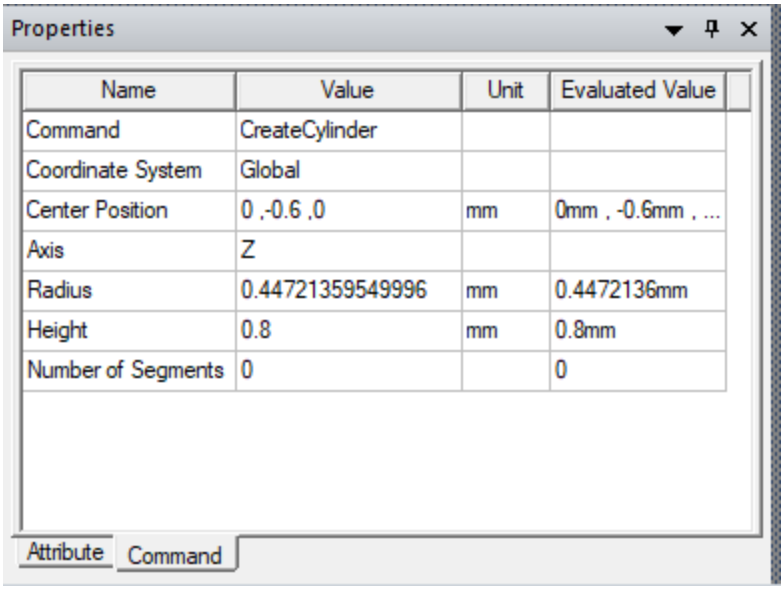
Note:

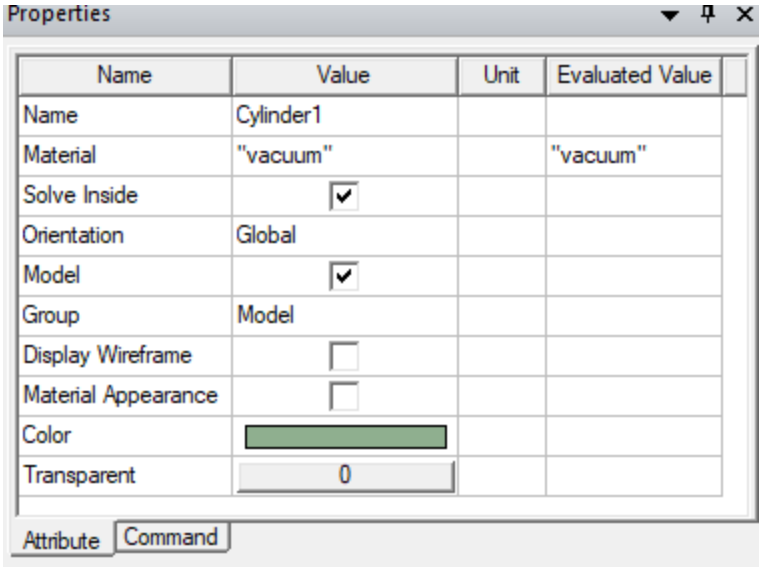
If you create a cylinder with a height of zero, Ansys Electronics Desktop draws a circular sheet object.

Double click, or right click and select **Done** from the short cut menu. If the Modeler option for editing properties of new primitives is checked, the *Properties* dialog box appears, enabling you to modify the object's Command and Attribute properties. You can also use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

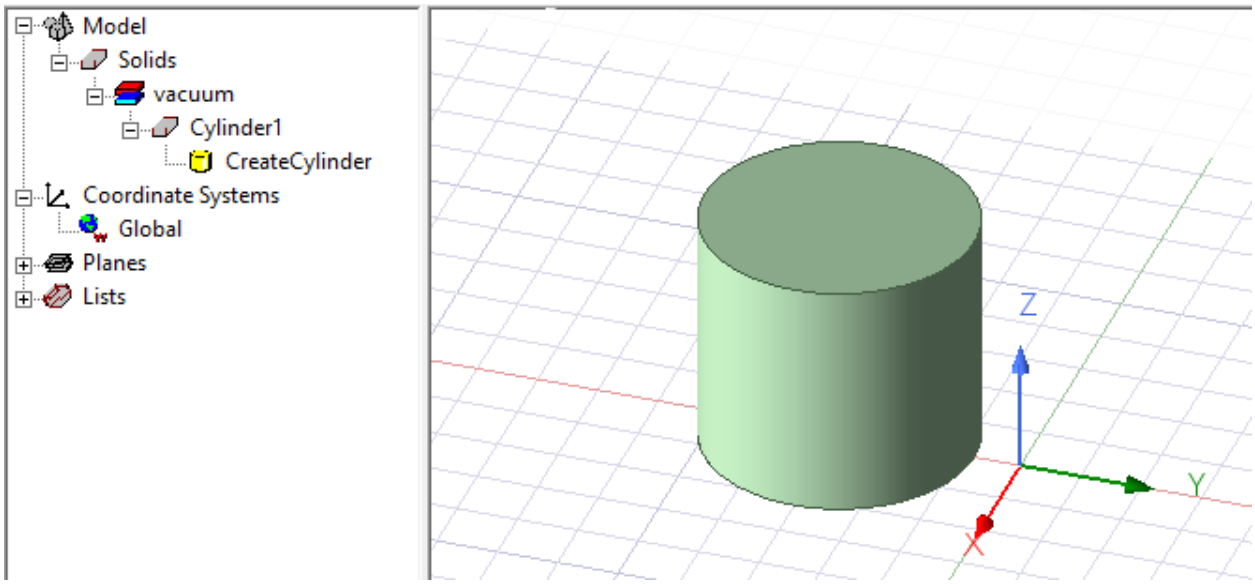
the docked Properties window.





5. If you used the Properties window, click **OK**. The cylinder appears in the Modeler window and is listed in the History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Modifying Surface Approximation Settings and related sections:

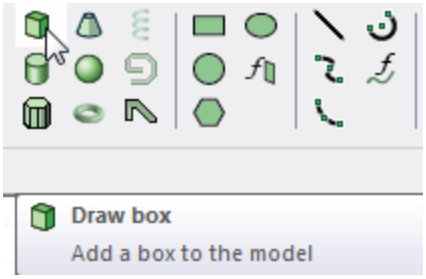
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Box

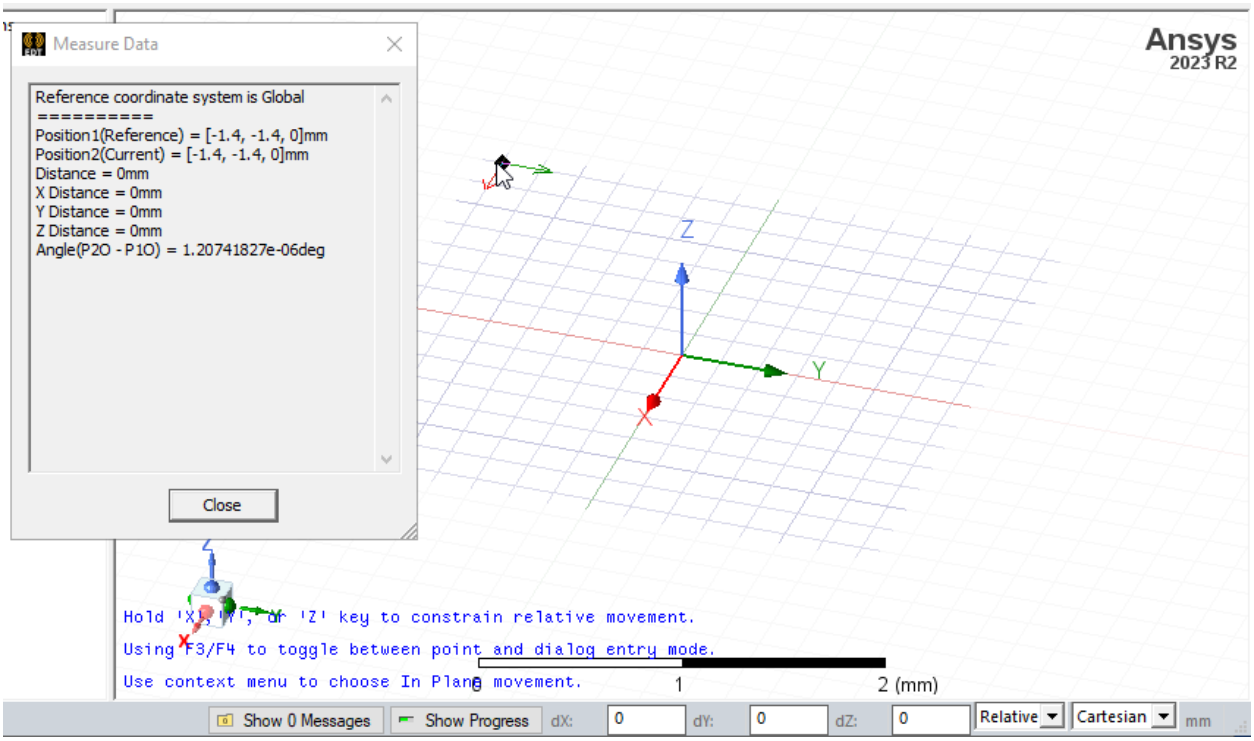
Draw a box by selecting two diagonally opposite corners of the base rectangle, then specifying the height. Before you draw a box, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Box** or, on the **Draw** tab of the ribbon, click the **Draw box** icon:

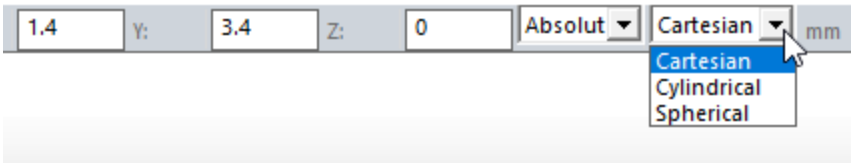
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the first diagonal corner of the base rectangle in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.



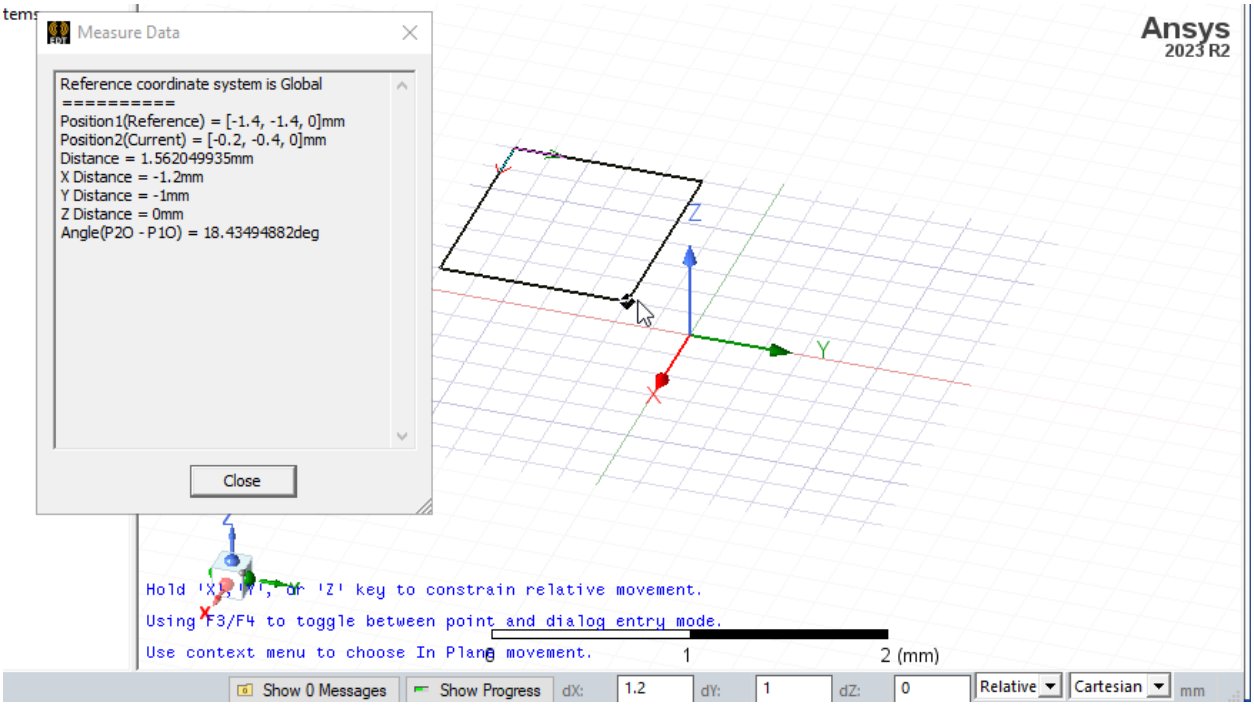
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the box in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

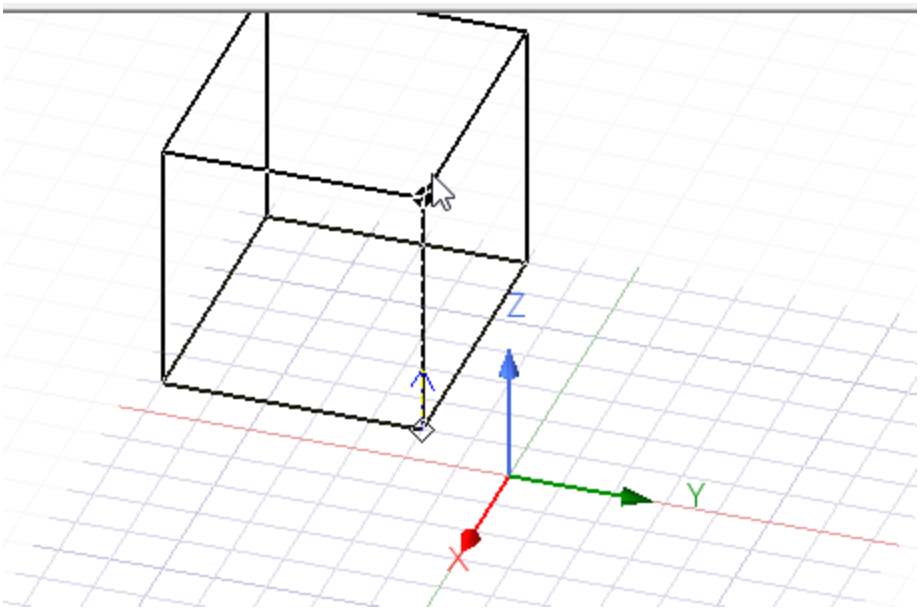
To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

3. Select the second corner of the base rectangle in one of the following ways:
   - Click the point.



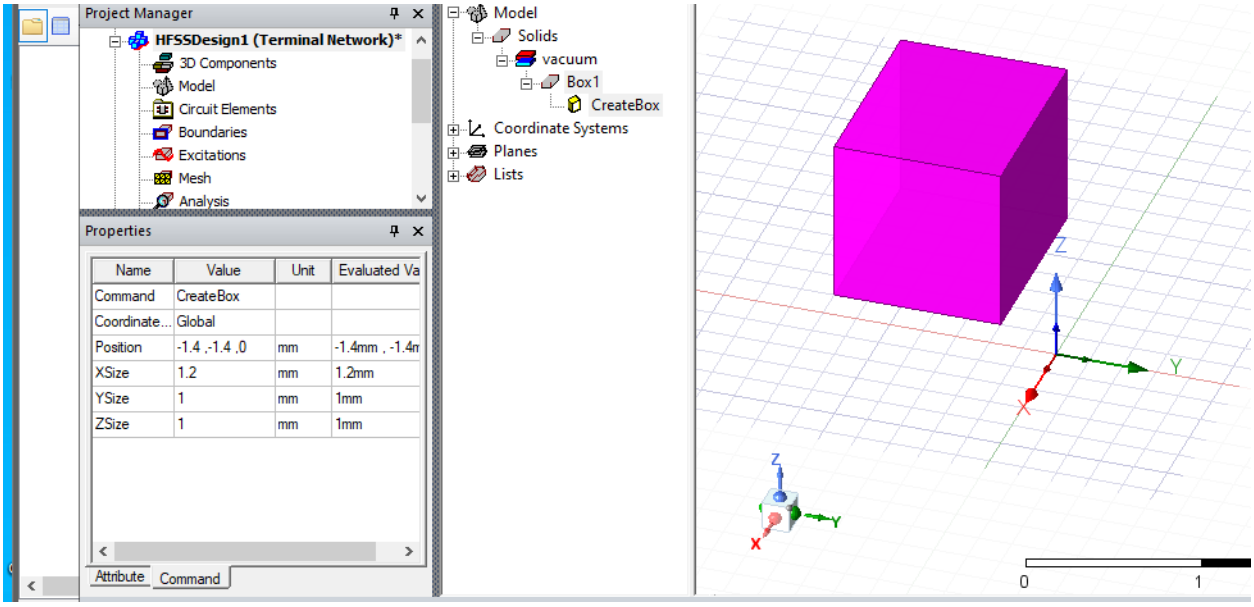
   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the height of the box by selecting a point on the axis perpendicular to the base rectangle. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Double -click or right-click and select **Done** from the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

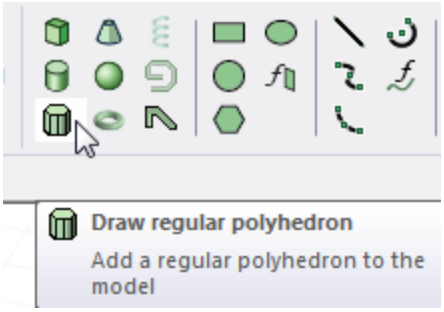


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

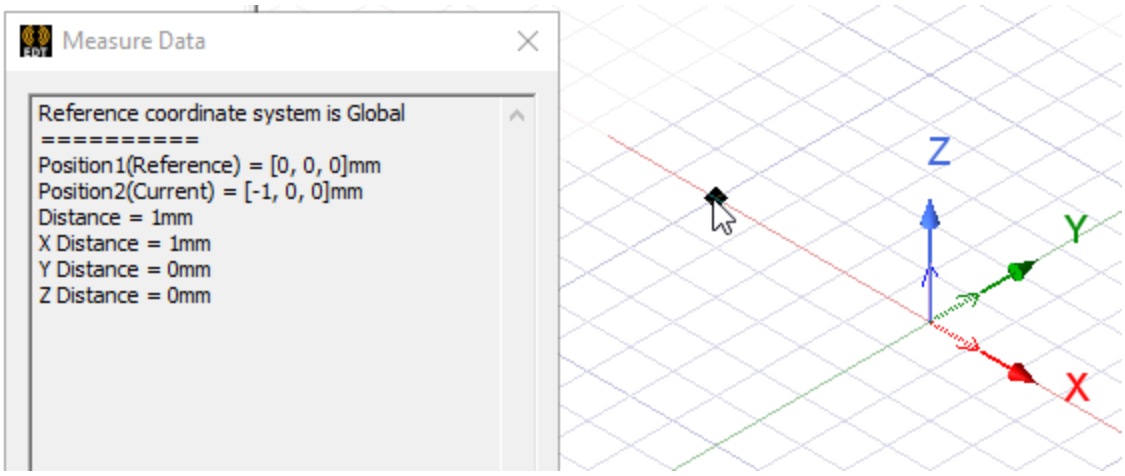
Drawing a Regular Polyhedron

In the modeler, regular polyhedrons are 3D objects with regular polygon faces; each face has three or more equal sides. Regular polyhedrons are useful for drawing faceted 3D objects. Before you draw a regular polyhedron, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

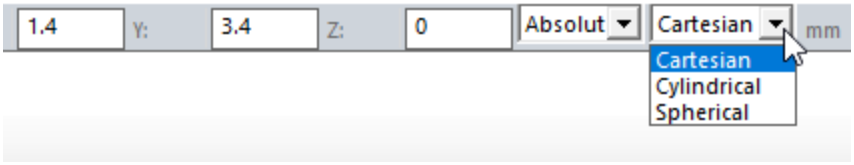
1. From the menu bar, click **Draw**> **Regular Polyhedron** or, on the **Draw** ribbon tab, click the **Draw regular polyhedron** icon:



2. Select the center point of the polyhedron in one of the following ways:
   - Click the point.

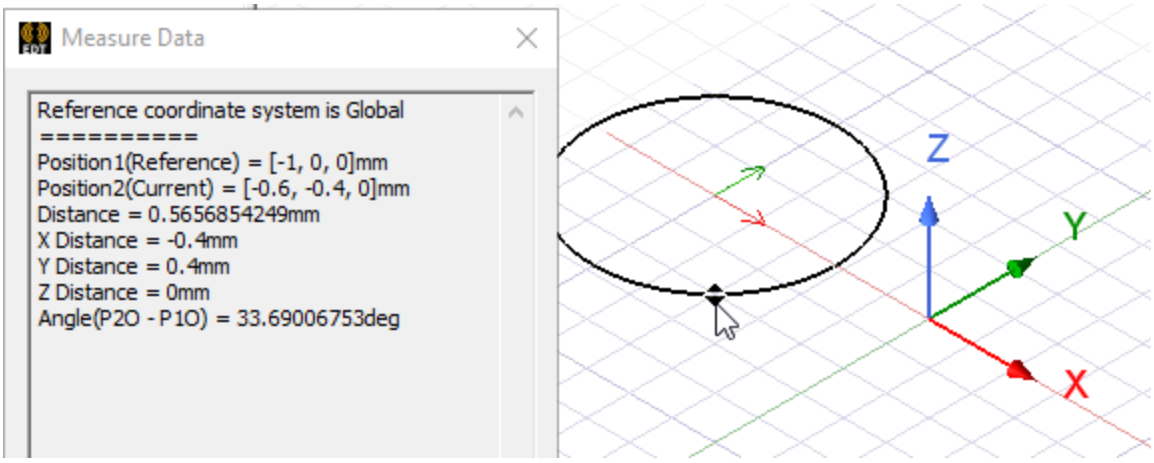


   - Type the point's coordinates in the text boxes in the status bar.
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



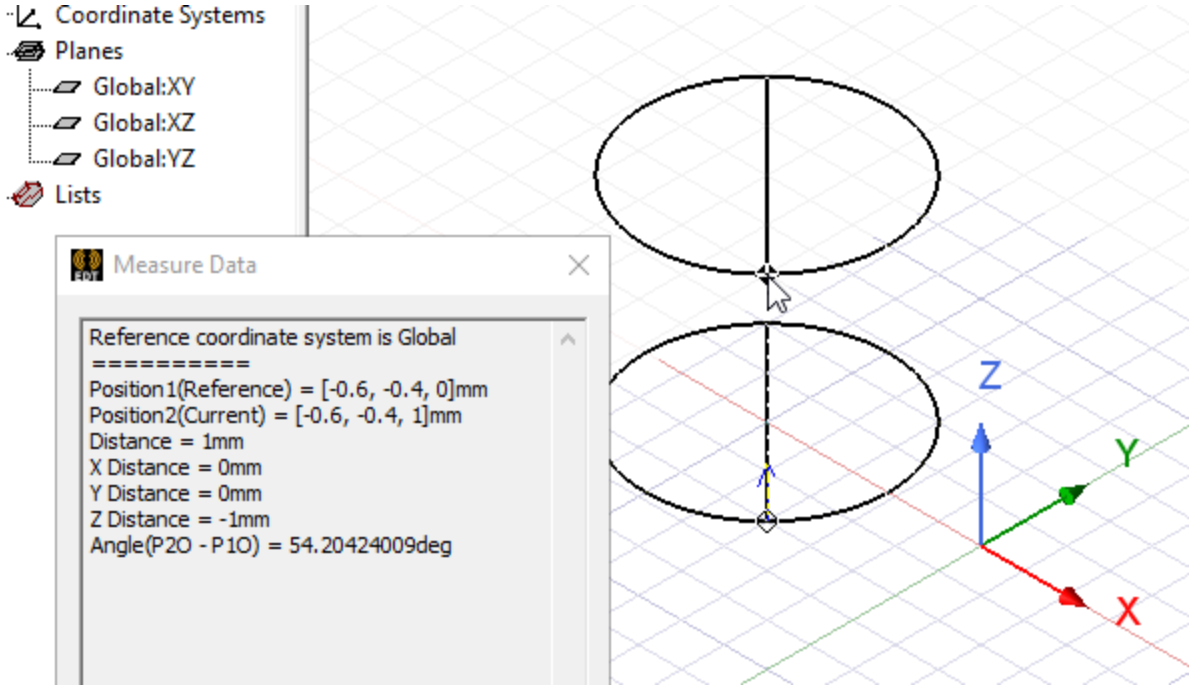
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the radius of the polyhedron, the distance from the center point to one of the polyhedron's vertices, in one of the following ways:
   - Click the point.



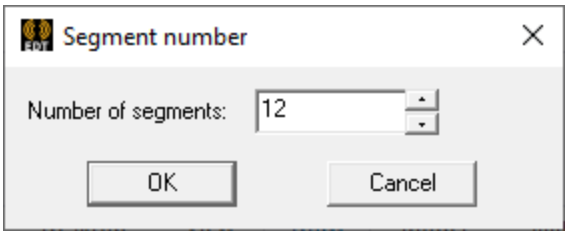
   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Select the height of the polyhedron, the distance from the origin plane along the normal from the radius point.

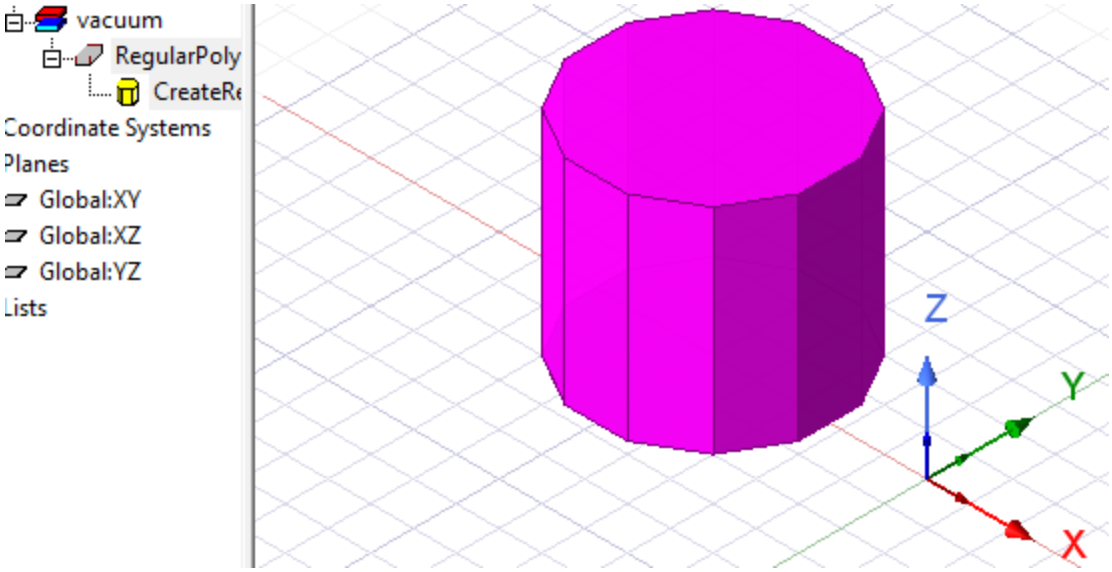


5. In the **Segment number** dialog box, enter the **Number of segments** in the polyhedron, and then click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **OK**. The **Segment number** dialog closes and the regular polyhedron appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

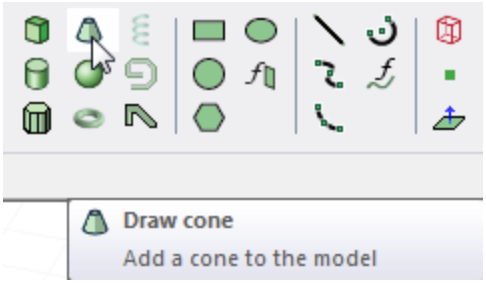
If the **Automatically cover closed polyline** option is selected in the **Modeler Options** dialog box, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

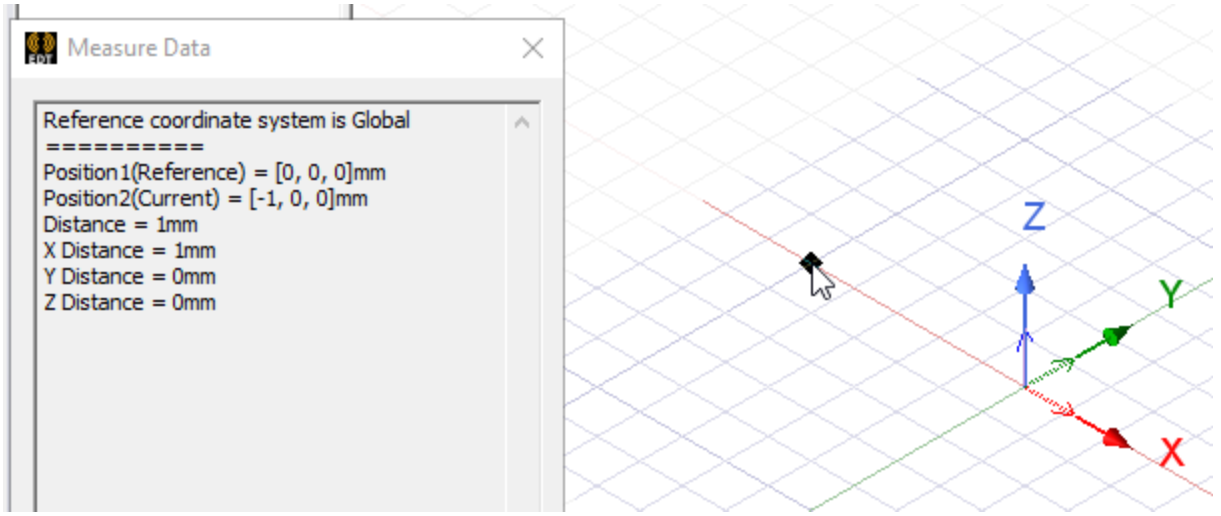
Drawing a Cone

Draw a cone by selecting the center point and radius of the cone's base circle, then specifying the radius of the cone's top circle and the cone's height. Cones are drawn as true surfaces in the modeler. Before you draw a cone, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

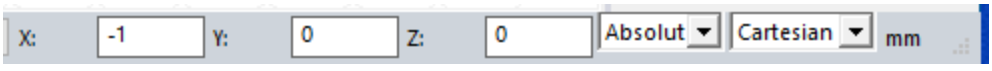
1. From the menu bar, click **Draw**> **Cone** or, on the **Draw** ribbon tab, click the **Draw cone** icon:



2. Select the center point of the cone's base circle in one of the following ways:
   - Click the point.



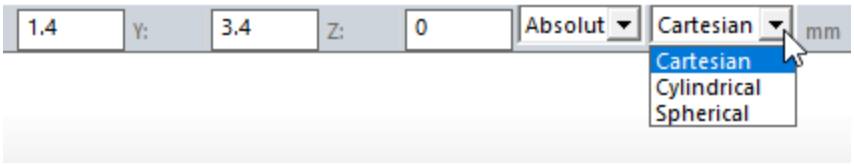
   - Type the point's coordinates in the text boxes in the status bar.



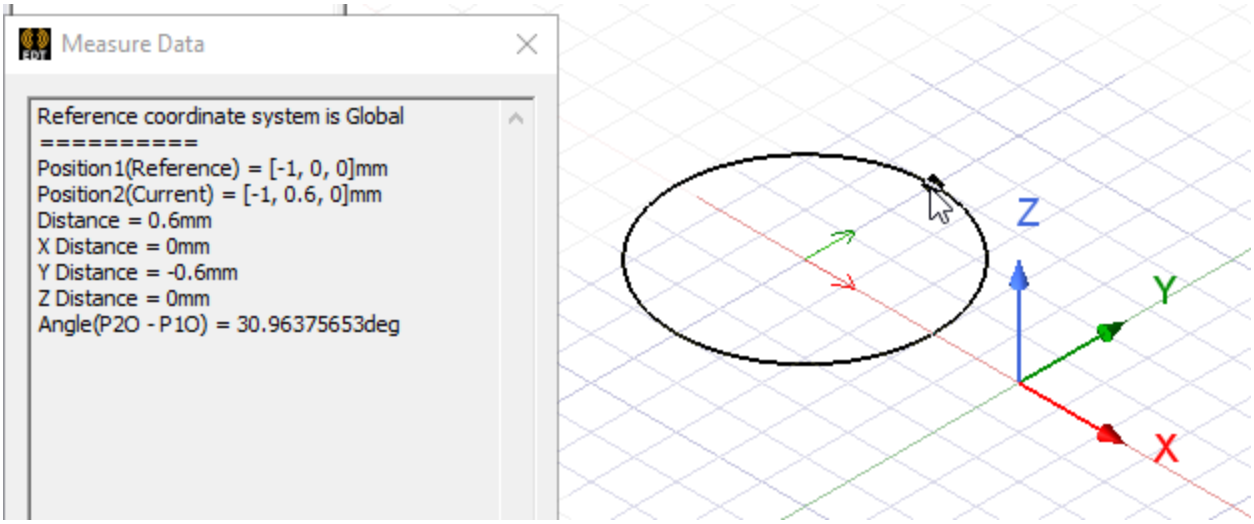
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cone in Cartesian,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

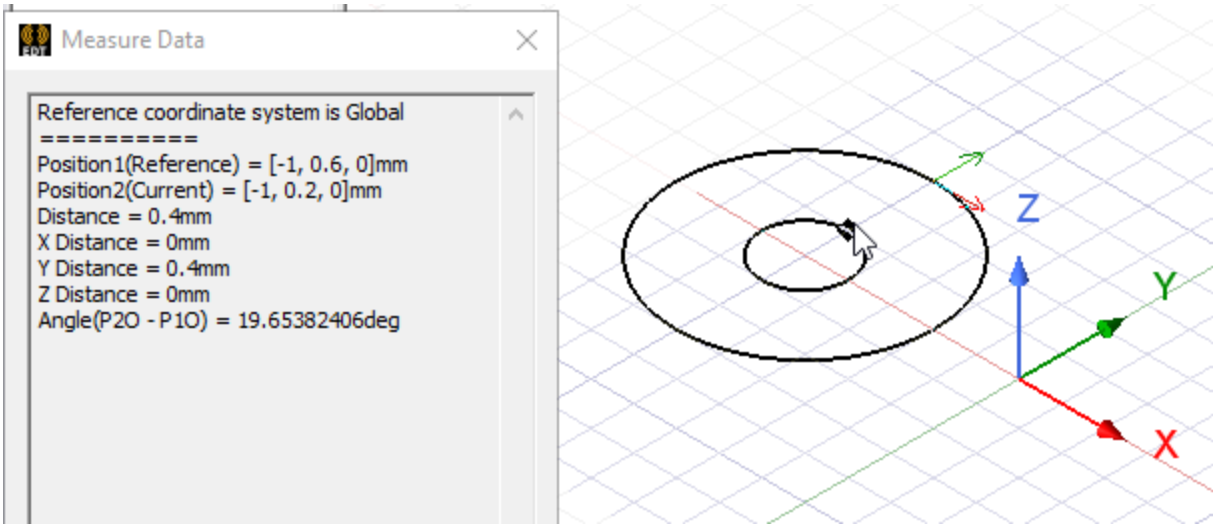
Cylindrical, or Spherical coordinates.



3. Specify the radius of the cone's base circle by selecting a point on the base circle's circumference. Select the point in one of the following ways:
   - Click the point.

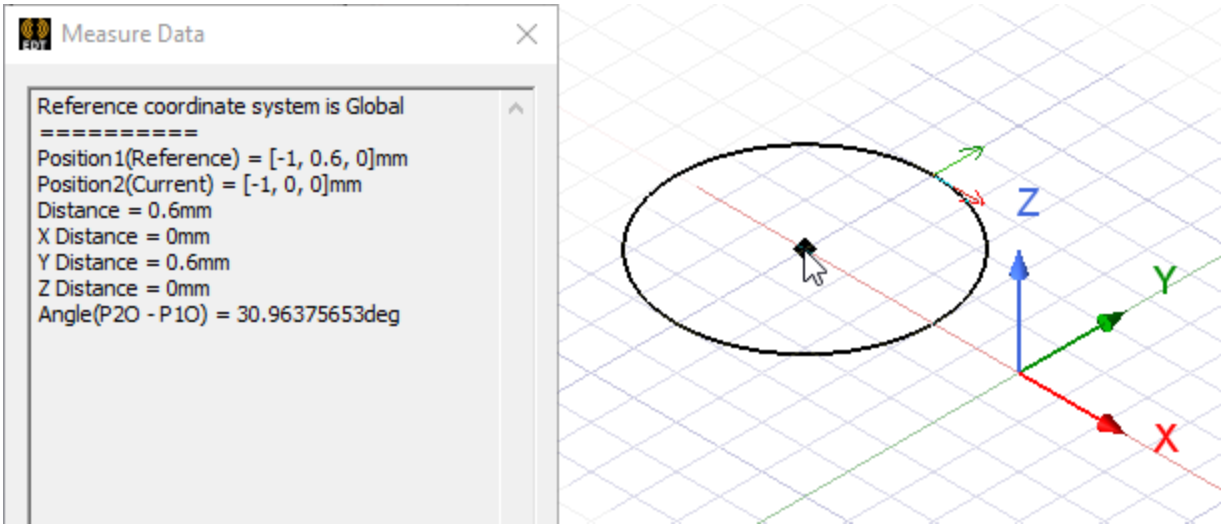


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the radius of the cone's top circle by selecting a point on its circumference. Select the point by clicking it or typing its coordinates in the **dX, dY,** and **dZ** boxes.

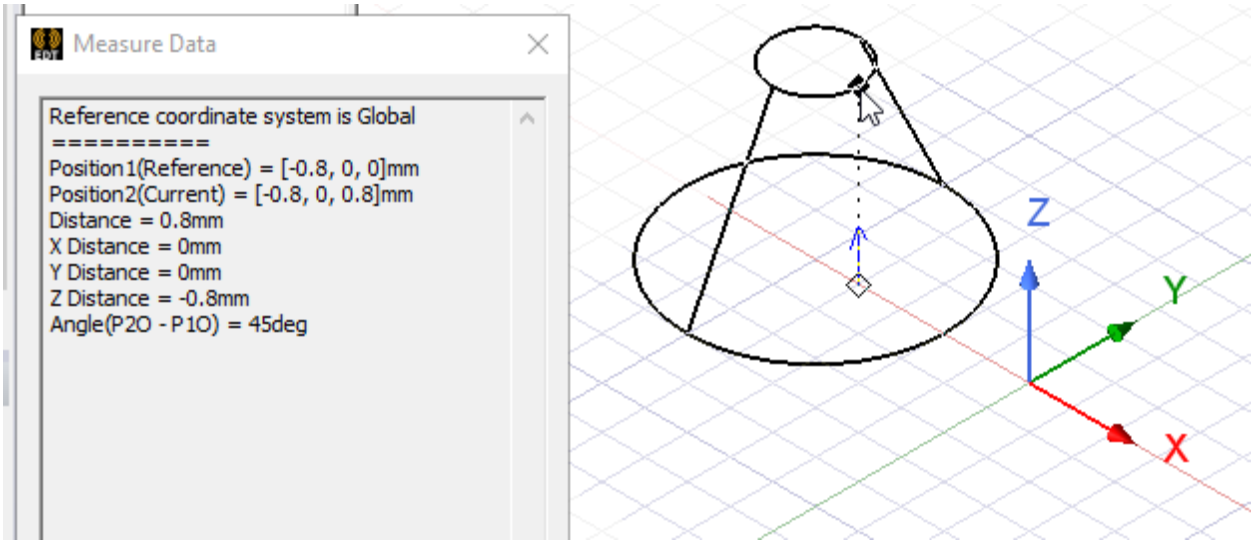


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create an apex, select the same center point as the cone's base circle.



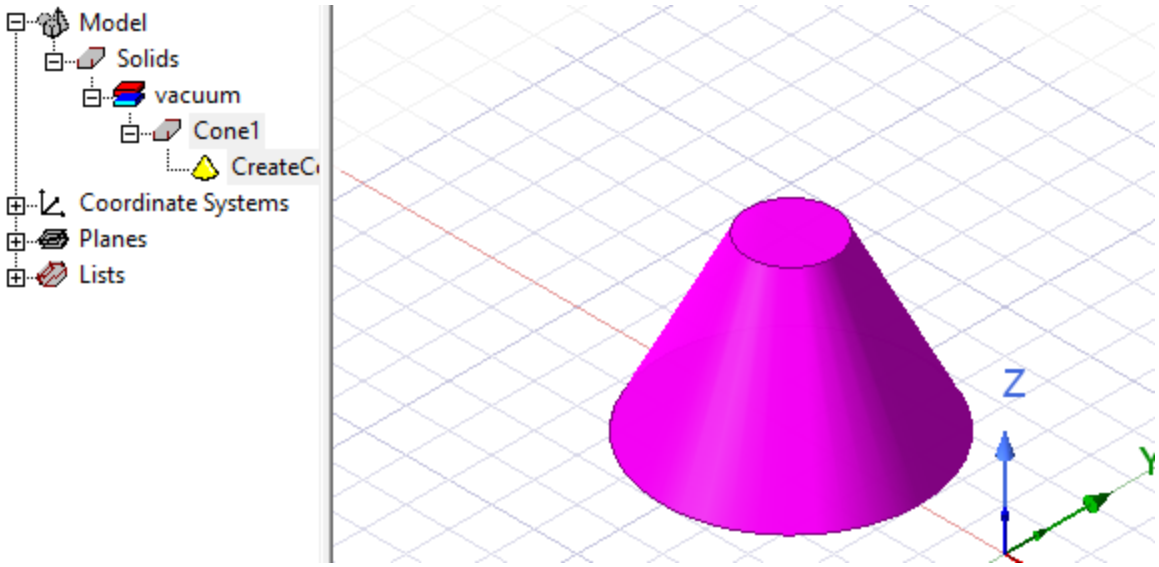
5. Specify the height of the cone by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.



6. Double click, or right click and select **Done** from the short cut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The cone appears in the Modeler window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and in the History Tree.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Modifying Surface Approximation Settings and related sections:

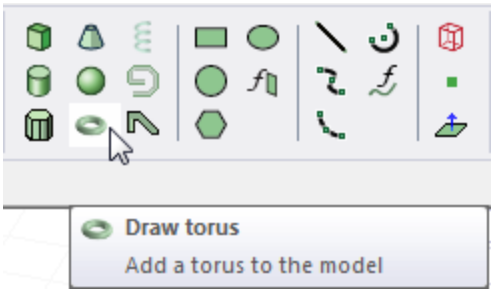
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Torus

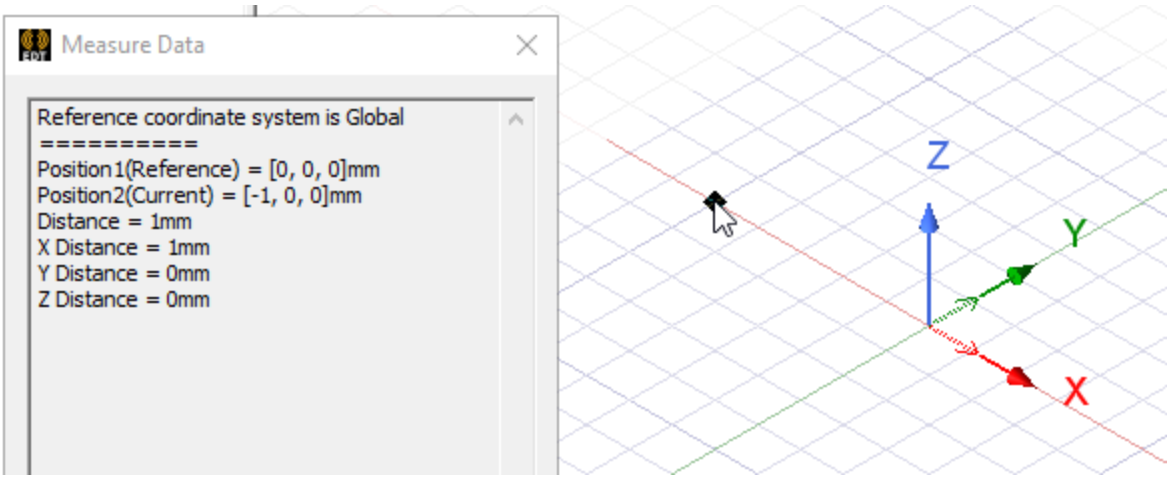
Draw a torus by selecting its center point, major radius, and minor radius. The modeler then sweeps a circle around a circular path. A torus is drawn as true surfaces in the modeler. Before you draw a torus, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Torus** or, on the **Draw** ribbon tab, click the **Draw torus** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

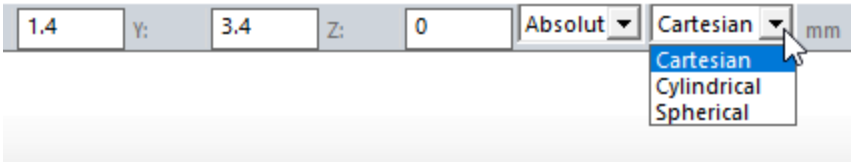


2. Select the center point of the torus in one of the following ways:
   - Click the point.



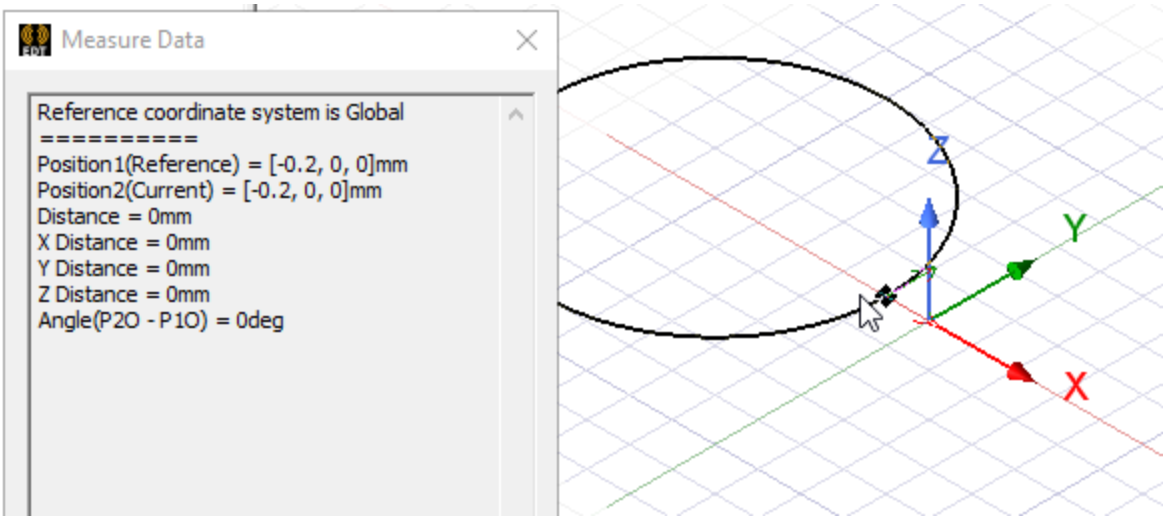
   - Type the point's coordinates in the text boxes in the status bar.

     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the torus in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the major radius by selecting a point in one of the following ways:
   - Click the point.

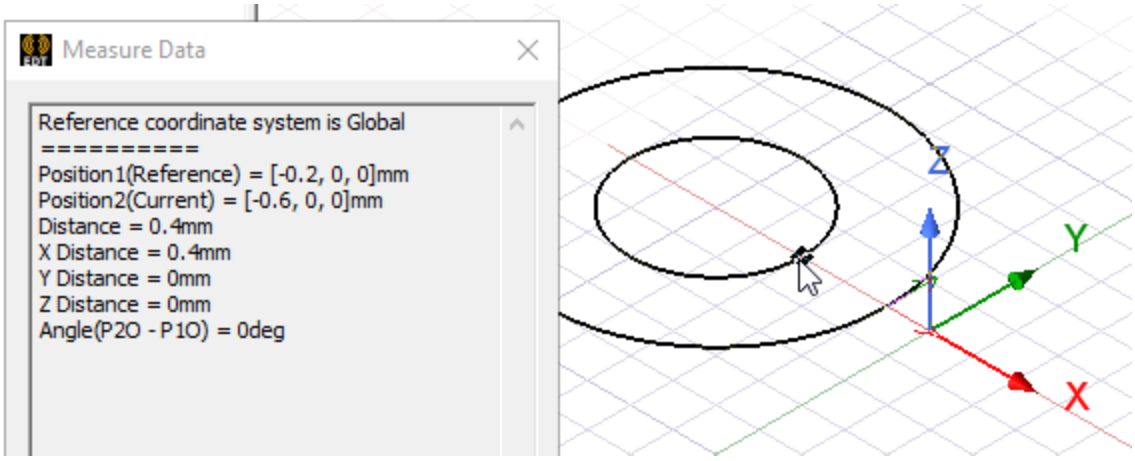


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The major radius determines the diameter of the torus.

4. Specify the minor radius by selecting a point relative to the major radius point.

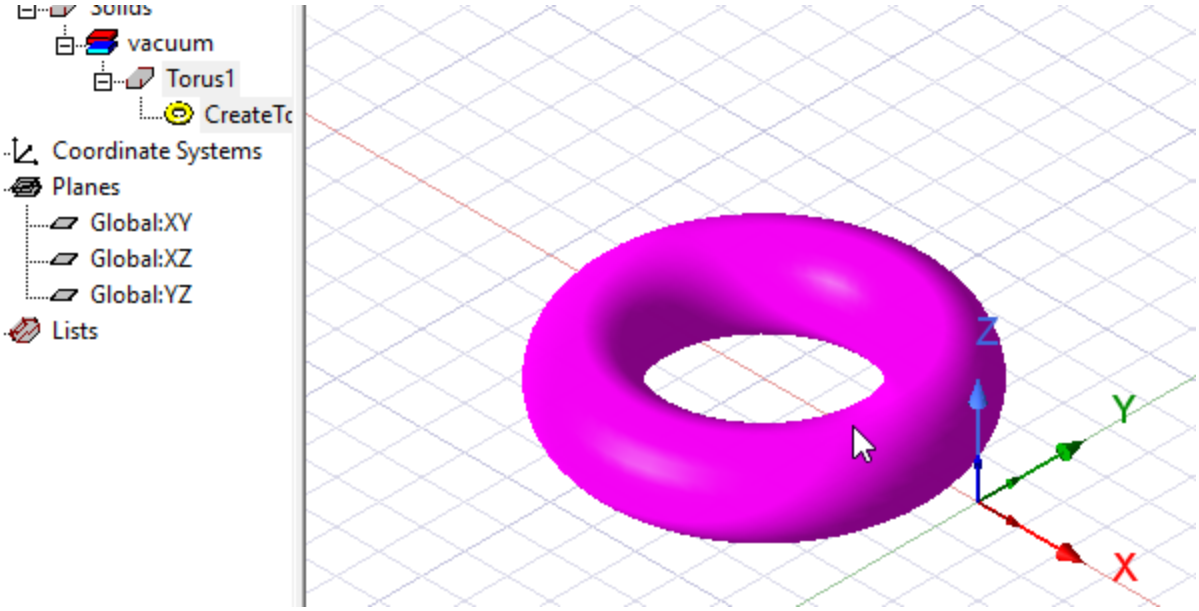
   The minor radius determines the diameter of the "donut hole".



   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double-click or right-click to open the shortcut menu and click **Done**. The torus appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

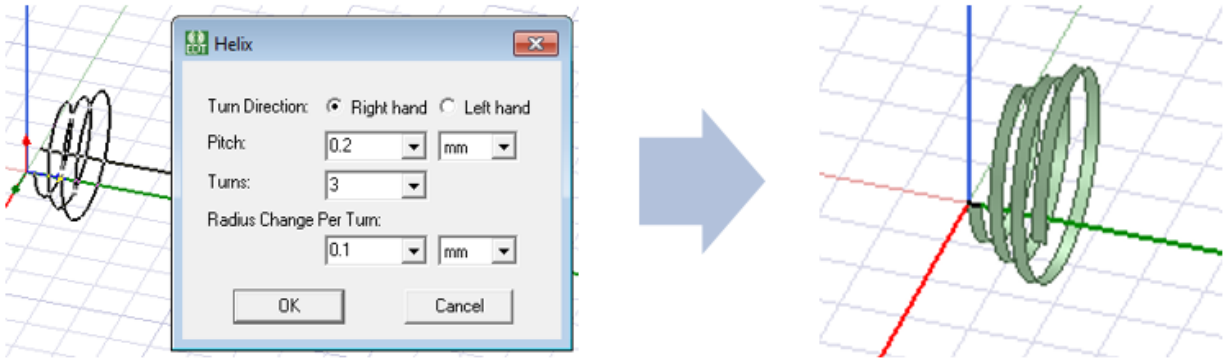
> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:
>
> Modifying Surface Approximation Settings and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Helix

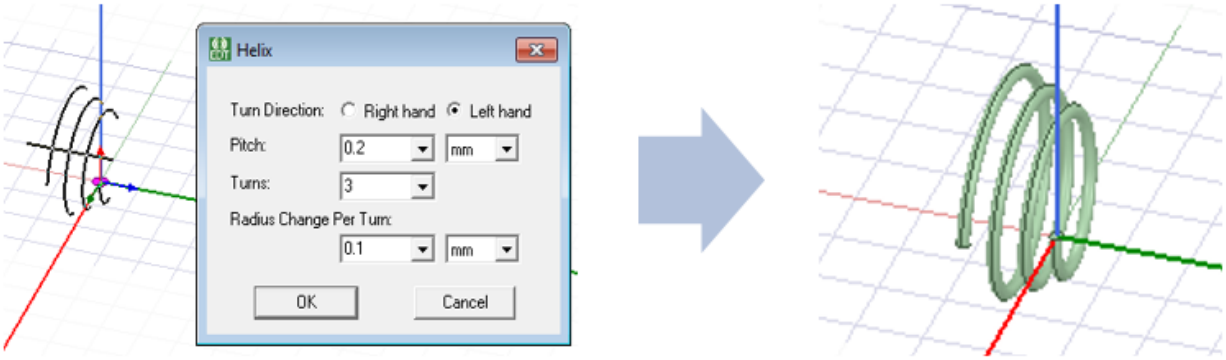
A helix is a 3D spiral object created by sweeping a line or 2D sheet object that you create along a vector that you define. The center of the helix is defined by the placement of the vector relative to the swept object. You must define the vector so that the swept object will not self-intersect, the rule being, do not draw the vector so that it centers on or overlaps a selected 2D object. After you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

create the vector, the **Helix** dialog opens, where you then specify the turn direction, pitch, number of turns, and radius of the turn to define the helix.

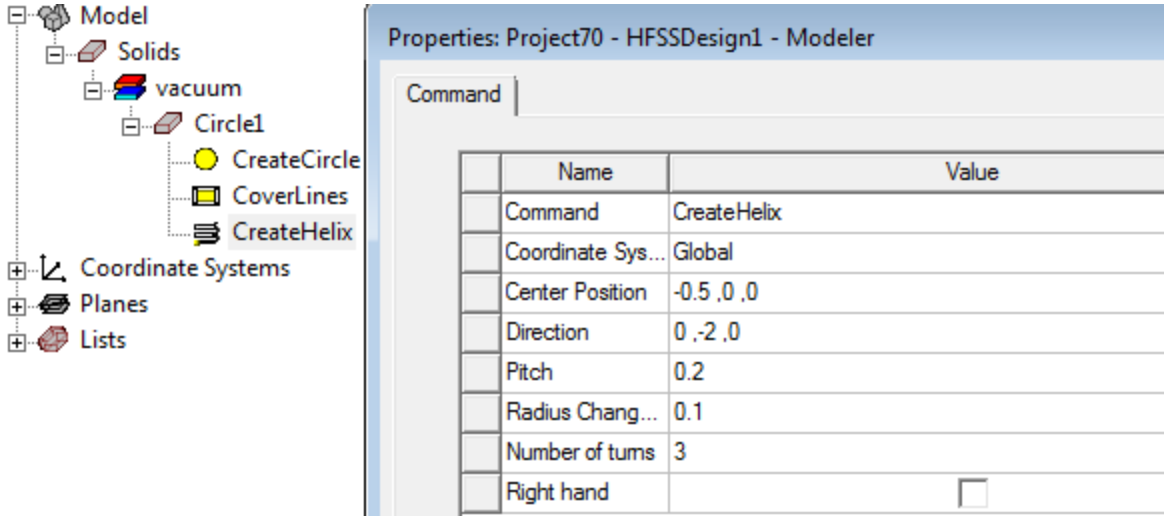


Sweeping a 2D object results in a hollow 3D object. Sweeping a 2D sheet object results in a 3D solid object.



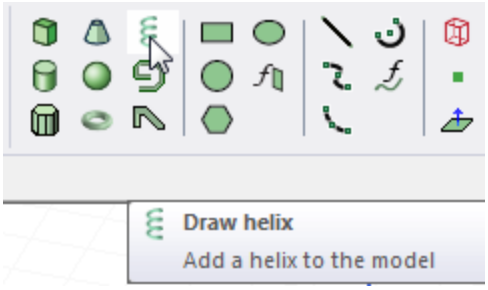
Once you have created a helix, you can changing it by edit the properties of the original object, and by editing the helix command properties for coordinate system, center position (which corresponds to the vector placement), direction, pitch, turns, radius change, and direction.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To create a helix:

1. Create and select the line or 2D object you want to sweep to form a helix. Selecting a valid object enables the **Helix** icon on the **Draw** tab of the ribbon.
2. From the menu bar, click **Draw**> **Helix** or, on the **Draw** ribbon tab, click the **Draw helix** icon:

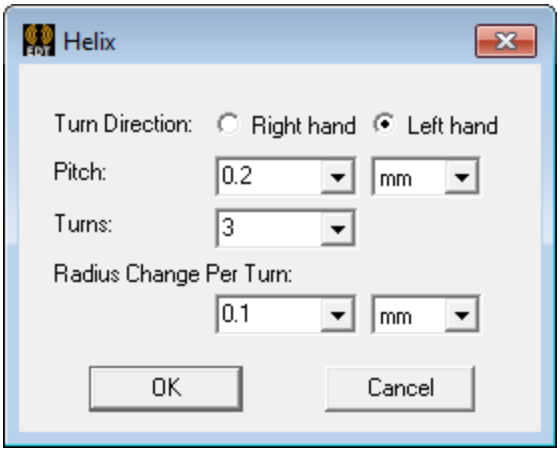


3. Draw the vector you want to sweep the object around. The two points that describe the vector affect axis location and direction only and not the helix length. (The vector definition corresponds to the **Center Position** and **Direction** properties of the **CreateHelix** command in the History tree.) The helix length is determined when you enter the pitch and number of turns in the **Pitch** and **Turns** text boxes. The initial radius of the helix is determined by the axis position relative to the object being swept. It is important that the swept object cannot intersect itself. For a 2D object, you should ensure that the vector you draw does not intersect the interior. If it does, the object cannot be drawn due to self-intersection and you receive a message that "Body could not be created because of invalid parameters."
    a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes. (Remember that you can also edit the parameters of a completed helix).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

The **Helix** dialog box appears:



4. For **Turn Direction**, select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Pitch** text box, type the distance between each turn in the helix, and click a unit in the drop-down menu.
6. In the **Turns** text box, type the number of complete revolutions the object will make along the vector.
7. In the **Radius Change per Turn** text box, type a number for the increase in the radius and select the units from the drop-down menu.
8. After you set these values, the selected object is swept along the vector to form a helix. The original object you swept is deleted.
9. Click **OK** to create the Helix.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also select the commands for the object and the helix command in the History Tree to access and edit their properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Modifying Surface Approximation Settings and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a polygon cross-section. Before you draw a helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **PolygonHelix**.

   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

PolygonSegments	Number of segments in the polygon cross-section. Enter zero (0) for true circle
PolygonRadius	Radius of the polygon cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.
RadiusChange	The radius change per turn of the helix.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

3. Click **OK**.

Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a rectangular cross-section. Before you draw a segmented helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **RectHelix**.

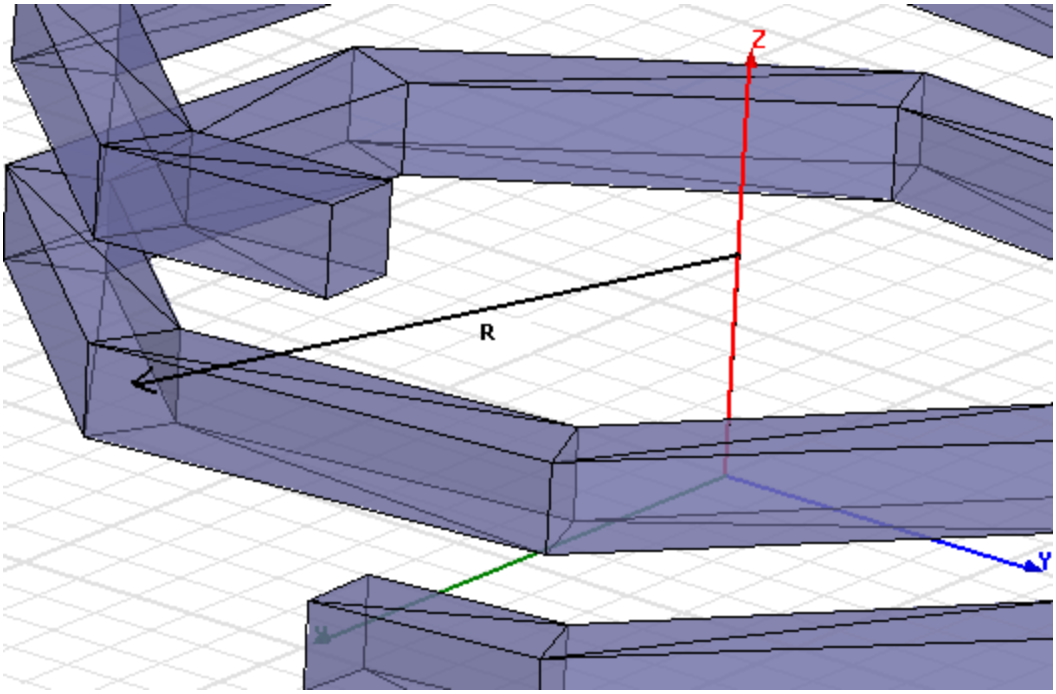
   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

RectHeight	Height of rectangular cross-section.
RectWidth	Width of rectangular cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.
RadiusChange	The radius change per turn of the helix.
Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

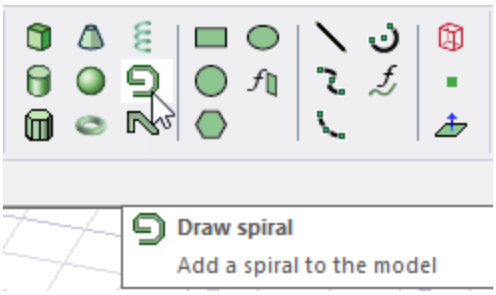
3. Click **OK**.



Drawing a Spiral

A spiral is a 2D or 3D spiral object created by sweeping an object around a vector. Sweeping a 1D object results in a 2D sheet object. Sweeping a 2D sheet object results in a 3D solid object. Before you draw a spiral, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Select the 1D or 2D object you want to sweep to form a spiral.
2. From the menu bar, click **Draw**> **Spiral** or, on the **Draw** ribbon tab, click the **Draw spiral** icon:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Draw the vector you want to sweep the object around:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes.
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

   b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

   The **Spiral** dialog box appears.

4. Select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Radius Change** text box, type the difference in radius between each turn of the spiral.

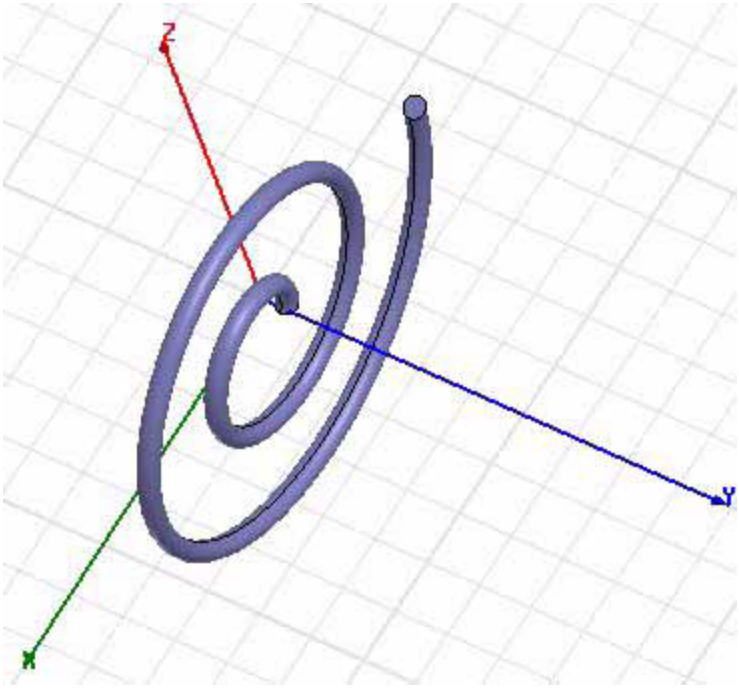
   The radius of the first turn is measured from the center point of the 1D or 2D object you are sweeping to the vector you drew.

6. Click a unit for the radius in the pull-down list.
7. In the **Turns** text box, type the number of complete revolutions the object will make around the vector.

   The selected object is swept around the vector to form a spiral. The original object you swept is deleted. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **OK**.



This 3D spiral was created from a 2D circle drawn at z = 0. The turn direction was right hand, the radius change was set at 2, and the number of turns was set at 2.

> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:
>
> Modifying Surface Approximation Settings and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Spiral Using User Defined Primitives

Ansys provides you with a DLL and a Python script to define the parameters of a rectangular spiral. Before you draw a spiral, you can specify the coordinate system, and you can set the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **User Defined Primitive** > **Examples** > **RectangularSpiral (DLL)** or **Rectangular Spiral (Python)**.

   The **User Defined Primitive Operation** dialog box appears. The **Parameters** tab permits you to see and edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

Xpos	X location of the starting point.
Ypos	Y location of the starting point.
Dist	The separation distance between turns.
Turns	The number of complete revolutions the object will make around the vector
Width	The width of the spiral.
Thickness	The thickness/height of the spiral. If you specify the thickness as zero, the modeler draws a sheetobject.

3. Click **OK**.

   This creates the primitive and displays the **Properties** dialog box for the new object.

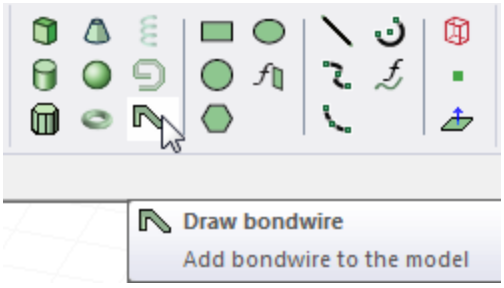
> **Tip:**
>
> To see newly created DLLs, click **Draw** > **User Defined Primitive** > **Update Menu**.
>
> To see the primitives that you have created, click **Draw** > **User Defined Primitive** > **UserLib.**

Drawing a Bondwire

A bondwire is a thin metal wire that connects a metal signal trace with a chip. Please see the topic Bondwires in the *HFSS: Technical Notes* before drawing a bondwire.

1. From the menu bar, click **Draw**> **Bondwire** or, on the **Draw** ribbon tab, click the **Draw bondwire** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the bond pad point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.
3. Select the lead point by clicking the point or typing the coordinates in the text boxes in the status bar.

   The **Bondwires** dialog box appears.
4. In the **Type** list, click the modeling standard shape you want the bondwire to have: **JEDEC 4-point**, **JEDEC 5-point, or Low**.

   The **Type** selection changes the dialog bondwire graphic, and shows options for that type.
5. Enter the number of facets in the bondwire in the **No. of Facets** text box.

   The minimum value is 3. The value describes the number of faces that make up the circumference of the bondwire.
6. In the diameter field, specify a diameter value and select the units from the drop-down menu.
7. Enter the height between the bond pad and the top of the loop in the **h1** text box. Include the height's unit of length.
8. The value in the **h2** text box is the height between the bond pad and the lead point. It was calculated by Ansys Electronics Desktop based on the lead point you selected. If you modify the value of **h2**, the lead point will be modified.

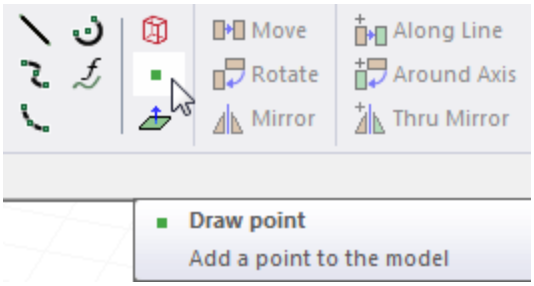
   Optionally, type a new value in the **h2** text box. Include the height's unit of length.
9. If you selected **JEDEC 5-point** or **Low**, do the following:
   a. Type the angle between the horizontal plane and the wire at the bond pad point in the **alpha** text box.
   b. Type the angle between the horizontal plane and the wire at the lead point in the **beta** text box.
10. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Point

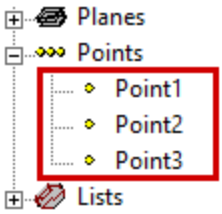
Drawing a point object within the problem region enables you to plot fields or perform field computations at that point. Points are always considered non-model objects by the modeler. Before you draw a point, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Point** or, on the **Draw** ribbon tab, click the **Draw point** icon:



2. Select the point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   The point is listed under **Points** in the History Tree:



> **Note:**
>
> There is no visible indication of the point in the drawing area unless it is selected in the History Tree. Also, the point cannot be selected graphically once added to the model. It must be selected from the History Tree or from the **Geometry** window within the **Fields Calculator**.

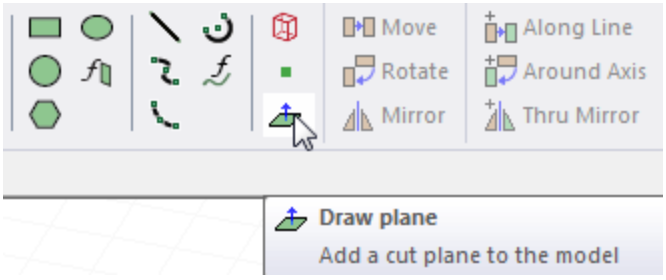
Drawing a Plane

A plane object is a cut plane through the problem region. You can use field overlays (see Modal or Thermal examples) to perform field computations on its surface. Planes are always considered non-model objects by the modeler. Before you draw a plane, you can specify the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Plane** or, on the **Draw** ribbon tab, click the **Draw plane** icon:

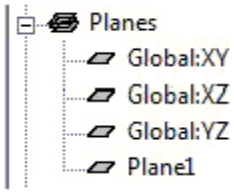


2. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   To delete the selected point and start over, press **Esc**.

3. Select a normal point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the origin in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The plane is created. Its center point is located at the origin you specified and oriented perpendicular to the normal point you specified. The new plane is listed under **Planes** in the History Tree.

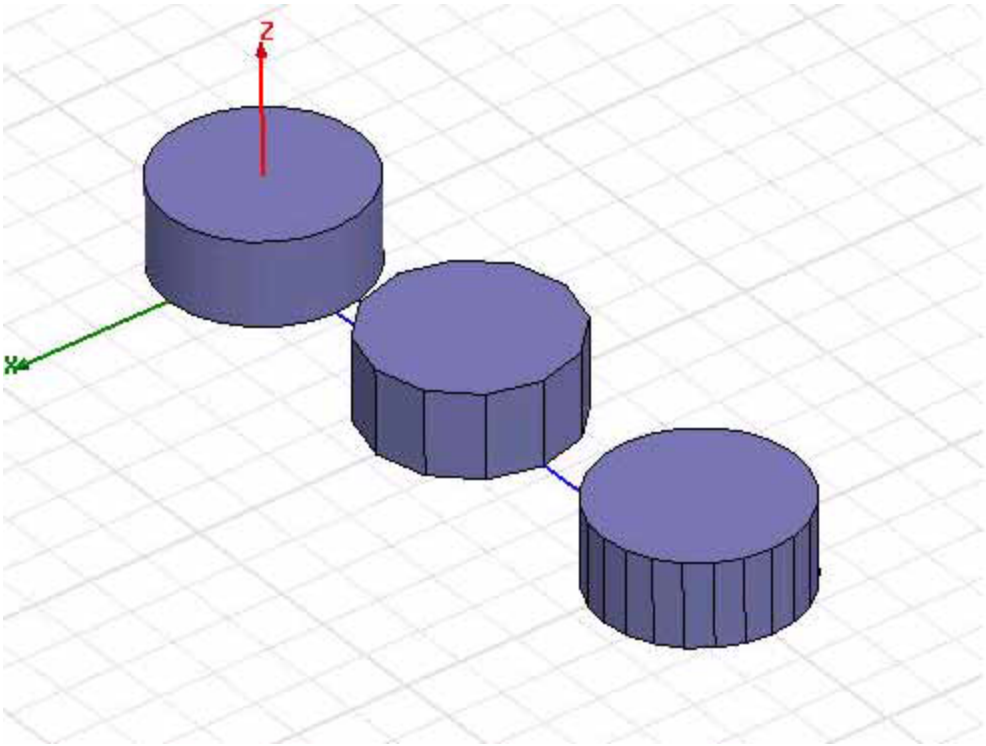


> **Note:**
>
> You only need to draw a plane that does not lie on a predefined xy, yz, and xz plane. Default planes are created on the xy, yz, and xz planes of the global coordinate system as well as any new coordinate system you create.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Segmented Geometry

For some structures, you may want to create segmented as opposed to smooth (or True) surfaces. The following figure shows a comparison of a cylinder created with true surfaces and with segmented surfaces.



The following model objects can be created as segmented structures:

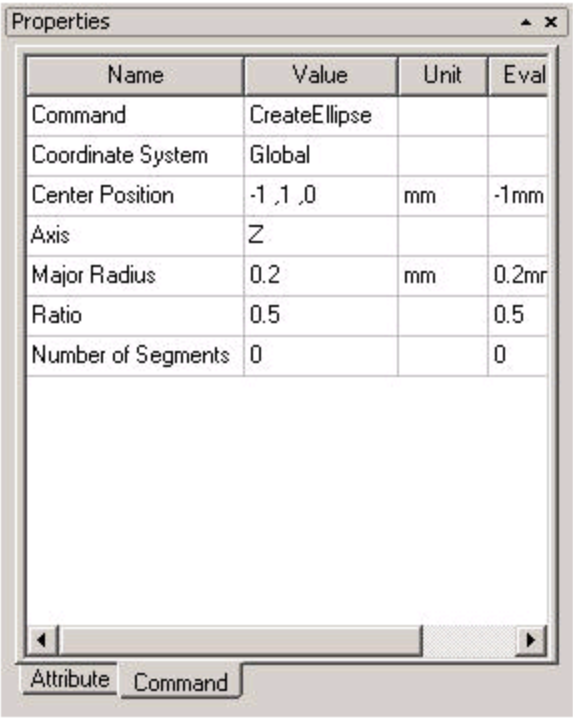
Circle, Ellipse, Cylinder	See: Segmented Objects
Polyline, Arc, Line Segment	See: Converting Polyline Segments

Segmented Objects

To create segmented circles, ellipses, and cylinders use the **Number of Segments** parameter on the Command Tab of the **Properties** dialog as shown below. To convert an object from true surface to segmented, do the following:

1. Select the **circle**, **ellipse**, or **cylinder** in the modeler window or in the History Tree.
2. In the command tab of the properties window (shown docked below), change the **Number of Segments** to an integer value of three or greater and press **Enter**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Values of 1 and 2 are not valid values for the circle, ellipse, or cylinder command and will cause an error.

Drawing Non-Model Objects

If you want to create an object that does not affect the geometric model, define the object as *non-model*. This ensures that the object is used for analysis only; it will not affect the solution process. After drawing the non-model object, assuming it lies in the problem region, you can use it in the reporter as a place on which to plot field quantities. For example, draw a non-model line across the design, then (in the reporter) plot Mag_E on every point along that line.

You can assign output variables (default) and design variable as property values for non-model objects.

Following are examples of using non-model objects to analyze a solution:

- Draw a polyline along which to plot fields or perform field computations. Note that when you create a value versus distance plot, by default, the line will be divided into 100 equally spaced points. You can modify the number of points into which the line is divided in the **Edit Sweeps** dialog box. For more information, see: Specifying Variable Values for a Sweep Definition.
- Draw a rectangle upon which to plot fields in the reporter or perform field computations.
- Draw a volume box to analyze fields in areas of the problem region that are not occupied by an object or that consist of parts of several objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw a plane, which is always a non-model object.
- Draw a point, which is always a non-model object, in order to plot fields in the Reporter or perform field computations at that point.

What do you want to do?

Switch to non-model drawing mode. Objects you draw in non-model mode will not be included in the solution process.

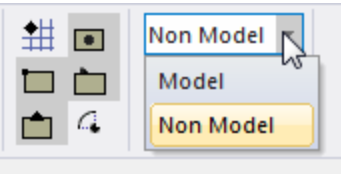
Modify an existing model object to be a non-model object.

Selecting Non-Model Drawing Mode

To switch to non-model drawing mode:

1. Click **Modeler**> **New Object Type**> **Non Model**.

   Alternatively, on the **Draw** ribbon tab, choose **Non Model** from the **Set to model non model type** drop-down menu:

2. Draw the object.

Changing an Object to Non Model

To modify an existing object to be a non-model object:

1. Select the object you want to modify.
2. In the docked **Properties** window, clear the **Model** option.

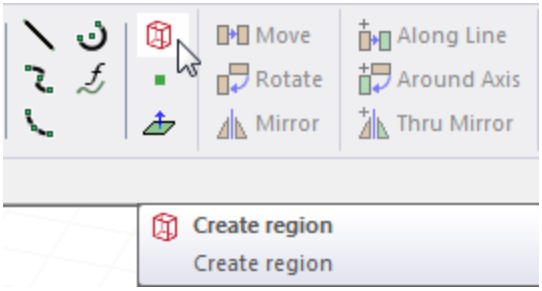
   The object will not be included in the solution process. If the object lies in the problem region, you can plot solution quantities on it.

Drawing a Region

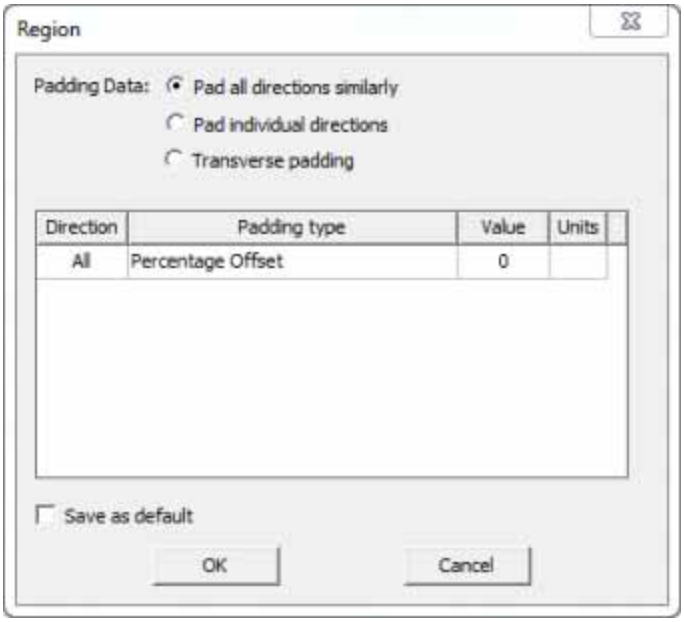
To draw a region encompassing the objects in the current project:

1. From the menu bar, click **Draw** > **Region** or, on the **Draw** ribbon tab, click the **Create region** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

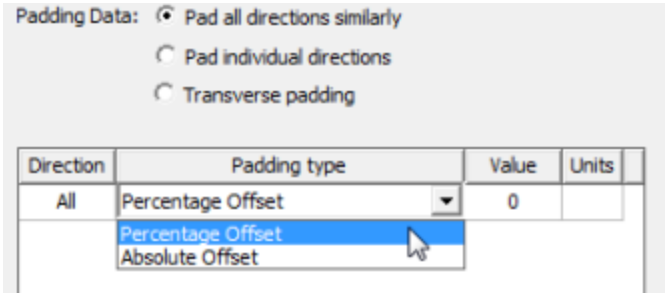


This displays the **Region** dialog box.



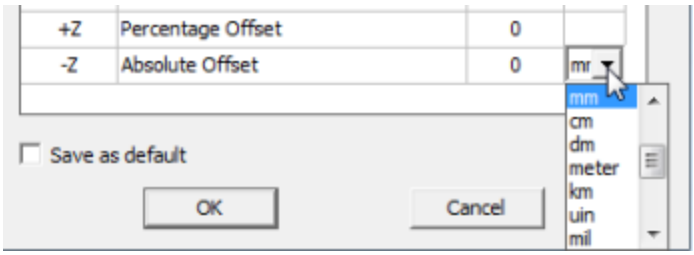
2. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

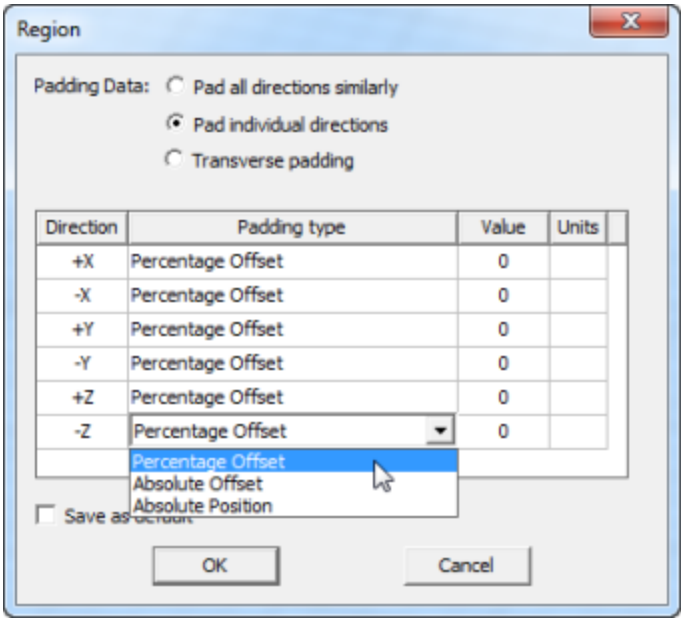


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.



Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.



Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

3. Specify the Padding values in the fields and select the units from the drop-down menu.
4. If desired, select the check box to save the values as Default.
5. Click **OK** to close the dialog and create the region.

The region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system. The Properties dialog for the Create Region icon in the History tree shows

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the coordinate system and Padding values. The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

If you try to create a region that does not contain all of the objects in your model, the modeler automatically expands the region to cover all objects. The region also updates automatically as your geometry changes.

Only one region can be created for a single project using the **Draw** > **Region** command. If you try to create a second region, the **Properties** window appears for the existing region, allowing you to change operation parameters and attributes.

Related Topics

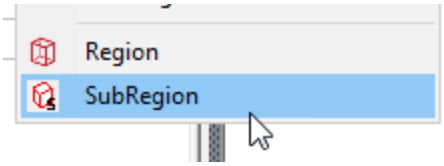
Drawing a Sub-Region

Drawing a SubRegion

The Modeler allows you to define a subregion that encompasses a set of objects inside previously defined region. Like regions, subregions automatically resize if the objects contained in these subregions change. You can create nested subregions, and reassign the content of a subregion.

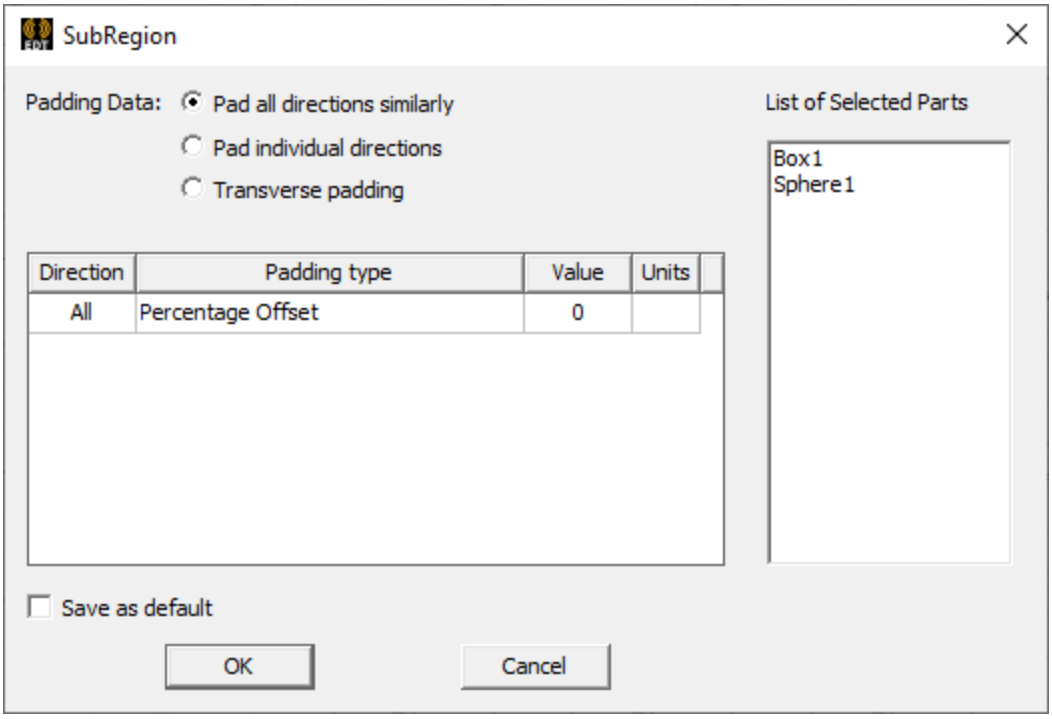
To draw a sub-region in the current project:

1. Select the parts of components that you want to include in a subregion.
2. From the menu bar, click **Draw** > **SubRegion**:



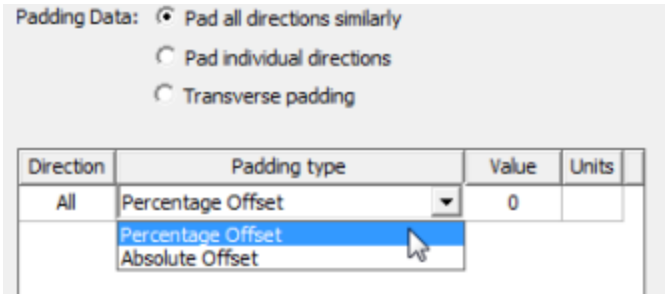
This displays the **SubRegion** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



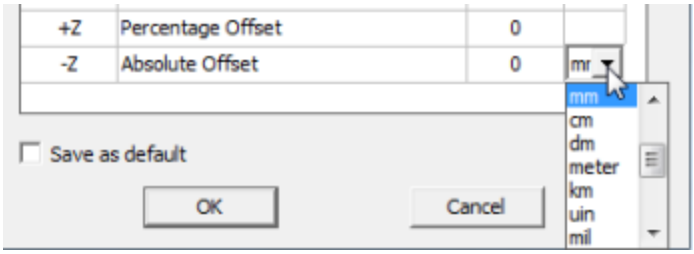
3. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

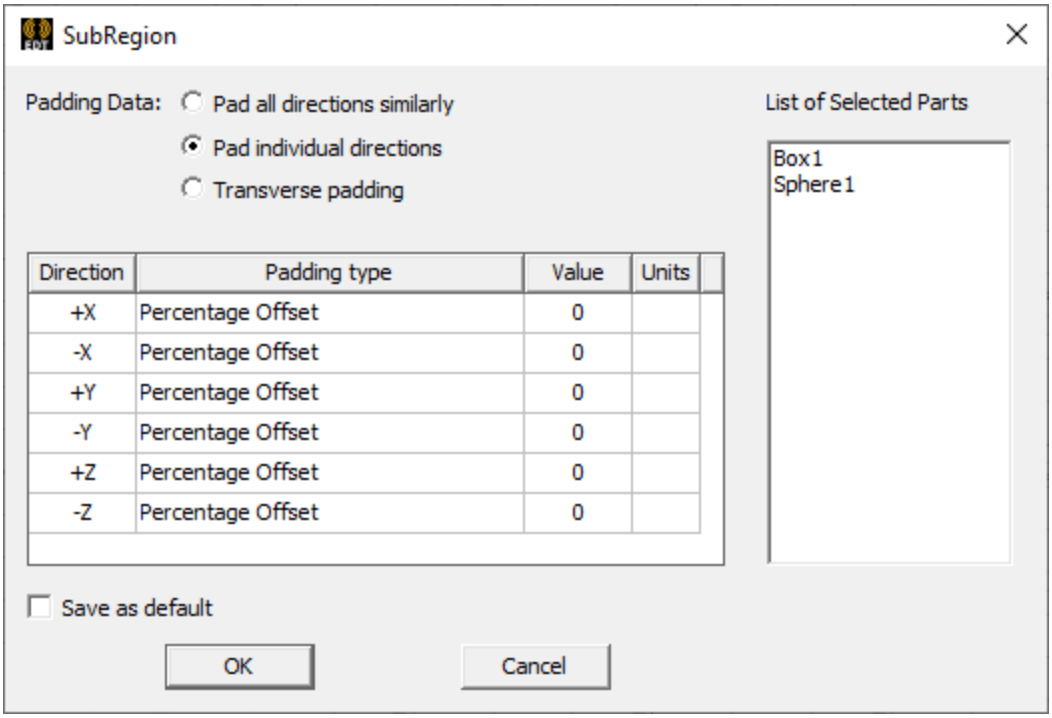


   If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

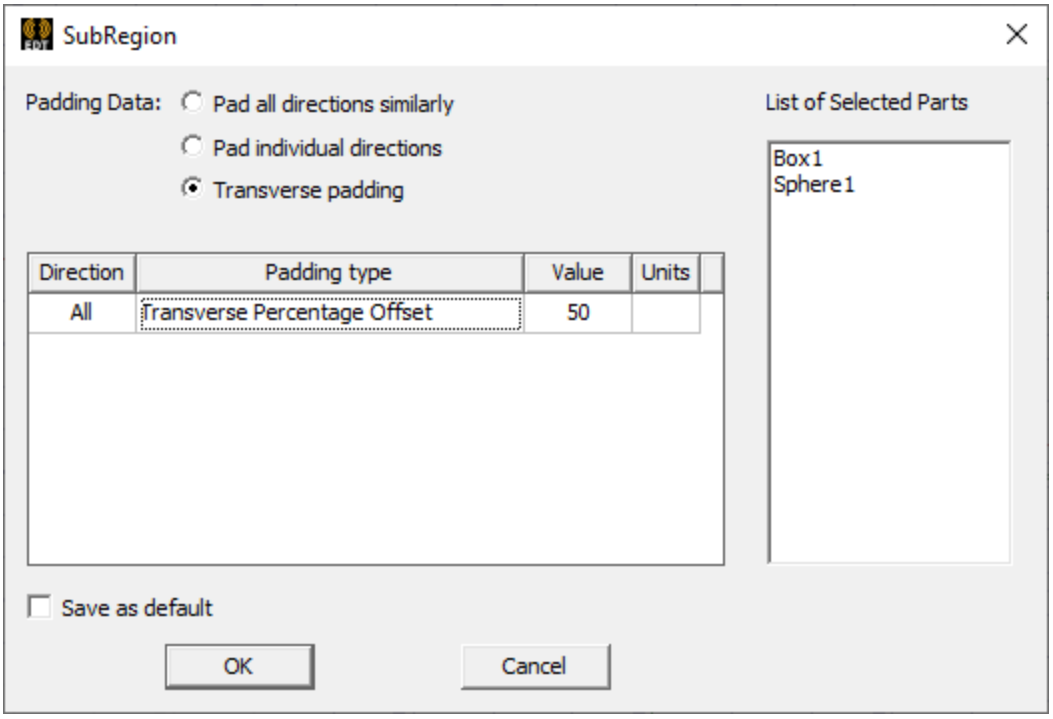


Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.



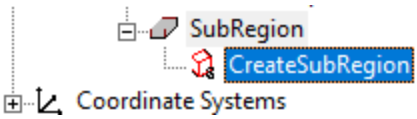
Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Specify the Padding values in the fields and select the units from the drop-down menu.
5. If desired, select the check box to save the values as Default.
6. Click **OK** to close the dialog and create the sub-region.

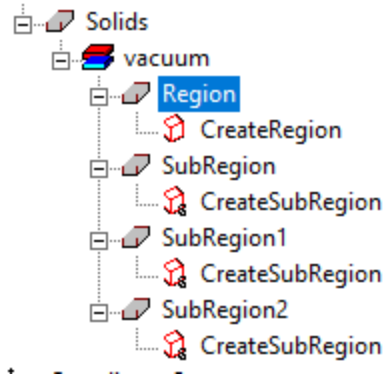
The sub-region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system.



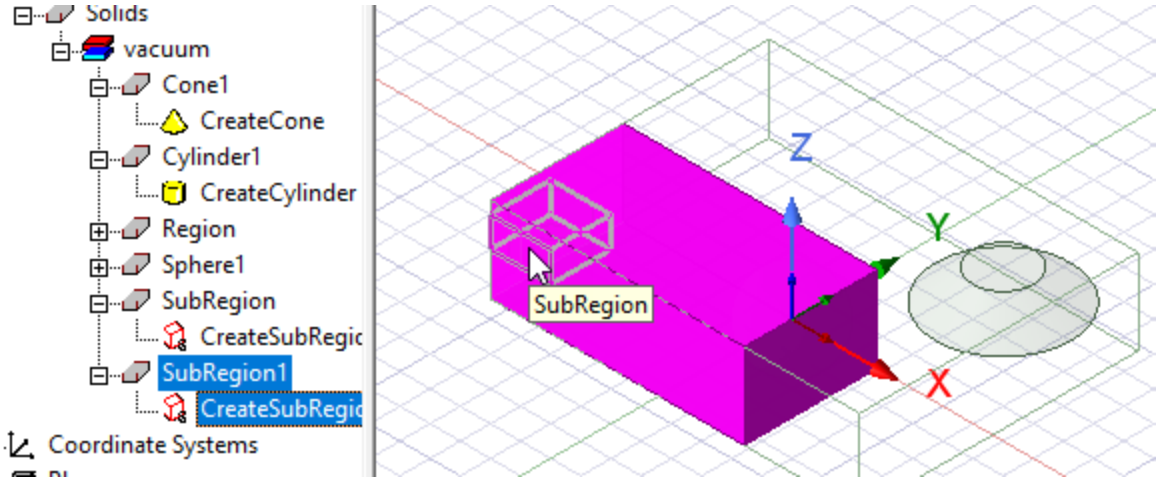
Additional SubRegions and Nested SubRegions.

If you create additional sub-regions, these appear in the History Tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



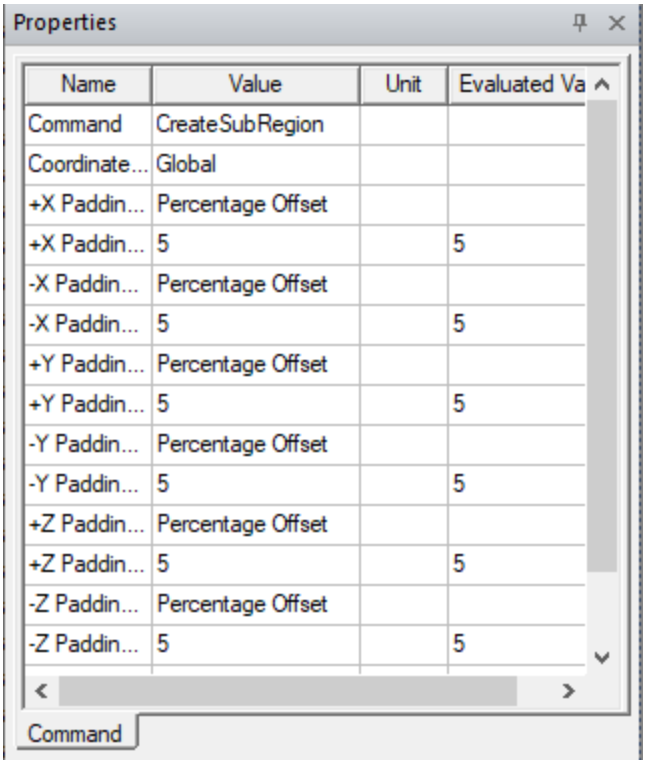
SubRegions can contain other SubRegions, that is, SubRegions can be nested.



Properties for SubRegions

The Properties dialog for the Create SubRegion icon in the History tree shows the coordinate system and Padding values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

You can use modeling operations on these sub region parts with following rules

- Geometry of non-subregion parts should not depend on sub-region parts. That is, sub-region parts should not be allowed to be used as a tool, not even after cloning them
- Simple local operations like face move, fillet, and arrange operations should be allowed on sub-region parts.

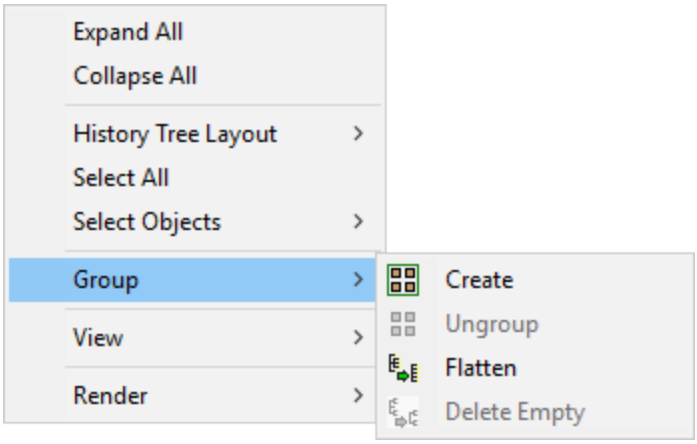
Global region should take the sub-regions into account in its computation.

Sub-Regions can be included in components.

Group Commands for Modeler Objects

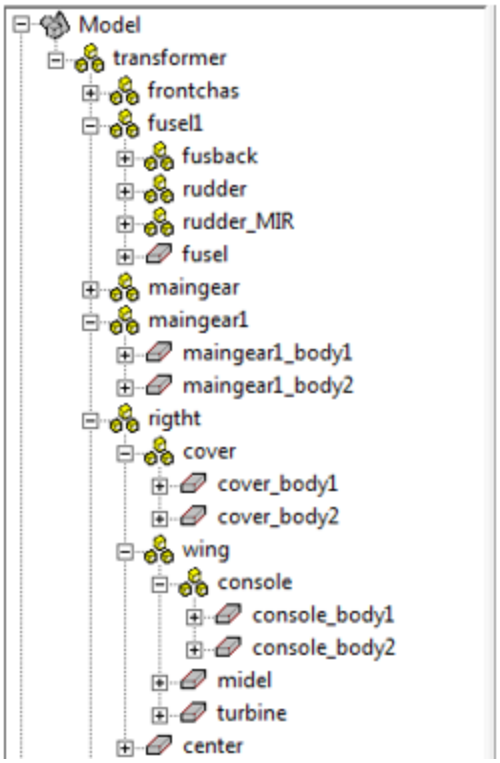
The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Group** commands support the following features:

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-group containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.
- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Accessing Group Commands

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout>Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Operations on Groups

Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Copy and Paste of Objects in Groups

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

You can select group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

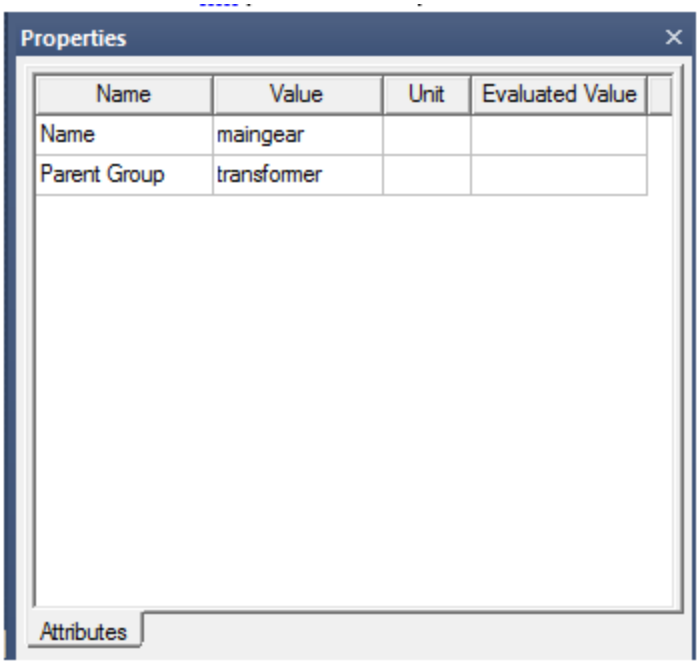
Other Operations on Groups and Objects in Groups

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See: 3D Modeler Options: Group Options.

Group Properties

A Group's Property tab is shown when you select a group in the history tree. The Group's properties include Name and parent Group.

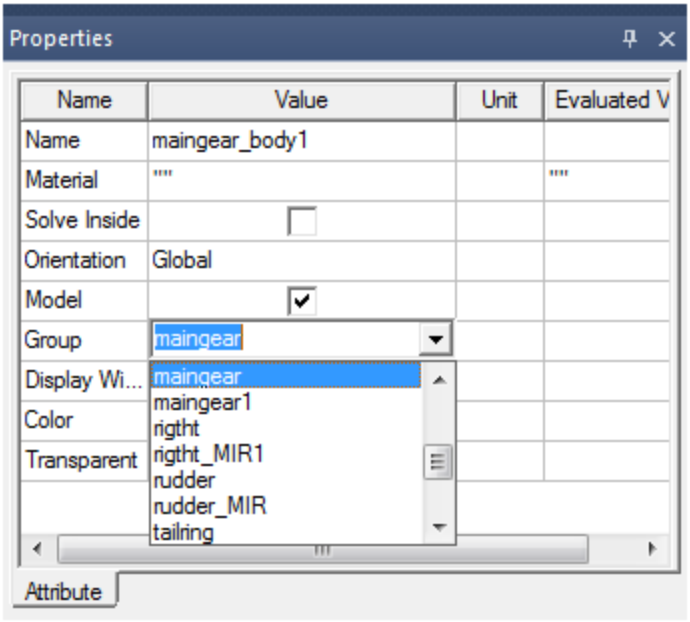


Group Property Window

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

existing groups listed or create a new group.



Group assignment could be changed in Object's property window

Model Analysis

For some models it may be beneficial to remove unnecessary small entities and to fix object misalignments to avoid potential mesh issues. The Modeler window includes Model Analysis functions to help you evaluate models you have imported or created. Select **Modeler** > **Model Analysis** to see the menu options. Depending on the design and the current selection, some features may not be enabled. The menu includes the following commands.

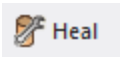
- **Analyze Objects**
- **Analyze InterObject Misalignment**
- **Analyze Surface Mesh**
- **Show Analysis dialog**

> **Note:**
>
> Before running model analysis, you must remove all command history for the selected object by using the Purge History command. If you need to save the object history, save a separate copy.

After import, you typically perform validation check. This lets you focus on objects and object pairs that have errors and or warnings. The objects that fail should be analyzed by using the **Modeler** > **Model Analysis** > **Analyze Objects** menu item.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the objects and invoke **Modeler** > **Model Analysis** > **Analyze Objects**; or, on the **Model** ribbon, click **Heal**.

   This displays the **Analysis Options** dialog box, which allows you to specify settings for entity check level and small feature detection.

   When you **OK** this dialog box, the initial analysis executes and the **Model Analysis** dialog box is displayed.

2. Choose the objects that have "Invalid Entities Found" and **Perform** > **Heal Objects**.

   In most cases, the objects will be healed and the errors fixed.

3. If errors still persist, choose the edges and faces and click on **Delete**.

This will replace the selected face/edge object by a tolerant edge/vertex respectively. In some cases the replacement of the face/edge by tolerant edge/vertex will fail.

When models pass the initial validity checks, mesh generation could still fail. The following errors can be present in models: (See Error Detection.)

- Non-manifold topology. These are non-manifold edges and vertices that are present in the model.
- Object pair intersection. This detects whether pairs of objects intersect.
- Small feature detection - small edge length, small face area and sliver face detection.
- Misaligned entities detection - detects pairs of faces from objects that can be aligned to remove object intersections. This improves the probability of mesh success.
- Mesh failure error display. This is available for single object, object pairs and last simulation run (all objects in a model). Errors reported by the meshing module are reported to the user.

Small feature errors and misaligned entities should be resolved before you invoke the meshing for the model.

By default, the **Heal** command is automatically applied to imported objects.

Analysis Options Dialog Box

To perform analysis on an object according to specified features and tolerance values:

1. Select the object you want to analyze and click **Modeler** > **Model Analysis** > **Analyze Objects**.

   This displays the **Analysis Options** dialog box, with the **Analysis Options** tab selected. Selecting **Modeler** > **Model Analysis** > **Heal** also displays this dialog box. If, during

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeler > **Import...** you select **Heal Imported Objects** and **Manual**, you also see this dialog box.

2. If desired, check the **Perform Entity Check Errors** check box.

   This enables the **Check Level** menu. The setting can be **Basic**, **Strict**, or **Comprehensive**. See: Modeler Validation Settings.

3. If desired, click the check boxes to enable and set the **Detect Feature** settings:
   - **Detect Holes**, and specify the Maximum Radius.
   - **Detect Chamfers**, and specify the Maximum Width.
   - **Detect Blends**, and specify the Maximum Radius.
4. If desired set the **Detect Small Entities** features and tolerance values.
   - **Small Edges**, length less than
   - **Small Faces**, area less than
   - **Sliver Faces**, which enables:
     - Object Bounding Box Scale Factor
     - Sliver Edge Width
5. Click the **Properties** tab to see a listing of the geometric properties of the selected object.
6. Clicking **OK** on this dialog displays the **Model Analysis** dialog box, which contains the results of the analysis.

Analyzing the Surface Mesh

To set the options to analyze the surface mesh:

1. Select an object of interest.

   This enables the Analyze Surface Mesh command in the menu.

2. Click **Modeler** > **Model Analysis** > **Analyze Surface Mesh**.
   The **Surface Mesh Analysis Options** dialog box appears. This dialog box allows you to set parameters to remove.

   You can also open the **Surface Mesh Analysis Options** dialog box from the **Model Analysis** dialog box via the **Perform** drop-down menu on the **Objects** tab.

3. Select or clear the **Perform Object Pairs Analysis** check box. Selecting this option evaluates the mesh for all combinations of the selected objects.
4. Select or clear the **Ignore Objects Separated by greater than** check box, and enter a value in the text box. Selecting this option means that object pairs are disregarded from analysis if their separation is greater than the specified value.
5. Click **OK** to perform the analysis with the selected options.
   The **Model Analysis** dialog box appears, displaying the results of the analysis.

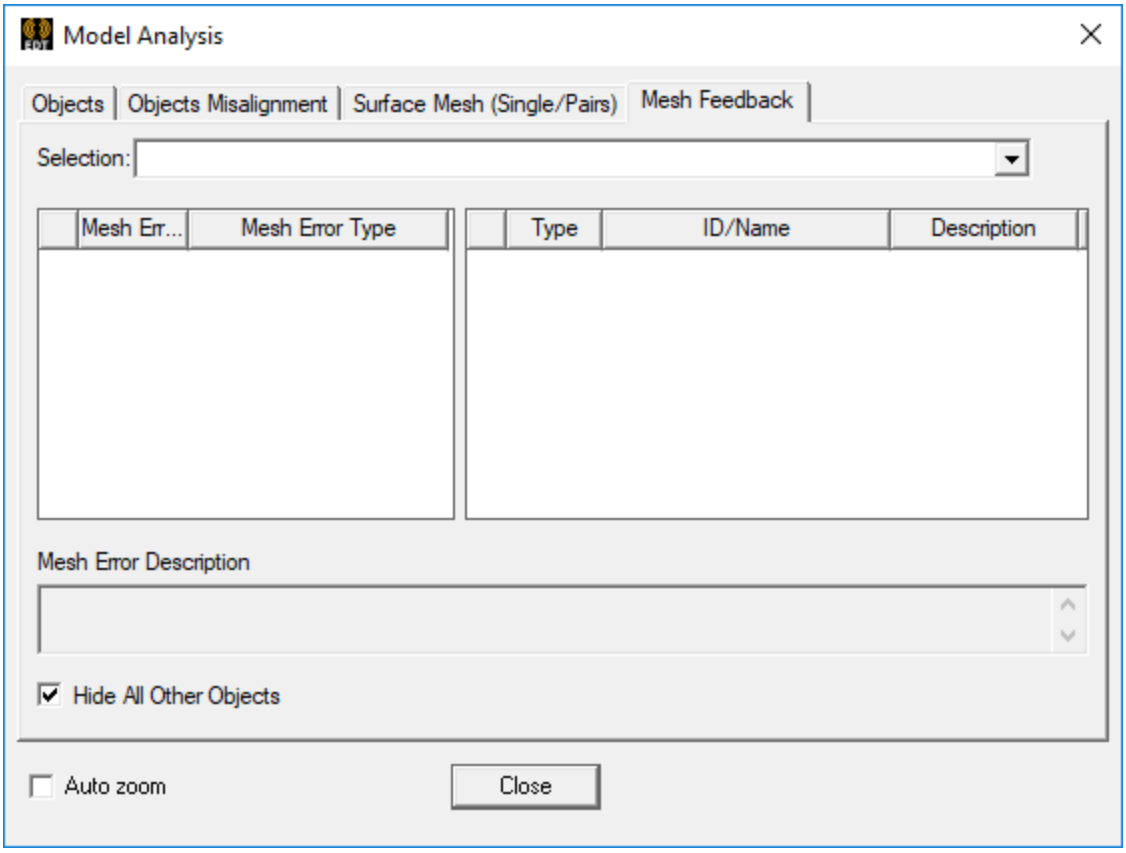
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Analysis Dialog Box

This dialog box contains results for all model analysis, including diagnostic information relating to mesh issues. To view the analysis options:

1. Click **Modeler** > **Model Analysis** > **Show Analysis Dialog**.
   A submenu appears.
2. Select one of the following from the submenu:
   - **Objects**
   - **Objects Misalignment**
   - **Surface Mesh**
   - **View Mesh Feedback**

   The **Model Analysis** dialog box appears (This dialog box also appears automatically after clicking **OK** in the **Analysis Options** dialog box).



3. You can select **Hide All Other Objects** to suppress display of objects, faces, or surfaces in the Modeler window not selected in the **Mesh Feedback** panel

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Auto zoom to selection** check box to automatically zoom to the item selected on the **Objects** tab.
5. Make the desired changes on each tab in the **Model Analysis** dialog box.
   - **Objects** tab
   - **Objects Misalignment** tab
   - **Surface Mesh (Single/Pairs)** tab
   - **Mesh Feedback** tab
6. Click **Close** to close the **Model Analysis** dialog box.

Objects Tab

All results relating to model analysis of specific objects are presented under the **Objects** tab.

1. The results table contains the following information.
   - **Object** – column listing the objects in the current design.
   - **Last Analysis status** – column giving the analysis status of the listed objects. Objects can have the following status:
     - **Good** – the object contains no invalid geometry entities given the tolerance values specified in the **Analysis Options** dialog box.
     - **Null Body** – the object is non-existent.
     - **Analysis not performed** – the object was not selected for analysis.
     - **Invalid entity errors** – these are api_check_entity() errors and non-manifold errors which Ansys EM recommends that you fix before meshing.
     - **Small entity errors** – small faces, sliver faces and small edges that are optionally detected based on the tolerance limits specified in the Analysis Options dialog box.
2. Select any object name in the table which contains errors to display a table with a list of Error types, corresponding Entity List IDS for associated faces, edges and vertices, and error descriptions. Selecting the row for each error causes the Modeler window to highlight the associated entity.

   > **Note:**
   >
   > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the **Model Analysis** dialog box.

3. Select the face, edge or vertex entity from the list to view the error description in the **Description** field.
4. Select the **Delete** button if you want to remove a selected face or edge entity.
5. Select the **Perform** button to list the commands that you can execute on the selected objects in the Results table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Heal Objects** – repairs invalid geometry entities for the selected objects within the specified tolerance settings. The **Healing Analysis** dialog box will appear.
- **Analyze Objects** – evaluates the object status. Selecting this displays the Analysis Options dialog.
- **Analyze Surface Mesh** – invokes a mesh for each selected object and reports analysis results under the **Surface Mesh (Single/Pairs)** tab. Selecting this option displays a dialog with radio buttons to select.
  - **Perform Object Pairs Analysis** – evaluates mesh for all combinations of the selected objects.
  - **Ignore objects separated by greater than a specified value** – object pairs are disregarded from analysis if their separation is greater than the specified value.
  - Click **OK** to perform the analysis with the selected options.
- **Analyze Interobject Misalignment** – determines any misalignments between two selected objects in the results table. The results are reported under the **Objects Misalignment** tab.
- **Display Healing Log** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object during the healing process.

Object Misalignment Tab

The table in this panel displays results of an **Interobject Misalignment** analysis. It contains a list of Alignable Faces, described in a list of Object Sets, and corresponding Misaligned Faces.

All misaligned face pairs corresponding to the analyzed objects are listed in the table.

- **Align Faces** – select a face pair in the table and click **Align Faces** to align selected faces.
- **Clear All Analysis Data** – this button removes all information from the tables.
- **Display Log for Object Set** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object set during the healing process.
- **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the table.

After validation check is performed, the pairs of objects that intersect are chosen for analysis. Use the analysis results to find whether objects have faces that can be aligned.

Choose all the bodies that intersect with another body.

1. From the **Model Analysis** dialog box choose perform/Analyze Interobject misalignment. Or you can run **Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the analysis finds object pairs that can be aligned, they will be displayed in the **Objects Misalignment** tab.

2. You can select individual or multiple rows and perform Align Faces. In some cases, face alignment will fail if the topology of the body changes by a large factor after alignment.
3. Identify individual bodies and body pairs that fail to mesh.
4. Perform Mesh analysis on individual objects and object pairs.
5. Review the reports and fix the errors.

Surface Mesh (Single/Pairs) Tab

The panel displays the results of a surface mesh analysis.

1. You can display results for:
   - Individual Objects
   - Object Pairs

   > **Note:**
   >
   > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the object or object pair selected.

2. The results table contains the following information:
   - **Object** – column listing object name or a pair of object names.
   - **Last Analysis Status** – column stating the meshing status of the object or object pair.
     - Mesh Success
     - Mesh Failure
   - **Error Type** – this column gives the category of error that caused the mesh failure.
   - **Error Detail** – provide specific geometry information regarding mesh error location.

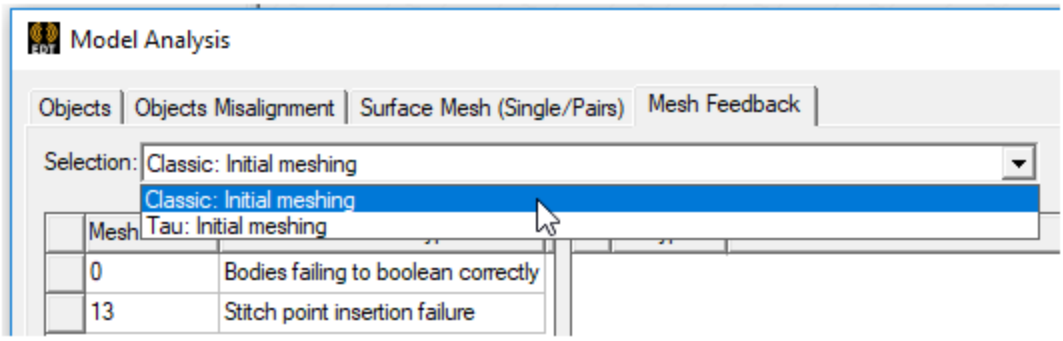
   Display options include:

   - **Display Mesh Analysis** – checking this displays further details concerning each error to be listed.
   - **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the error.

Mesh Feedback Tab

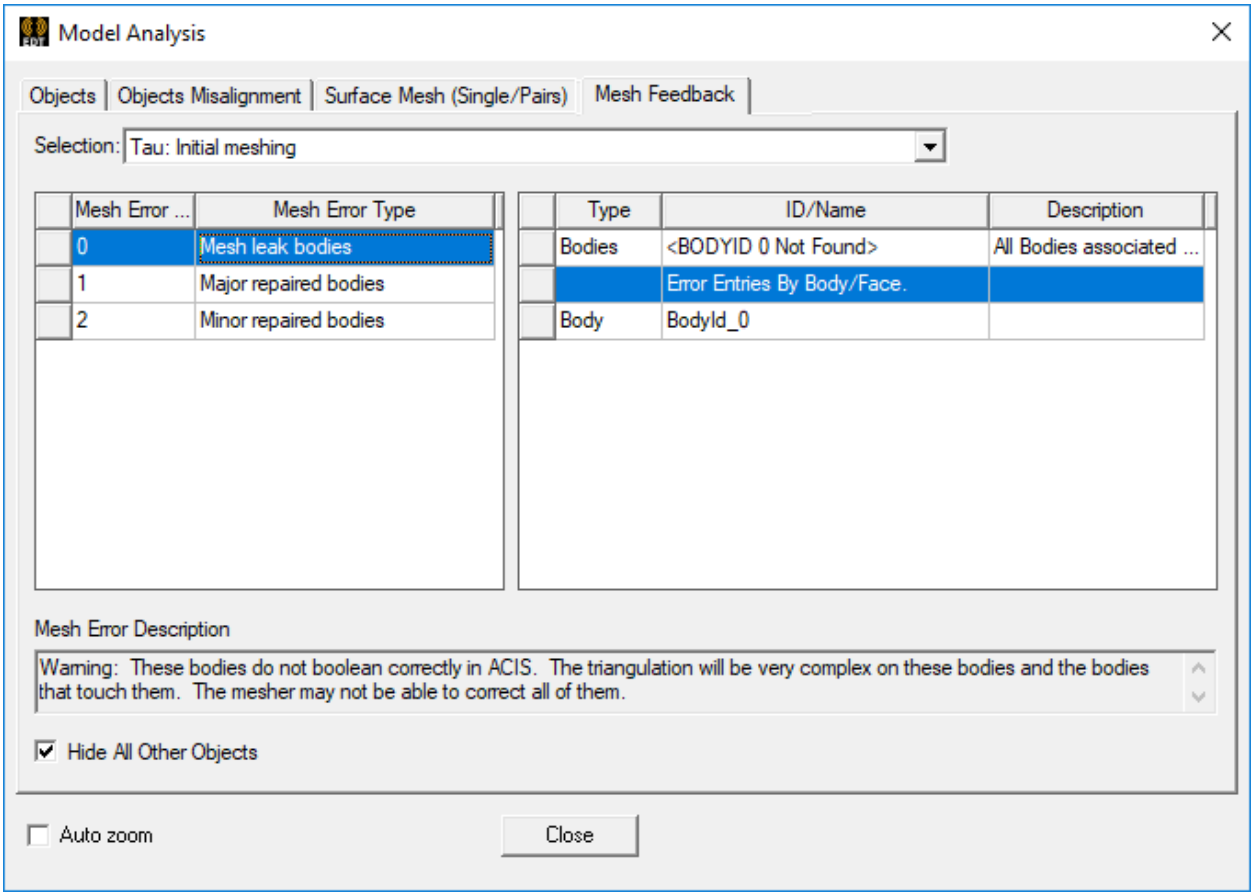
The tables in this panel list model errors as viewed by the mesher. If the project has solutions by different solvers, the **Selection** drop down lets you select which to view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left side table has columns for Mesh Error Serial ID and Error Type.

- **Mesh Error Serial ID** – the 0 item is the body with which the subsequently listed entries are associated.
- **Mesh Error Type** – this column gives the category of error that caused the mesh failure, for example, Non-Manifold Edge, or Point Insertion Failure.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The display in the table on the right side of the **Model Analysis** window depends on your selections from the left hand table. When you select a row in errors panel (the left panel as shown in above figure), all the faces are selected (second row in right panel) and if you enable **Auto Zoom**, the view fits to the errors:

- **Type** – whether the error applies to Bodies, Faces, or Triangles. If you select a particular error type, the columns for Type, ID/Name and Description are filled in.
- **ID/Name** – the object name or object ID for the error type.
- **Description** – the first three rows are for Bodies, Faces, and mesh Triangles. Subsequent rows, are of Error Entities By Body.Face.

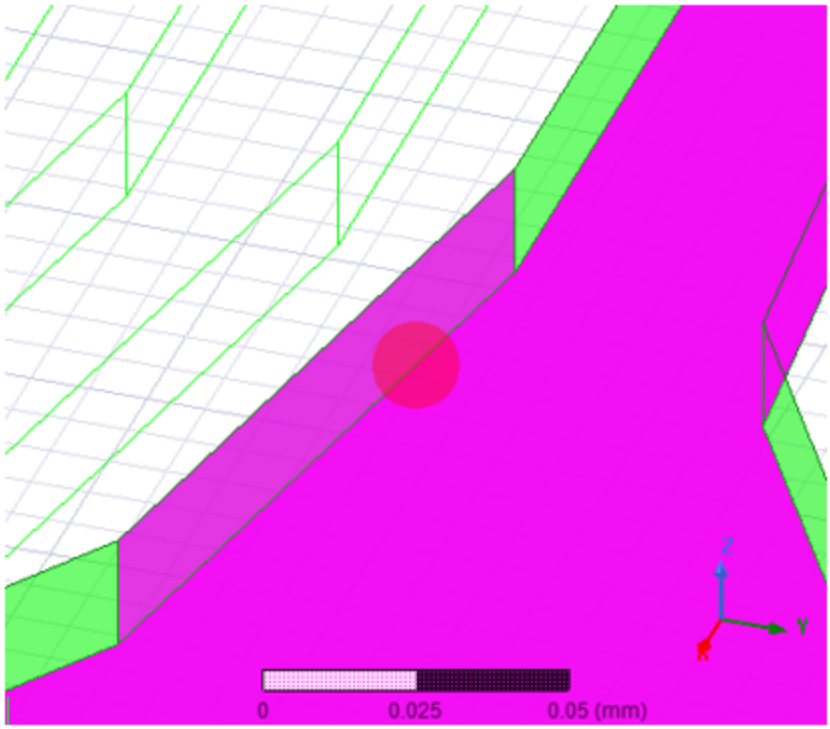
The **Mesh Error Description** field describes error message describing the nature of the selected Mesh Error Serial ID, the implications, and provides a recommended response.

Display options include:

- **Hide All Other Objects** – hides objects in the Modeler window not associated with the selected Bodies, Faces, or Triangles.
- **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the selected Mesh Error Serial ID (on the left side table) or the ID/Name (on the right side table).

Use of these selections let you more easily view and respond to the errors. If you enable **Auto Zoom**, any selection whether it is body(ies), face(s) or error triangle(s) or error segment(s) will fit to view. If all the faces are also selected (second row in right panel), then errors and selected entities will fit to view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The error shown in the figure above is a contact error between the faces. Focusing only on the errors when an error is selected in right panel, improves visualization of these meshing errors and provides you with precise information in order to take corrective action quickly.

Mesh Error Markers

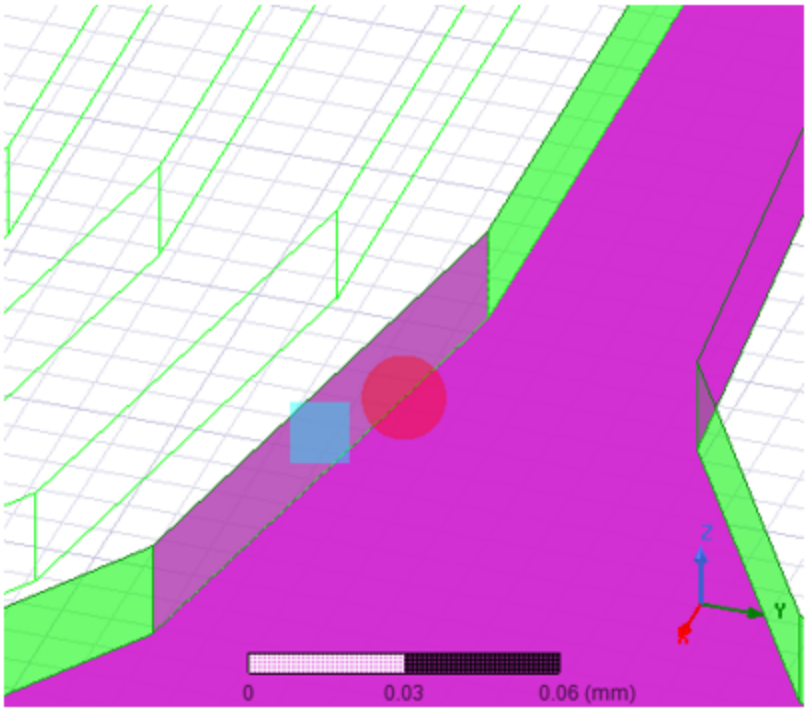
As shown above the mesh errors are marked by ‘Red Circles’. The following table describes all such markers:

	Mesh errors
	Mesh warnings
	Mesher information
	Selected Faces

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These markers are located at the center of errors or center of faces and their size correspond to the maximum side size of the entity, such as error triangles, error segments or faces. The size of these markers is clamped to a maximum and a minimum size. Additionally they have some transparency. The size and transparency of these markers cannot be changed by the user.

If you select an individual face in the right panel, then the face is marked using a blue square as shown below:



If many individual faces are selected in the right panel, each face is marked by a blue square. In order to find which blue square corresponds to which face in the right panel in the dialog box, click on the blue square. This action selects the corresponding face row in the right panel of the dialog.

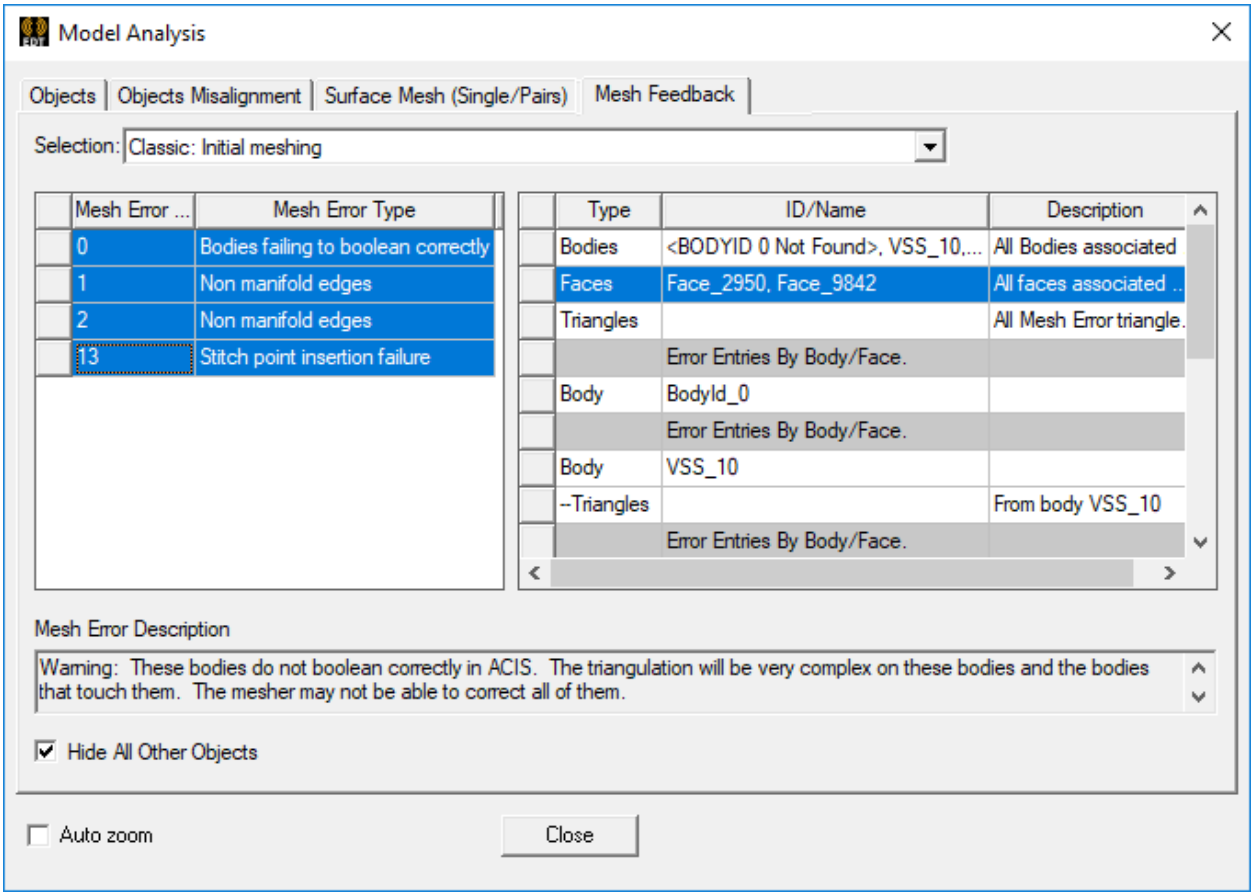
Face Selection

When the all faces row or individual face rows are selected, the Modeler selects the face(s). This enables the face related commands in the Modeler window. Also when you select faces this way, it makes the corresponding bodies of the selected faces visible, even when ‘Hide all other objects’ is enabled. This provides some contextual information about the face location with respect to its body.

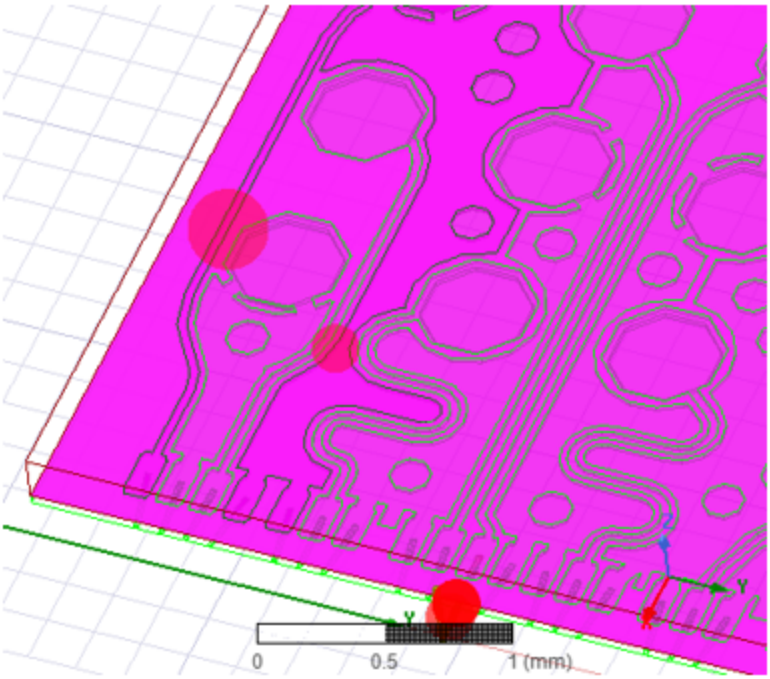
Multiple Selection of Mesh Errors

You can select multiple rows in the right pane of the **Mesh Feedback** tab. To view all the meshing errors, select all the rows and enable **Auto Zoom**, as shown in figures below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Auto Zoom** enabled, the view fits to all the errors, providing an overview of all the mesh errors, as shown in above figure.

Also, when multiple errors are selected, right panel shows the merged information about all bodies, faces, error triangles etc. And Mesh Error Description text box shows description of all the errors.

Design Settings

The tabs and settings within the *Design Settings* dialog box are dependent on the design type, and there is a product-specific topic for each applicable design type. Please click the appropriate link below for the product of interest:

- HFSS
- HFSS Transient
- HFSS 3D Layout
- Icepak
- Mechanical
- Q3D
- 2D Extractor

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

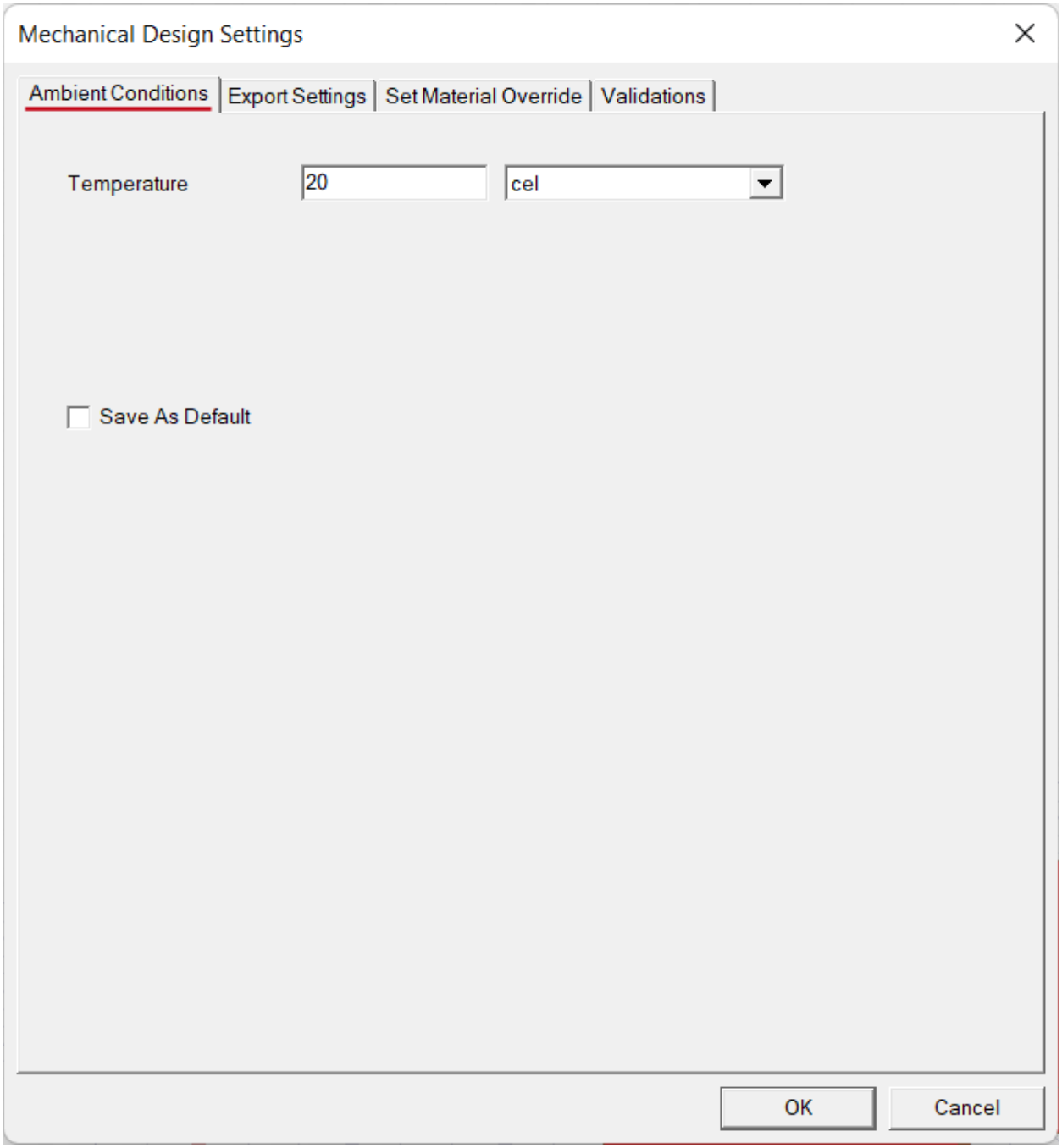
Design Settings for Mechanical

To set the design settings for Mechanical, use one of the following two methods:

- From the menu bar, click **Mechanical** > **Design Settings**. Or, right-click the active mechanical design in the Project Manager (**MechanicalDesign*x***, by default) and choose **Design Settings** from the shortcut menu.

  The *Mechanical Design Settings* dialog box appears with the *Ambient Conditions* tab initially selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



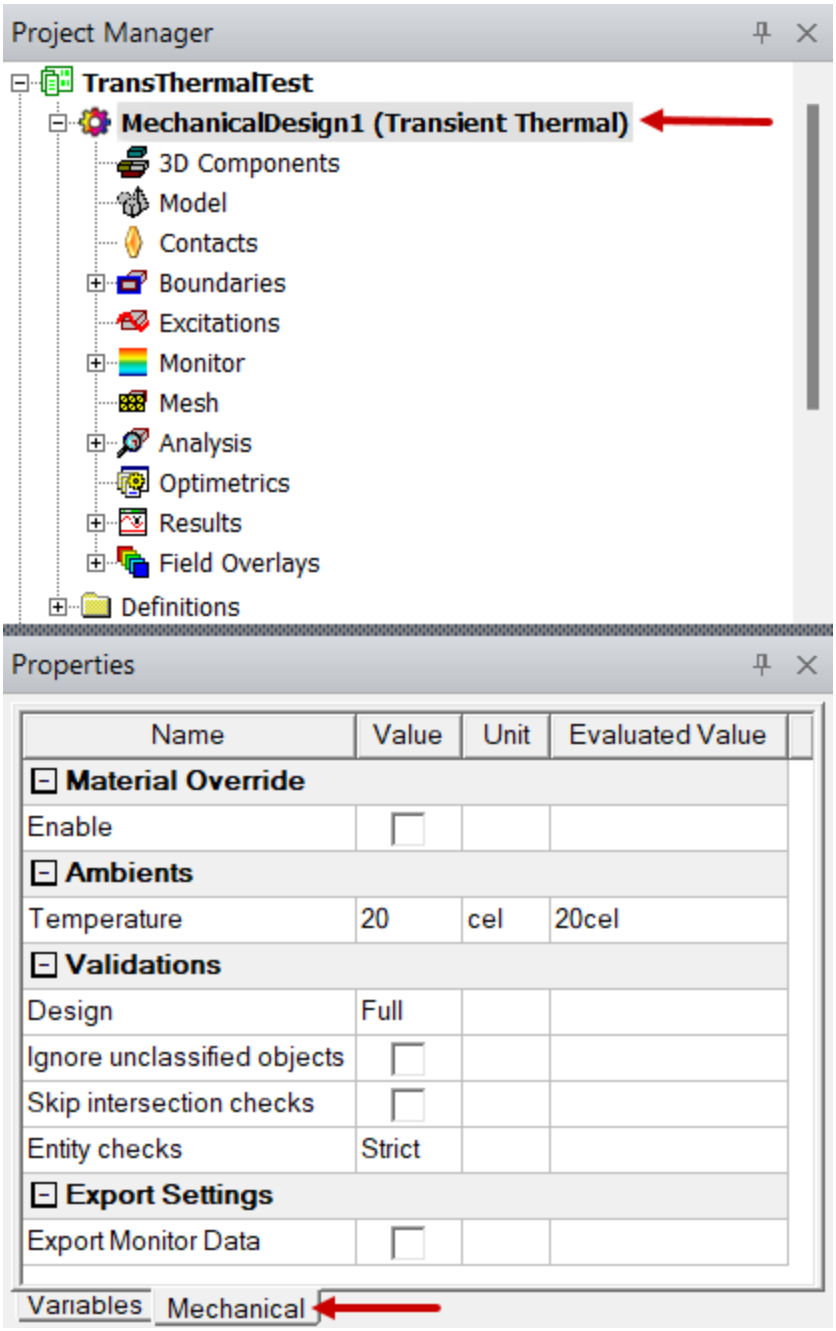
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The set of tabs that you see in the *Design Settings* dialog box is dependent of the currently active solution type, as detailed below.

- Alternatively, select the name of the active mechanical design in the Project Manager (**MechanicalDesign*x***, by default).

  The docked *Properties* window now includes a **Mechanical** tab, which contains a consolidated list of the settings from all of the *Design Settings* dialog box tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ambient Conditions Tab

This tab, and the corresponding *Ambients* group in the *Mechanical* tab of the docked *Properties* window, is applicable only to Thermal solutions (steady-state or transient). The following setting is available:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Temperature:** This value is the default ambient temperature for Convection Boundaries, which can be overriden for individual boundaries by specifying an alternative temperature in the *Convection* dialog box.

Export Settings Tab

This tab, and the corresponding *Export Settings* group in the *Mechanical* tab of the docked *Properties* window, is applicable only to Transient Thermal solutions. It provides options to export thermal monitor results and to specify where to locate the exported data, as follows:

- **Export monitor data after completing simulation:** By default, thermal monitor results are not exported. Select this option to automatically export the monitor results after the simulation has finished. The *result versus time* data for each monitor point is exported in a comma-separated-values (CSV) text format.
- **Override:** By default, the monitor data is exported to the same folder as where the current project file is located. Select this option to enable the specification of an alternative export path. This option is unavailable unless the option *Export monitor data after completing simulation* is first selected.
- **Path text box:** By default, the path to the folder where the current project file is saved appears in this box and cannot be edited. To change the path for storing the exported data, you must first select the *Override* option (previous bullet).
- **Ellipsis (...) button:** When overriding the default export folder path, click this button to open the *Choose Directory* dialog box for browsing to the desired folder. This button is unavailable unless the first two options are both selected.

Set Material Override Tab

This tab, and the corresponding *Material Override* group in the docked *Properties* window, is only applicable to Thermal solutions (steady-state and transient). It contains a single setting that controls how objects are meshed and how materials are assigned in regions where objects overlap each other.

This option is only applicable when intersection checking is performed, and it has no effect when the **Skip Intersection Checks** option is selected.

- When **Skip Intersection Checks** is **selected** (under the *Validations* tab), the following will occur, regardless of the **Enable material override** state:

  Intersecting bodies will *not* cause the design solution to fail. At each intersection, the body with the smaller total volume will take precedence over the body with the larger total volume. The volume of the intersection region is not considered, only the full object volumes. Within the intersection region, the mesh and material properties will be dictated by the smaller object.

  For the remaining conditions, the **Skip Intersection Checks** option is **cleared**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When **Enable material override** is cleared (*not selected*):

  The design will fail with an object intersection fatal error, both when performing a **Validation** in the user interface and during an attempt to solve the analysis.

- When **Enable material override** is **selected**:

  The design will not fail in validation or during the solution. The body having a material with thermal conductivity above a fixed threshold (specifically 10 W/(m·°C) will take precedence in the region of intersection. If both objects are above this threshold, or if both are below the threshold, the body with the lower total volume takes takes precedence.

> **Note:**
>
> - For cases where the geometry or material properties are such that the object to use in the region of overlap is ambiguous or cannot be resolved with a high level of confidence automatically, avoid the overlap. Instead, manually subtract the volume of intersection from the appropriate object to eliminate the intersection.
> - Even in cases of ambiguous object overlap, the meshing routine will choose to subtract one of the objects from the other. In such cases, the Message Manager window reports the objects that are intersecting and the object material assigned in the overlap region. The automatic choice may not correctly represent the actual assembly that you intend. Carefully check such messages to ensure the proper material assignment in the overlapping regions.

Validations Tab

This tab, and the corresponding group of settings in the *Mechanical* tab of the docked *Properties* window, provides options affecting model validation. These options control the extent of 3D model and Mechanical setup validations performed, and therefore the time involved. You can adjust the degree to which the software checks a model for faults that could degrade accuracy. The following options are available:

Model Options

The following settings affect only the "3D Model" stage of a design validation:

- **Ignore Unclassified Objects:** Generally, all objects in the 3D Modeler's History Tree are classified as solid, sheet, line, or point objects. However, in some cases, a drawing operation may produce null geometry, or it may fail and produce invalid geometry. For example, subtracting a tool object from an equal-volume or smaller blank object completely contained within the tool object would produce a null object. That is, the subtraction would completely remove the original blank object geometry. An example of a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

failed drawing operation is attempting to unite two solid objects when they only share an edge (not a portion of a face or volume). The union creates a non-manifold, invalid solid body. While not visible in the Modeler window, null and non-manifold objects are listed in the History Tree in an "*Unclassified*" branch.

It is usually best to leave this global modeler validation option cleared so that unclassified objects are not ignored but are flagged as errors.

- **Skip Intersection Checks:** This option, when selected, prevents detected object intersections (that is, overlapping volumes) from causing a fatal error and aborting the analysis solution. As this option also affects the behavior of material overrides, see the Set Material Override Tab section on this page.
- **Entity Check Level:** There are four levels of model validation that you can specify:
  - **None:** Disables model validation. Use this option only for a known good model (for example, when you are solving variations of a previously validated model and wish to reduce the solution time).
  - **Warning Only:** Allows all models to pass 3D Model validation regardless of any faults that are found. These faults are posted in the Message Manager window as warnings. However, whenever possible, it is best to repair model errors to ensure the accuracy of solution results.
  - **Basic:** Allows a greater tolerance for model errors. As a result, many models will pass basic 3D Model validation. Significant faults are still flagged as model errors, thereby prohibiting the design from proceeding to the meshing stage of the analysis. Correct such errors before attempting to analyze the design. Alternatively, you could reduce the entity check level to *Warning Only* but at the risk of questionable results accuracy.
  - **Strict:** This option is the defaul setting. It enforces a tighter tolerance for model faults than the *Warning Only* or *Basic* setting. All model faults found during 3D Model validation are posted in the Message Manager window. Correct these errors before attempting to analyze the design using the Strict setting. Alternatively, you could reduce the entity check level to *Basic* or *Warning Only*, with the understanding that the solution accuracy might be affected.

Mechanical Options

The following additional validation options appear in this section:

- **Perform full validations:** All design validations are performed.
- **Perform minimal validations:** All design validations are performed except for boundary and excitation overlaps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

In the *Mechanical Design Settings* dialog box (but not the docked *Properties* window), you have the option of saving the specified settings under any tab as the default for future projects. Do so by selecting **Save As Default** in one or more tabs.

Environment Conditions Tab

This tab, and the corresponding *Environment* group in the *Mechanical* tab of the docked *Properties* window, is applicable only to Structural solutions. The following setting is available:

- **Temperature:** This input sets the value of the global *EnvTemp* variable. It is the temperature at which the objects comprising the model are assumed to be in a stress-free and strain-free state. Temperatures assigned in, or imported into, the structural solution are interpreted relative to the *EnvTemp* value. The relative temperatures are use to determine thermal expansion or contraction and the resulting deformation and stresses. See Specifying the Reference Temperature for more information.

Model Preparation Commands

Use the **Modeler** > **Model Preparation** commands to prepare a model.

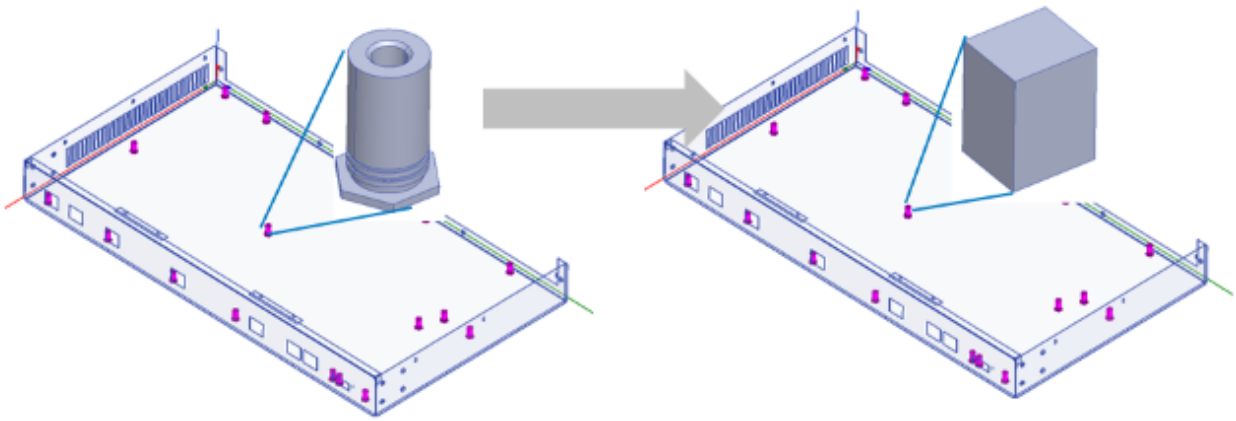
These commands include:

- Simplify
- Heal
- Stitch Sheets
- Align Faces
- Remove Faces
- Remove Edges
- Perform Explicit Subtractions

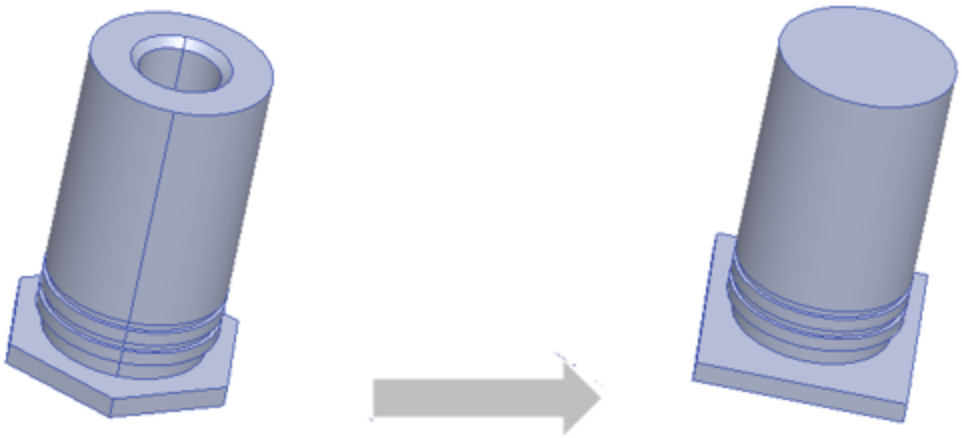
Simplify Command

Use the **Modeler** > **Model Preparation** > **Simplify** command to converts a complex MCAD object into simpler primitives which are easy to mesh and solve. The operation can be applied on any selected object and not just imported objects. You can specify the type of simplification as Bounding Box, Primitive Fit, or Polygon Fit.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Bounding Box Simplification

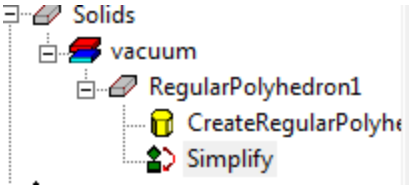


Primitive Fit Simplification

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

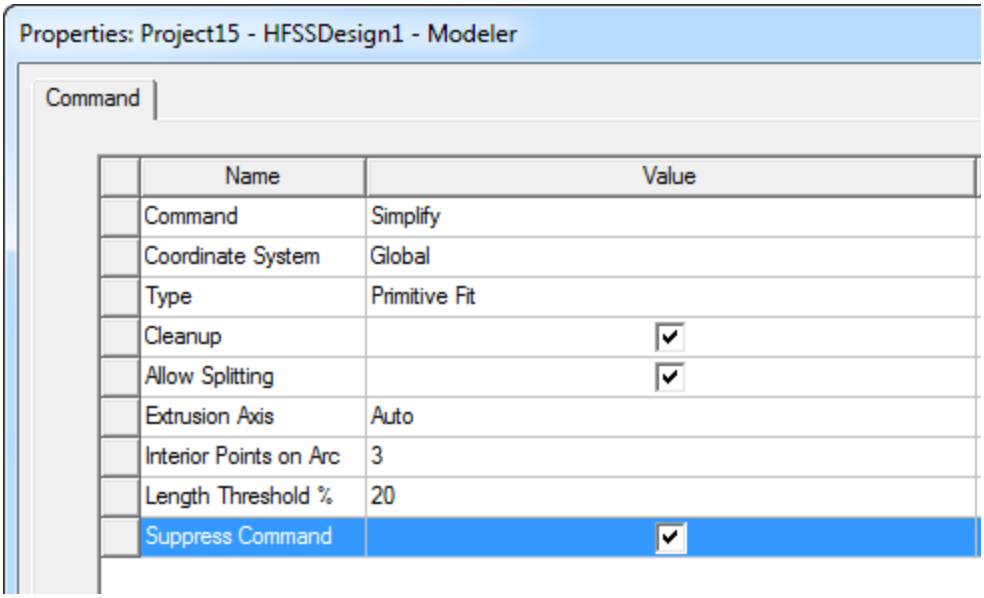
Polygon Fit Simplification

The History tree shows the **Simplify** command as being applied to the object.



Any parameters you specify in the dialog you can also edit in the Properties dialog for the History tree **Simplify** command to get a different simplification. The properties display in the docked properties window and include only the parameters that are used by your currently selected **Simplify** command type. None of the parameters of simplify operation accept variables.

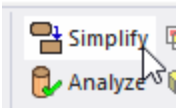
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Select an object.

   This enables the Simplify command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Simplify** or on the **Model** ribbon, select **Simplify**.



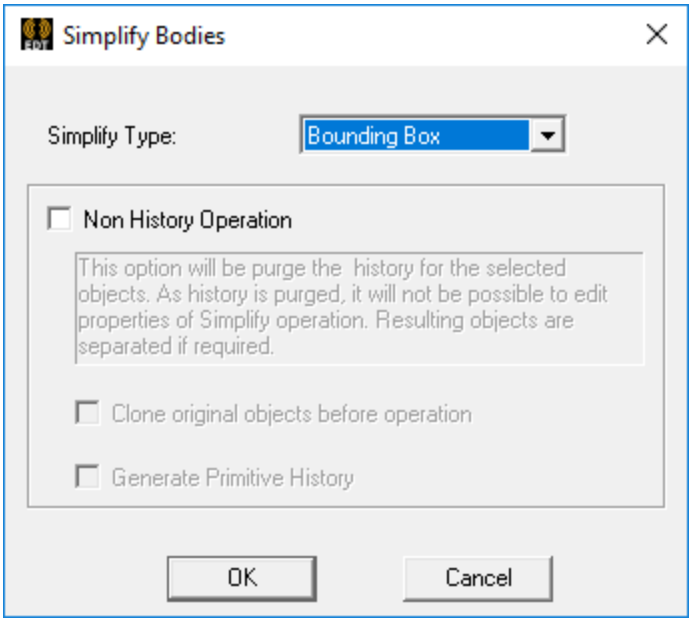
This opens the **Simplify Bodies** dialog box. A drop-down menu lets you select the degree of simplification, from Bounding Box, Primitive Fit, or Polygon Fit.

Bounding Box Simplify Type

If you select Bounding Box as the Simplify Type, there are no additional parameters. The object is replaced by its exact bounding box as computed in operation's coordinate system. You can change the operation CS to get a bounding box in an appropriate orientation. For the **Non**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

History Operation options, see below.

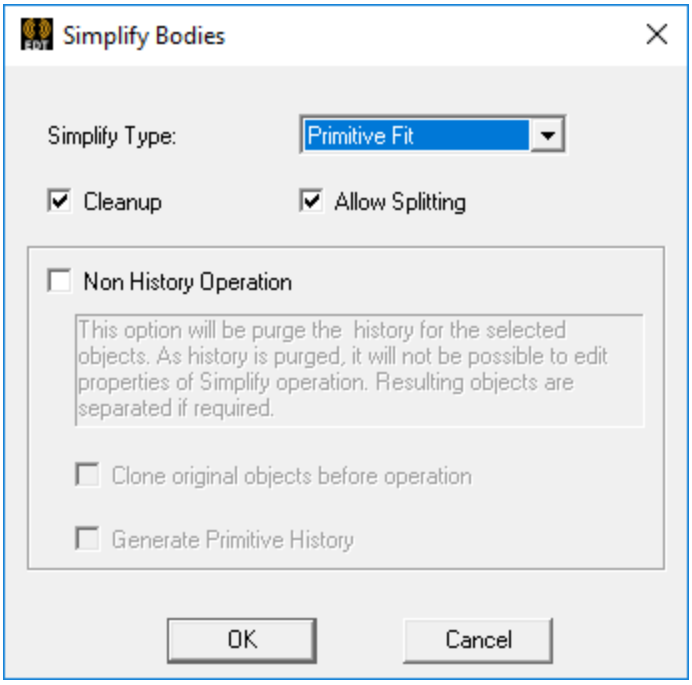


Primitive Fit Simplification

For Primitive Fit simplification the object is replaced by a set of primitives shapes like Prism, Cylinder, Cone and so forth. This simplification type typically produces the simplest geometry with highest number of primitives. Primitive fit has the following options: Cleanup and Allow Splitting, whether to Separate Bodies and Purge History, and if so, whether to clone original

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

objects before the operation.



For Primitive Fit, the parameters are:

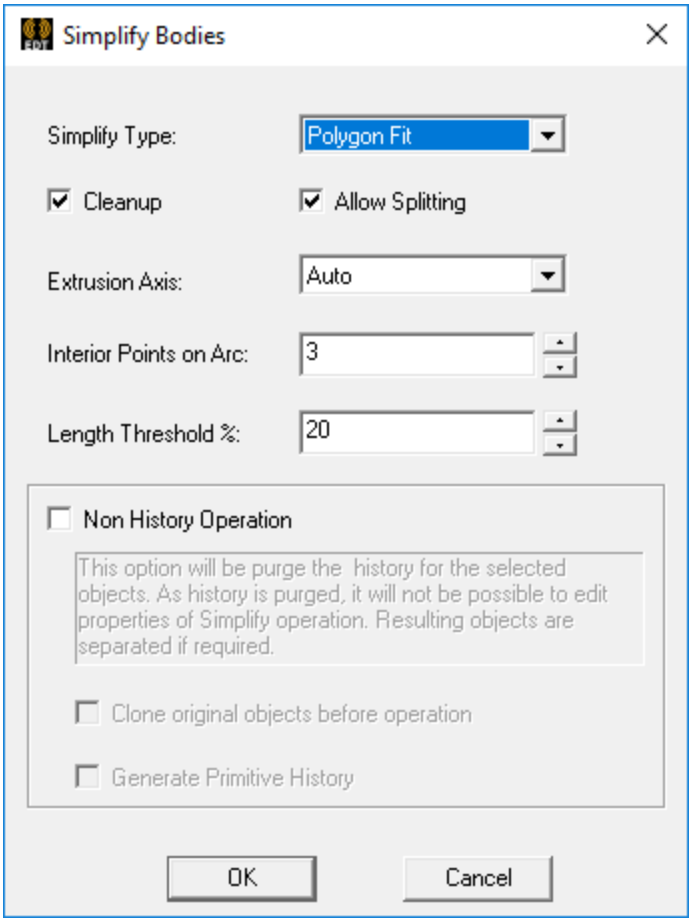
- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- For the **Non History Operation** options, see below.

Polygon Fit Simplification

For Polygon Fit simplification, the object is replaced by set of polygon swept along normal and other primitives like box and cylinder. This simplification type typically produces geometry closest to original object with highest number of primitives. If you select Polygon Fit as the Simplify Type, the parameters are Clean Up, Allow Splitting, Extrusion Axis, Interior Points on

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arc, and Length Threshold. For the **Non History Operation** options, see below.



For Polygon Fit, you specify parameters for

- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- **Extrusion Axis**, whether Auto selected, or X, Y, or Z. - This option specified a normal plane in which polygon profile is looked for. The default is Auto which allows an algorithm to determine best possible plane for polygon profile. You can specify a axis to help the algorithm, particularly when there is a draft or chamfer in the extrusion direction. The polygon profile is then swept in the extrusion direction.
- **Interior Points on Arc** - This specifies number of interior points created when a curve on the polygon profile is represented by line segments. The range is 1 to 100 with default

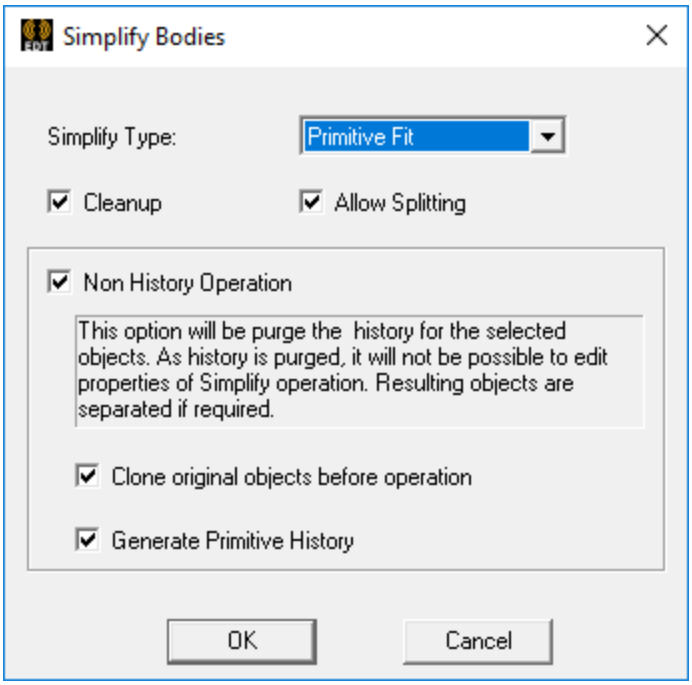
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

being 3. Total number of points used to represent a curve is 5 when number of interior points is 3. This number specifies the maximum number of interior points to add. The number of points added could be less if the Length Threshold is already met.

- **Length Threshold %** - This specifies the length of edge as percentage of maximum length of an edge in the profile. It is specified as percentage of maximum edge length. If length of edge is more than the specified threshold, more interior points are added until the number of “Interior Points on Arc” is met.
- For the **Non History Operation** options, see below.

Non History Operation Options

The Non History Operation options are available for all Simplify Types.



With this option selected, the input body is simplified into multiple primitives (or polygon shapes) and a new part is created for each simplified shape. This option also purges the history of original part and it is not possible Edit Properties of the Simplify operation. You can choose to clone original object before simplification to retain history of original part. You can also choose to generate Primitive History.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simplify result with separate bodies option

Use of Coordinate System

By default, the current working CS is used as the Simplify operation CS, but you can also change operation CS through the **Simplify** command property window or Property dialog.

Heal

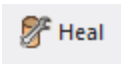
The **Heal** command provides a way to correct geometric violations and to remove specific kinds of small features. When models are imported, two types of errors can occur - geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. Ansys Electromagnetics recommends that these be fixed before you invoke mesh generation.

Imported objects which have only one operation on the history tree, can be healed. (Use the Purge History command to remove unwanted history operations before using **Heal**.)

> **Note:**
>
> If you need to save the object history, save a separate copy for that purpose before you heal the object.

Healing can be invoked in different ways.

- The menu command **Modeler** > **Model Preparation** > **Heal** command, or the **Model** ribbon **Heal** icon applies to a selected object.

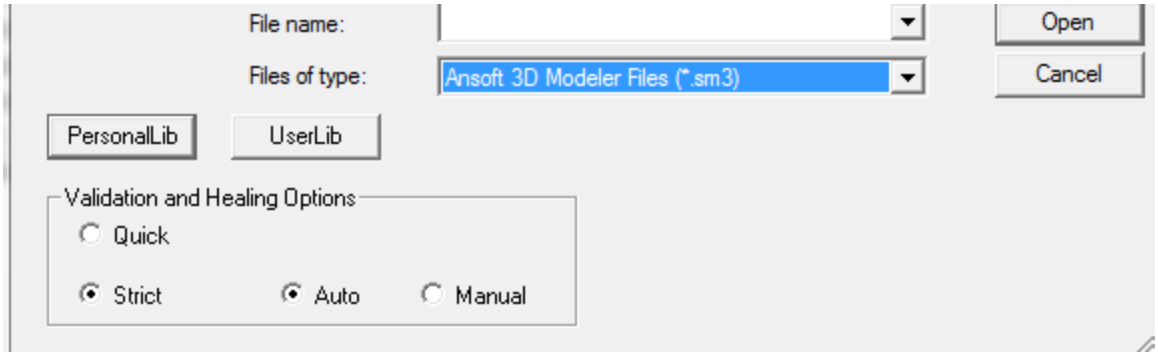
  

- Some formats permit healing during **Modeler** > **Import**. These are:
  - 3D Modeler file (*.sm3),
  - SAT file (*.sat),

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- STEP file (*.step,*. stp),
- IGES file (*.iges, *.igs),
- CreoParametric files (*.prt, *.asm),
- CATIA (*.model, *.CATpart), and
- Parasolid file (*.x_t, and *.x_b).

Selecting these formats enables a check box at the bottom of this window, **Validation and Healing Options**.



See this table for details. The Quick option allows you to switch off healing to speed up the import process.

- The **Model Analysis** dialog box that appears after running **Modeler** > **Model Analysis** > **Analyze Objects**, or **Modeler** > **Model Analysis** > **Show Analysis** dialog includes a **Perform** action menu with **Heal Objects** as a selection.

Any of these approaches leads to the same heal process.

Basic Steps in the Heal Process

There are several steps that are performed on selected objects.

1. Entity check, according to the **Analysis Options** settings.
2. Basic healing. This is done for all selected objects. Basic healing consists of fixing surface normals in the object and updating the orientation (to avoid having an object with negative volume).
3. Advanced healing. This is auto-heal. This is invoked on objects that require healing, that is, bodies that have errors, including have non-manifold errors.
4. Feature Removal. If you choose in the **Healing Options** to remove small holes, chamfers, blends, small edges, small faces and/or sliver faces, the actions are performed on all selected objects. There is no guarantee that small feature removal will be successful. (Also see Specifying the Model Resolution for defeaturing through the Auto Simplify and Model Resolution settings there.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The above actions are performed on the selected objects. If you choose objects for healing which have not been analyzed, analysis is performed to determine its state (that is, whether it has invalid entities, small entities, and so forth). Invalid objects have all the above steps performed. Advanced healing is not performed on objects that do not require it.

While working on analyzing complex bodies, it is sometimes useful to examine faces, edges and vertices. In particular it is useful to find the connected faces for a face or edge or vertex, connected edges for a face/edge/vertex and connected vertices for a face/edge/vertex. The additional selection modes are available under **Edit** > **Selection Mode**.

Validation and Healing Options for Import File

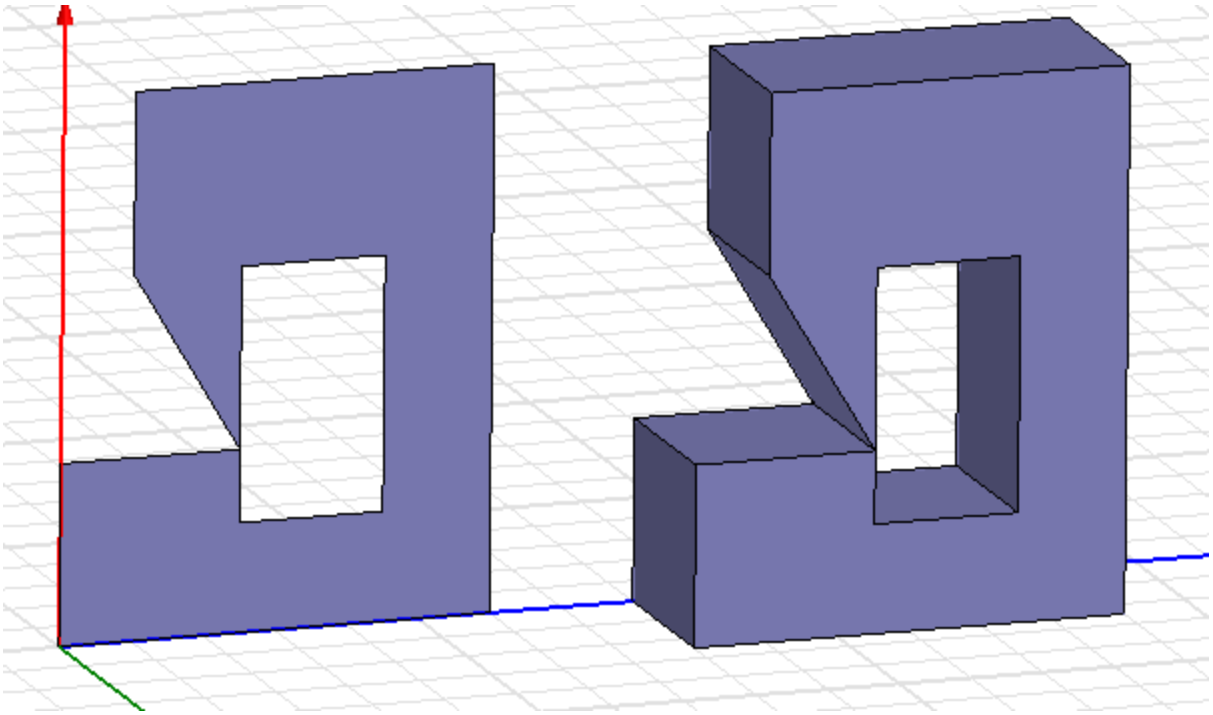
The **Import File** dialog box contains shows different Validation and Healing options for the seven formats listed here. The Quick option allows you to import these formats without healing.

Format	**Options Available**			
3D Modeler file (*.sm3),	Quick	Strict		
SAT file (*.sat),	Quick	Strict	Auto or manual	
STEP file (*.step,*.stp),	Quick	Strict	Auto or Manual	Stitch Tolerance
IGES file (*.iges, *.igs)	Quick	Strict	Auto or Manual	Stitch Tolerance
Creo Parametric files (*.prt, *.asm)	Quick	Strict	Auto or Manual	Import Free Surfaces. This imports such surfaces as well as parts.
CATIA (*.model, *.CATpart)	Quick	Strict	Auto or Manual	Stitch Tolerance
Parasolid file (*.x_t, and *.x_b)	Quick	Strict	Auto or Manual	

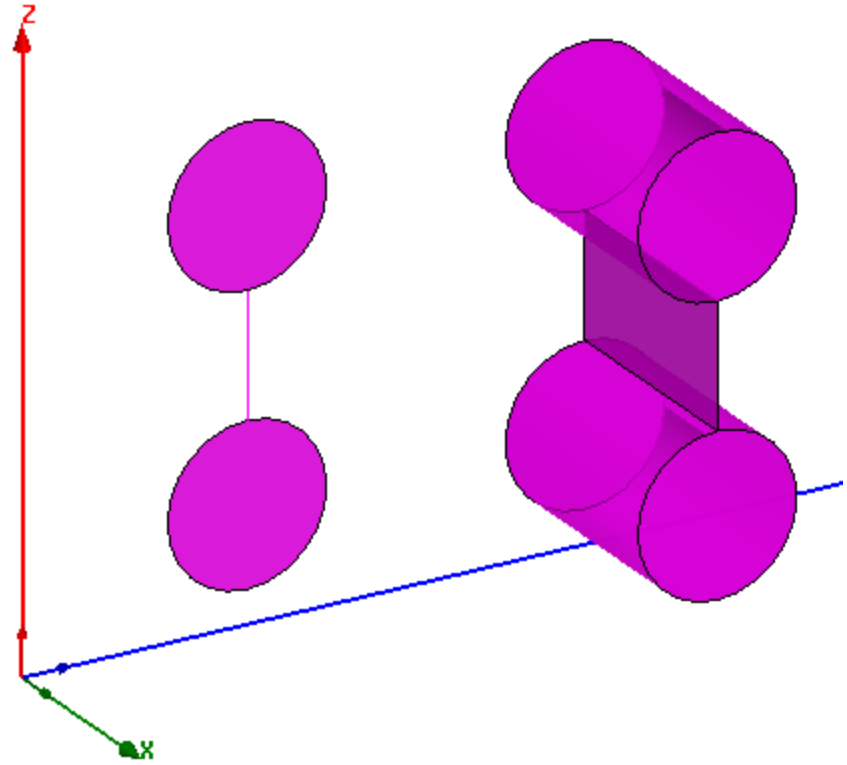
Healing Non-Manifold Objects

Non-manifold objects, in simple terms, are non-physical objects or objects that cannot be manufactured. For example, objects that intersect themselves (like the symbol for infinity in 2D) are clearly non-manifold. In addition objects that touch themselves may be non-manifold such as when a 2D object touches itself at a vertex, or a 3D object touches itself at a point or edge. These cases are shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Another type of non-manifold object has mixed dimensionality. For example, a pair of 2D objects connected by a 1D line segment, or a pair of 3D objects connected by a 2D sheet object. These cases are illustrated below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The criteria for manufacturability is a simple manifestation of a complex mathematical concept that must be adhered to in the solid modeling system. When creating geometry, either directly, or through boolean operations, you should always consider whether or not the resulting operation will result in an object that could not be manufactured. If this is the case, then the object will cause an error in the modeler or in the meshing system.

To heal non-manifold objects:

1. Identify an edge that is non-manifold.
2. Select the connected faces.

   You can use the face selection toolbar icons.

3. Create a face coordinate system on the planar face.
4. Create a small box to cover the non-manifold edge.
5. Either do a union or a subtraction to remove the faces that contain the non-manifold edge.

   The non-manifold edge is now removed. You may also remove or add a small portion of the model.

6. Repeat for all non-manifold edges.

Setting the Healing Options

The **Healing Options** let you control how healing proceeds with respect to a variety of features and issues.

1. Click **Modeler** > **Model Healing** > **Heal** to open the **Healing Options** dialog box. You can also open the **Healing Options** dialog box from the **Model Analysis** dialog via the **Objects** tab drop down menu.

   The **Healing Options** dialog contains three tabs:

   - **Healing Options**
   - **Feature Removal Options**
   - **Properties**, which lists the geometric properties of the currently selected object.

2. Select the **Healing Options** tab to specify the following:
   - **Heal Type** as: **Auto Heal** (default), **Manual Heal**, or **No Heal**. Selecting **Manual Heal** enables the **Manual Heal Options**:
     - **Perform Tolerant Stitching** check box: This enables a field for the **Stitch Tolerance** value, and a check box to **Stop After First Error**.
     - **Perform Geometry Simplification**: This enables fields for **Simplification Tolerance** and **Maximum Generated Radius** values. You can also select radio buttons to **Simplify Curves**, **Surfaces**, or **Both**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Tighten Gaps** settings: A check box to select **Perform Tighten Gaps**. A field to specify **Tighten Gaps Within** a given value in mm.

3. Select the **Feature Removal Options** tab to specify the following:

   Feature Removal Options:

   - **Remove Holes** check box and **Maximum Radius** value.
   - **Remove Chamfers** check box and **Maximum Width** value.
   - **Remove Blends** check box and **Maximum Radius** value.

   Remove Small Entity Options:

   - **Small Edges, Length Less Than**, less than a specified value.
   - **Small Faces Area e Less Than**, less than a specified area.
   - **Sliver Face Width Less Than**, less than either:
   - **Object Bounding box Scale Factor**, less than a specified scale factor
   - **Sliver Edge Width**, less than a specified value.

     > **Note:** Sliver faces have a maximum distance among the long edges that is smaller than the specified tolerance and have at least one short edge and at most three long edges. A short edge has a length less than the specified tolerance. A long edge has a length greater than the specified tolerance. You can give the tolerance as a absolute value or a factor of the bounding box containing the face.

   Control Object Properties Change options:

   - **Allowable Change in Surface Area** check box, and percent value.
   - **Allowable Change in Volume** check box, and percent value.

4. Select the **Properties** tab to view the geometric properties of the currently selected object.

5. Click **OK** to apply the specified Healing options and to open the Analysis dialog box.

Stitch Sheets

Use the **Modeler** > **Model Preparation** > **Stitch Sheets** command to stitch selected sheets.

1. Select two or more sheet objects.

   This enables the **Stitch Sheets** command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Stitch Sheets**; or, on the **Model** ribbon, click the **Stitch Sheets** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

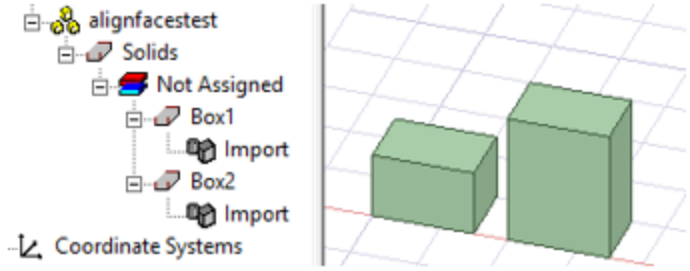
This displays a **Stitch** dialog box with a **Maximum Stitch Tolerance** field. The default value (auto) comes from the **Healing** dialog box's **Options** tab with Manual Healing selected. You may edit the value in the **Stitch** dialog box or in the **Healing Options**.

3. Click **OK**.

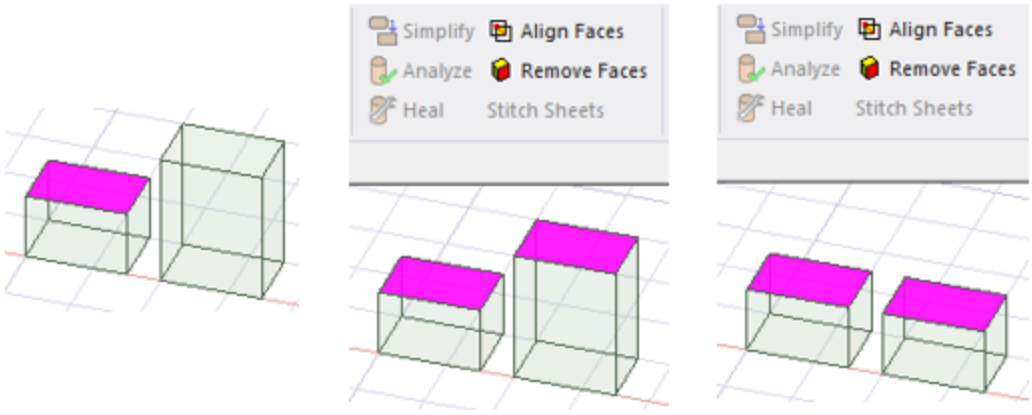
   This closes the dialog box and attempts to perform stitching on the selected sheets. If the sheets are separated beyond the stitch tolerance, stitching is not performed and a warning is issued.

Align Selected Faces

Use the **Modeler** > **Model Preparation** > **Align Faces** command to align the adjacent selected faces of imported objects that have only one operation in their History Tree. You can use **Undo/Redo** on this command.

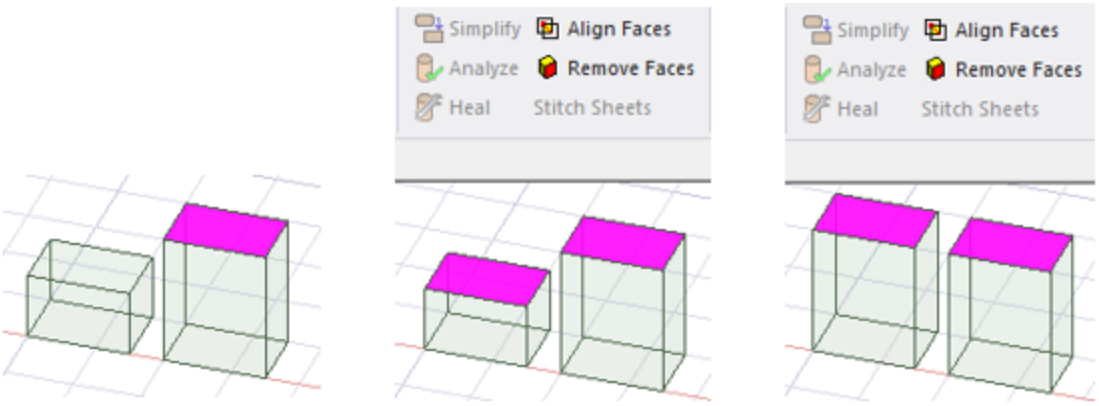


The **Model** ribbon also includes an **Align Faces** icon, which is enabled when you make appropriate face selection. The first selected face defines the plane of alignment for subsequent selection, that is, tool faces to blank faces.

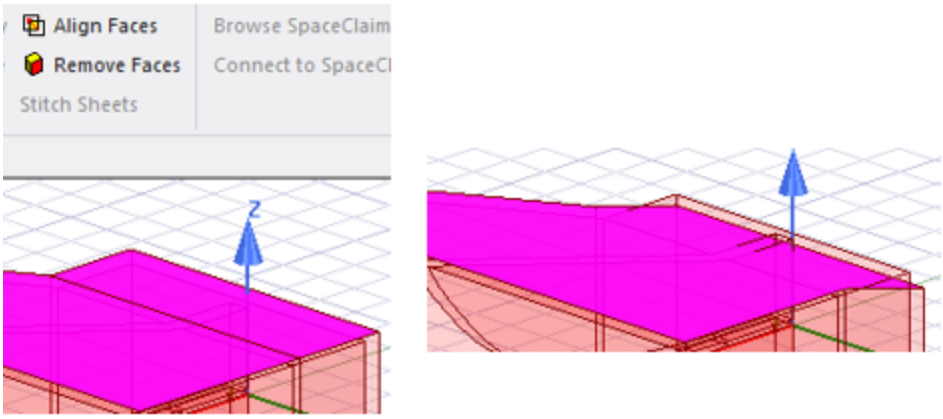


Selecting faces in a different order changes the alignment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

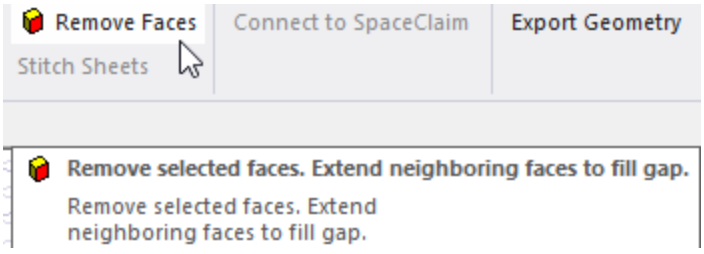


The selected faces can touch.



Remove Selected Faces

Use the **Modeler** > **Model Preparation** > **Remove Faces** command to remove the selected faces, extending the remaining faces to fill gaps. You can also use the **Model** ribbon **Remove Faces** icon when you have made an appropriate face selection.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you find object-pair intersections that healing does not fix, or that can be fixed (by alignment), you can correct the problem by one of the following methods.

1. Use the **Remove Faces** command (**Modeler** > **Model Preparation** > **Remove Faces**) or by performing Boolean subtract.
2. If overlap between objects is too large to be fixed by healing or by face alignment. Boolean intersect shows the common portion between the bodies. In this case, use a subtract operation to remove overlaps.

Remove Selected Edges

Use **Modeler** > **Model Preparation** > **Remove Edges** to remove selected edges. You can also use the toolbar icon when you have made an appropriate edge selection.

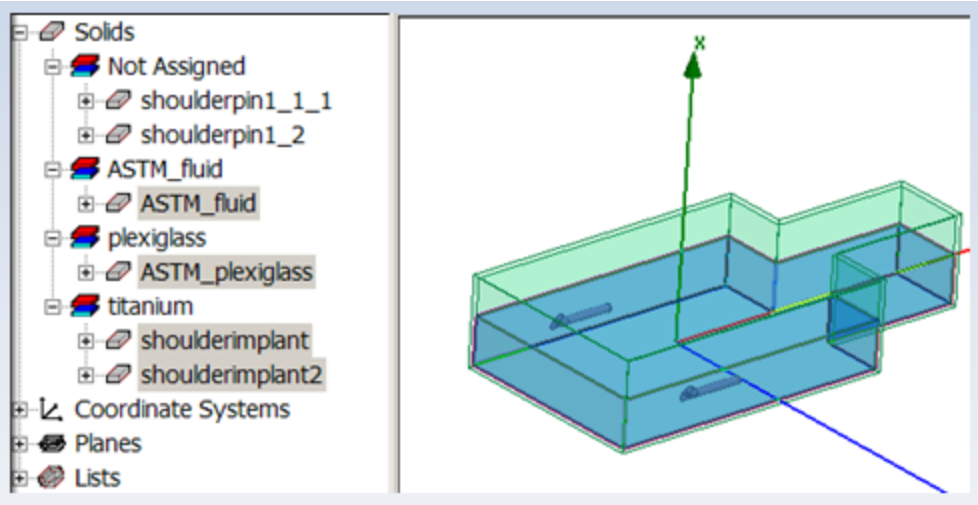
Perform Explicit Subtractions

Use **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** to subtract selected History Tree objects that are fully contained by other parts.

1. Select two or more history objects.

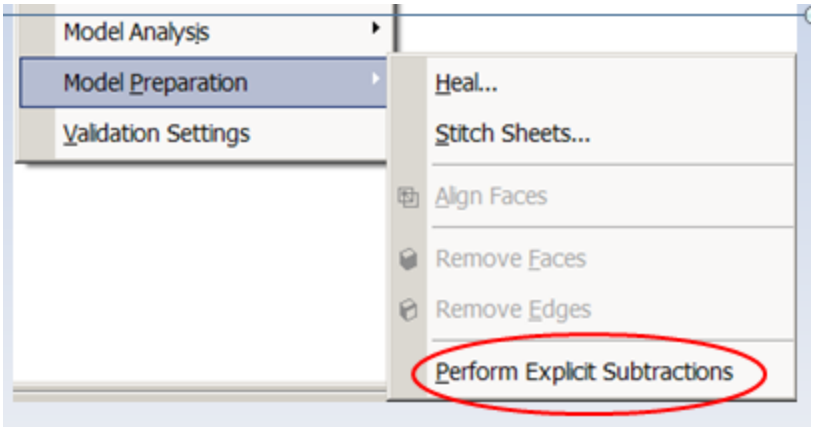
   This enables the **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** submenu.

   For example, consider the following model.

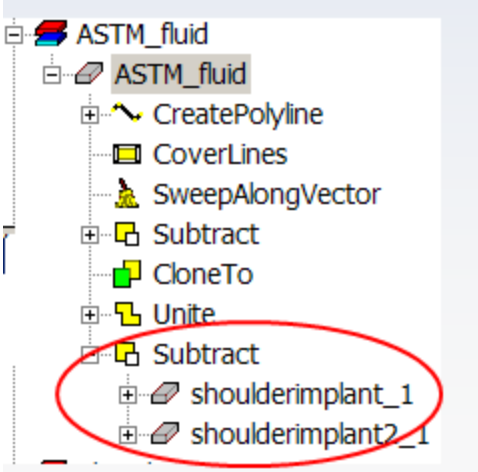


2. **Perform Explicit Subtraction** can be performed.

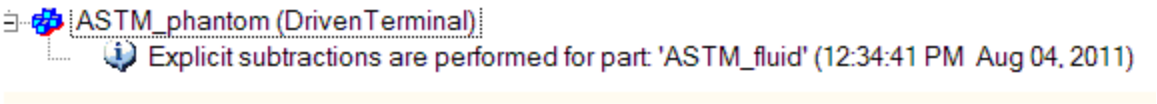
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The results appear in the History tree as shown:



The Message window reports this successful action.



If no parts are fully contained by other parts, a message explains that no explicit subtraction is needed.

Creating a User-Defined Primitive

The modeler allows you to generate user-defined primitives, primitives customized to suit any application. User-defined primitives are accessed using DLLs or IronPython scripts that you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

build and compile. When user-defined primitives exist in your UserLib or PersonalLib directory (given the paths specified in **Tools** > **General Options**, Project Options tab), they appear in the menu bar under **Draw** > **User Defined Primitives**. Newly created UDPs will appear after a restart or after executing the command **Draw** > **User Defined Primitive** > **Update Menu.**

For Python-based primitives, see the **Mechanical Scripting Guide**.

The modeler includes example C++ source and header files that can be used to generate DLLs. The files are located in the **UserDefinedPrimitives/Examples** subdirectory under the directory.

As an example, create the primitive **myUDP.dll** using Microsoft Visual C++ Developer Studio:

1. Create a directory to store all of the workspace information, call it UDPDir.
2. Use the sample workspace **RectangularSpiral.dsw** as a template:
    a. Copy **RectangularSpiral.dsw** and **RectangularSpiral.dsp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new files have write permissions.
    c. Rename the files to **myUDP.dsw** and **myUDP.dsp** respectively.
    d. Open the **.dsw** and **.dsp** files in a text editor, and replace every occurrence of **RectangularSpiral** with **myDLL**.
    e. Save **myUDP.dsp** and **myUDP.dsw**.
3. In the **UDPDir** directory, create a **Headers** subdirectory.
4. Copy the **UserDefinedPrimitiveStructures.h** and **UserDefinedPrimitiveDLLInclude.h** files from the **UserDefinedPrimitives/Headers** directory.

> **Note:**
>
> The header files include information on the methods that are available for use in your source code. They must be included when you compile the DLL.

5. In the **UDPDir** directory, create a **Source** subdirectory.
6. Use the sample source file **RectangularSpiral.cpp** as a template:
    a. Copy **RectangularSpiral.cpp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new file has write permission.
    c. Rename the file to **myUDP.cpp**.

The resulting directory structure will resemble the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPDir/
        myUDP.dsw
        myUDP.dsp
        Headers/
          UserDefinedPrimitiveDLLInclude.h
          UserDefinedPrimitiveStructures.h
        Sources/
          myUDP.cpp
```

7. Open **myUDP.dsw** using Microsoft Visual C++ Developer Studio, and edit the source code to create your desired primitive. You may also add additional headers and source files as appropriate.

   The UDP dll contains a data structure called UDPPrimitiveTypeInfo. This contains information about the udp, its purpose, company/author who created it, date created and the version number. When you select a primitive from your library, you see the **Create Primitive** dialog with a **Parameters** tab for setting the parameters, and an **Info** tab with the information from this data structure.

8. Build **myUDP.dll** using the **Win32 Release** configuration.
9. Copy the resulting file **myUDP.dll** to the **[*ProductInstallationPath*]/userlib/UserDefinedPrimitives** directory or the **[*PersonalLibPath*]personallib/UserDefinedPrimitives** directory.
10. To view your primitives, click **Draw** > **User Defined Primitive** > **Update Menu** and then click **Draw** > **User Defined Primitive** > **[*PrimitiveToDraw*]**.

> **Note:**
>
> On Linux, you may use the same example directory structure, source, and header files to build and compile a shared library using C++. The resulting shared library will have a **.so** extension and needs to be placed in the same */userlib/UserDefinedPrimitives* directory.
>
> As with the Windows DLL, the compiled library will work only on the operating system on which it was built.

User Customization through User Defined Primitives (UDPs)

User Defined Primitives (UDPs) allow users to add customized geometric modeling commands to the Ansys Electronics Desktop. UDPs are C++ compiled or Python libraries that can be added to the desktop interface and shared between users with common modeling needs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

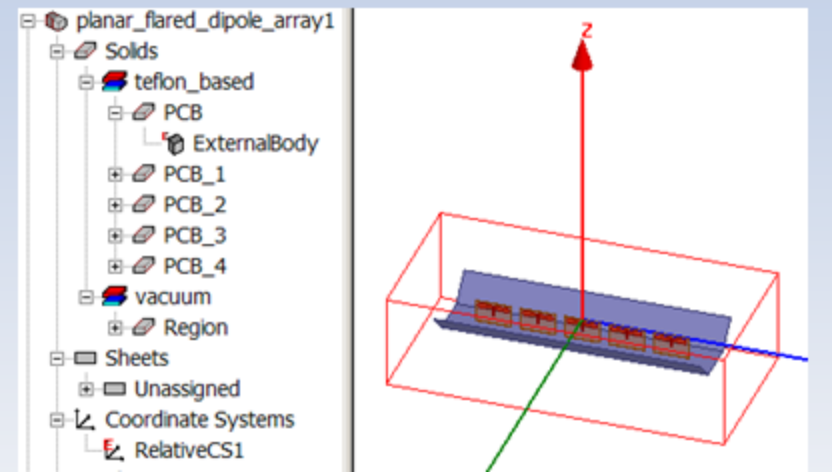
To create a UDP, see Creating a User Defined PrimitiveCreating a User Defined Primitive for requirements and the procedure for building a proper DLL or Python library.

In order to share UDPs between users, an existing DLL or Python library may be copied into the **UserLib** > **User Defined Primitives** subdirectory which can be given the paths specified in **Tools** > **General Options** under *General* > *Directories*. Placing an appropriately constructed DLL or Python library in this subdirectory and executing **Draw** > **User Defined Primitives** > **Update Menu** adds a new menu item in the **Draw** > **User Defined Primitives** menu to allow access to the UDP.

User Defined Model (UDM)

A User Defined Model (UDM) is collection of externally defined parts imported into an Ansys EM 3D Modeler or created using a C or Python script.

- UDM includes part attributes (like name, color etc) and material assignment
- UDM can also have external coordinate systems and corresponding planes
- UDM parts can be parameterized and manipulated in Ansys EM modeler just like any other part



UDM can either represent Static geometry models or

- Dynamic links to models of external geometry editors

UDM uses same plugin technology as User Defined Part (UDP)

See the following sections:

UDM compared to User Defined Primitives

Insert UDM Command on Draw Menu

UDM Properties

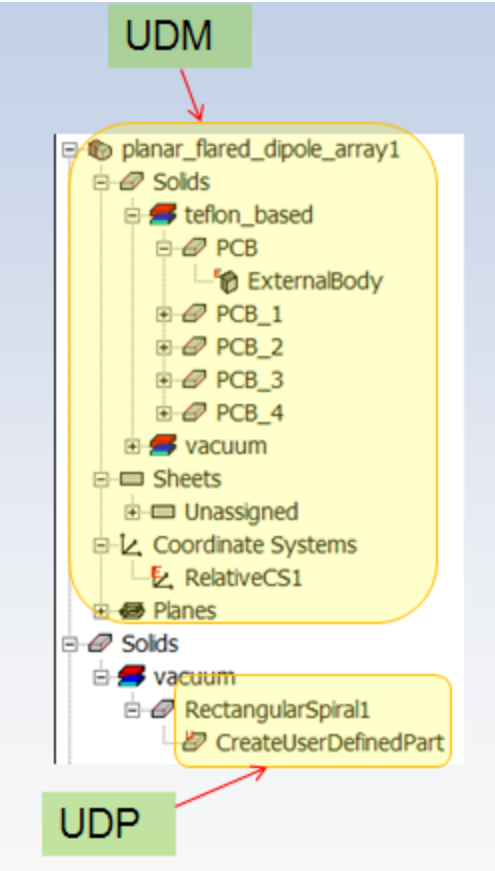
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Library of Models for CAD Integration

See the **Mechanical Scripting Guide** for information about Creating User Defined Primitives and User Defined Models in Python Scripts.

UDM Compared to User Defined Primitives

User Defined Models (UDM) resemble User Defined Primitives (UDP):



- Ansys EM products can be extended by users through new UDMs
- UDM plugins are discovered by searching standard directory paths
- Plugins for static UDM can build model using 'callback interfaces' (like create-box, create-cylinder, subtract etc) similar to UDP.
- UDMs run inside Ansys EM applications
- UDMs provide geometry, topology, persistence and parameters

  In contrast to UDP:

- UDM provides multiple Parts/CS/etc.

  UDP provides primitive operation only for a single Part

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- UDM provides part attributes and material assignment

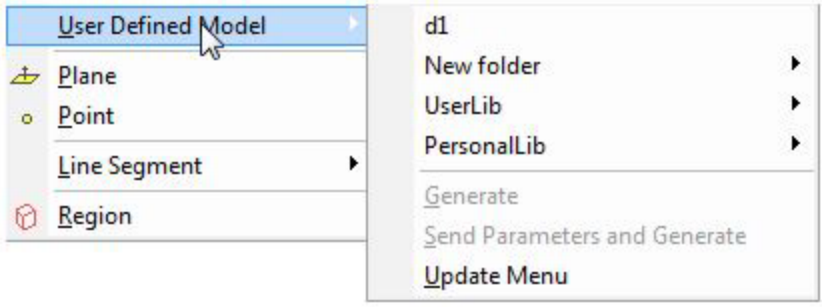
  UDP does not define part attributes or material

  UDM Properties have four tabs - Definitions, Parameters, Options and Info

- Definition tab has:
  - UDM name
  - Coordinate system used to position UDM
  - May have external reference to file
- Info tab has:
  - UDM dll or .py name, dll or .py location, version etc
- Option tab:
  - may have options if any

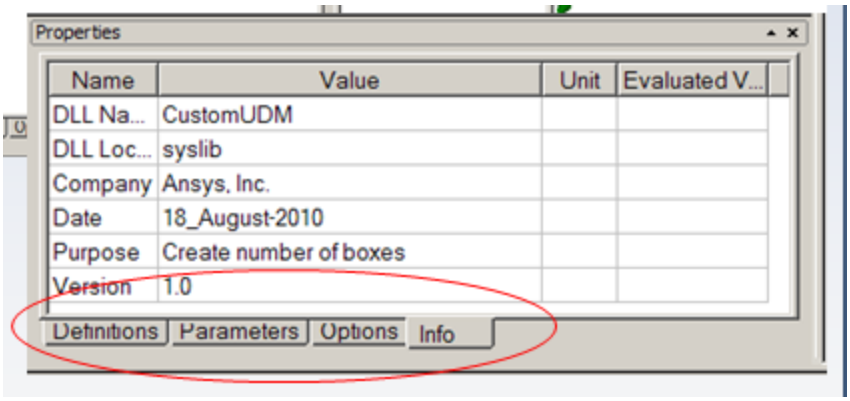
Insert UDM Command on Draw Menu

To insert a UDM into a design, use the User Defined Model command on the Draw menu for the Modeler window.



UDM Properties

UDM Properties have four tabs - Definitions, Parameters, Options and Info.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition tab has

- UDM name
- Coordinate system used to position UDM
- May have external reference to file

Info tab has:

- UDM dll or .py name, dll or .py location, version etc

Option tab:

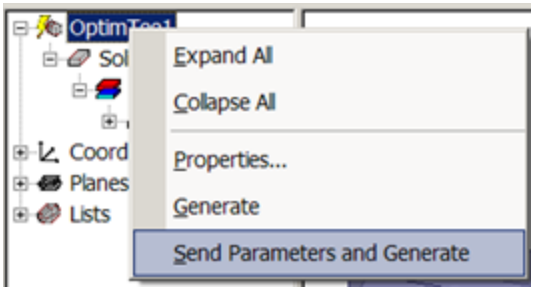
- may have options if any

UDM Parameters

UDM geometry can be manipulated through its parameters.

- Can be mapped to design or project variable for animation, parametric analysis.
- IDs are persisted (allowing to retain boundaries) during parameter edits.
- In such cases, UDM shows a lightning bolt icon by the model name when parameters are edited.

- You must run the **Send Parameters and Generate** command to synchronize parameters with geometry



UDM Part Edits

Several modeling operations are allowed on UDM parts

- Operations will be part of history tree and retained during model refresh

The following operations are not allowed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Non history tree operations like healing, defeature.
- Operations which use UDM parts as tool, such as sweep or boolean (but allowed when clone tool option is selected)

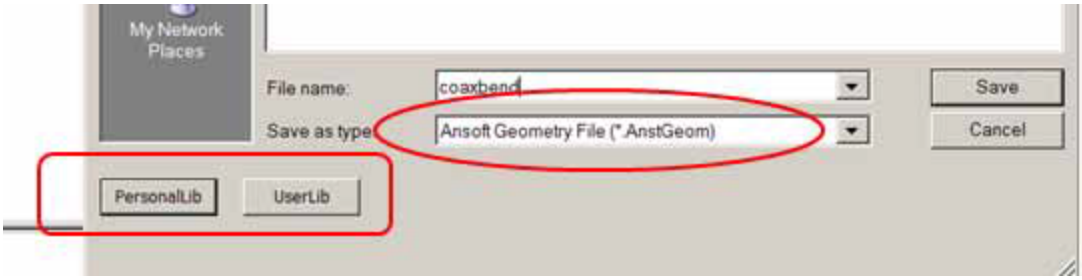
Following part attributes can be modified for UDM parts

- Model/Non Model flag
- Solve Inside flag
- Part orientation

Library of Models for CAD Integration

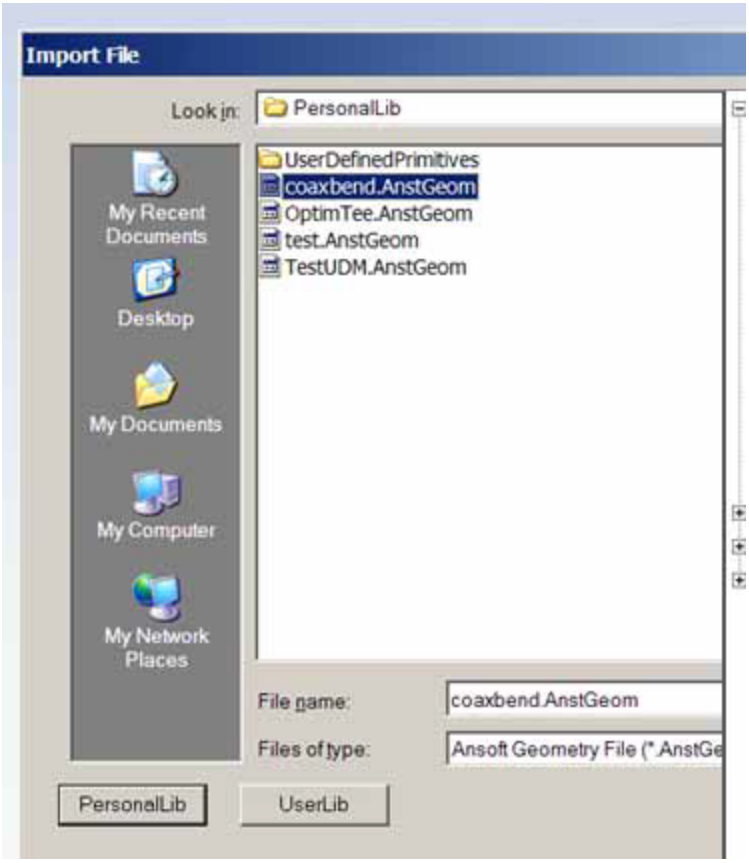
UDM technology allows a library of models

- Any Ansys EM model can be exported as 'Ansoft Geometry File'

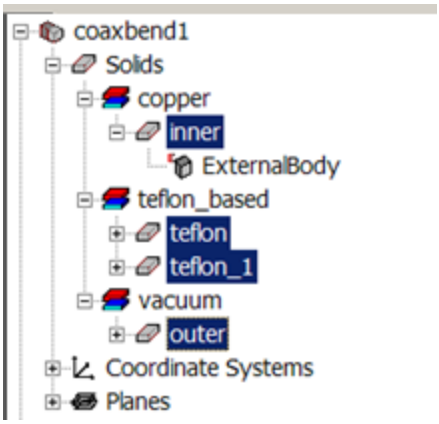


- An Ansoft Geometry File can be imported back as a UDM.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Any Design/project variables associated with model are brought in as UDM parameters.



Geometry computation for UDM (and also UDP) can be specified in the Modeler options as either done on.

- Engine side (default):

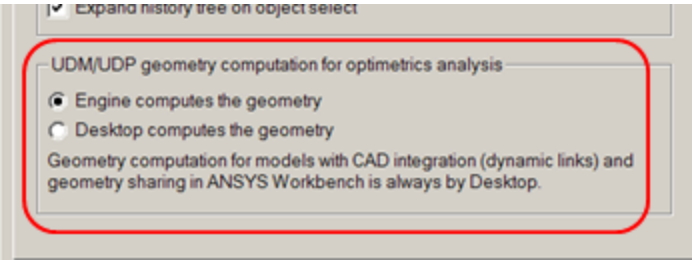
  Requires deployment of UDM on each node

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Desktop side:

  UDM need not be deployed on each engine

  Desktop will be busy during parametric analysis



3D Modeler Discovery Link

The 3D Modeler Discovery Link allows you to create a User-Defined Model (UDM) from the geometry model in an Ansys Discovery project. Discovery Link can be used only if Discovery with the same version as Ansys Electronics Desktop is installed on the same machine. Discovery Link is only available on Windows.

Discovery Link provides following functionality:

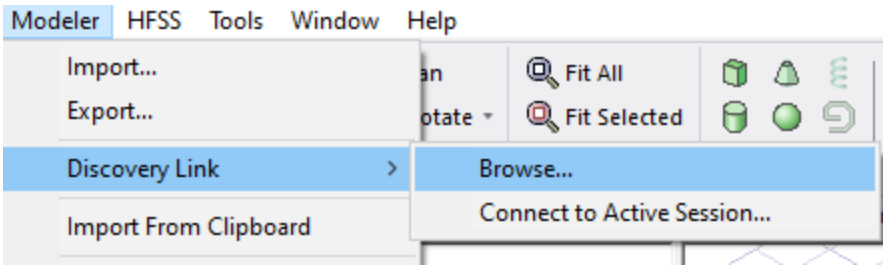
- Browse and select Discovery project files (*.dsco) to import geometry directly.
- Import geometry by connecting to an active Discovery session.
- The Discovery geometry is imported as a UDM along with geometry parameters with units, rendering attributes and material assignments. Object, material, and parameter names with non-ASCII characters are not supported.
- The Discovery geometries can be imported as faceted bodies if the design supports lightweight geometries.
- A Discovery document referenced by a Discovery link is included in the project archive automatically.
- You can break a Discovery link. In this case, all the parts in the UDM are converted to imported objects.

Importing Discovery Geometry into Ansys Electronics Desktop

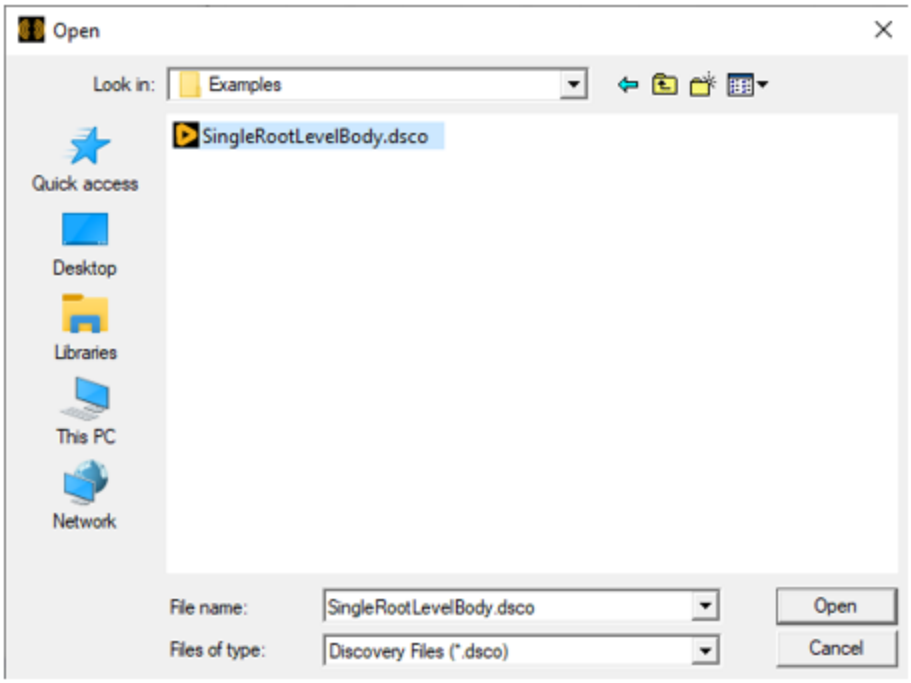
To select a Discovery file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Modeler** > **Discovery Link** > **Browse** or **Browse Discovery** in the **Model** ribbon.



2. Navigate to and select a *.dsco project to load into the Modeler window.

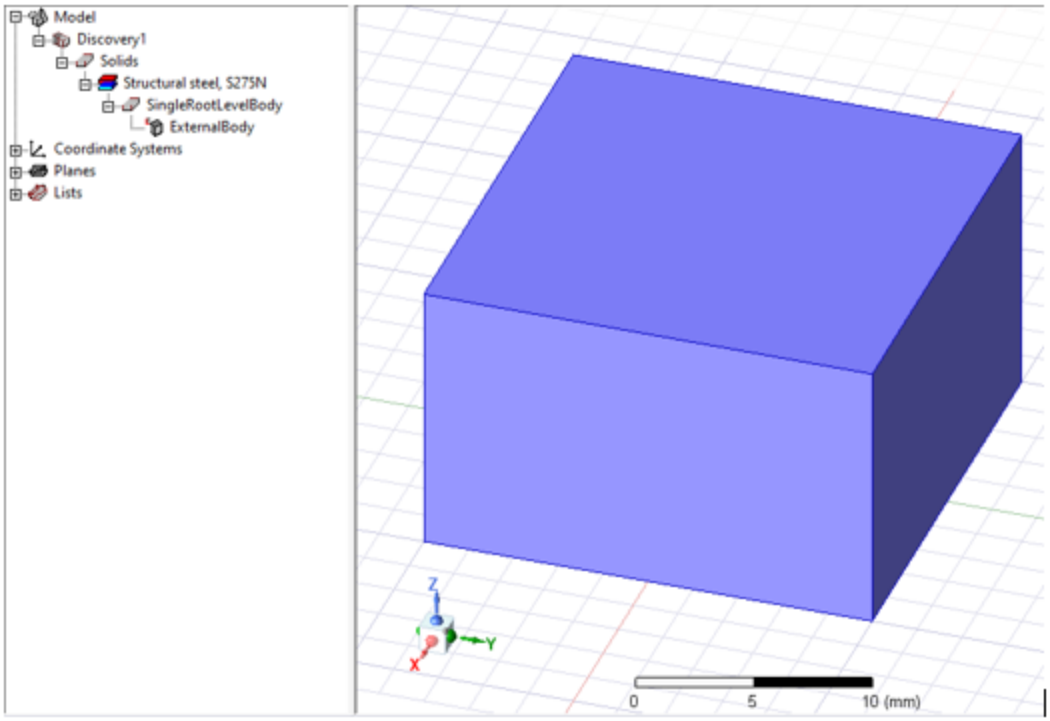


3. Click **Open**.

   The progress bar displays import progress. When the model has loaded, it appears in the Modeler window, and in the history tree as a User-Defined Model (UDM) listed under

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discovery.



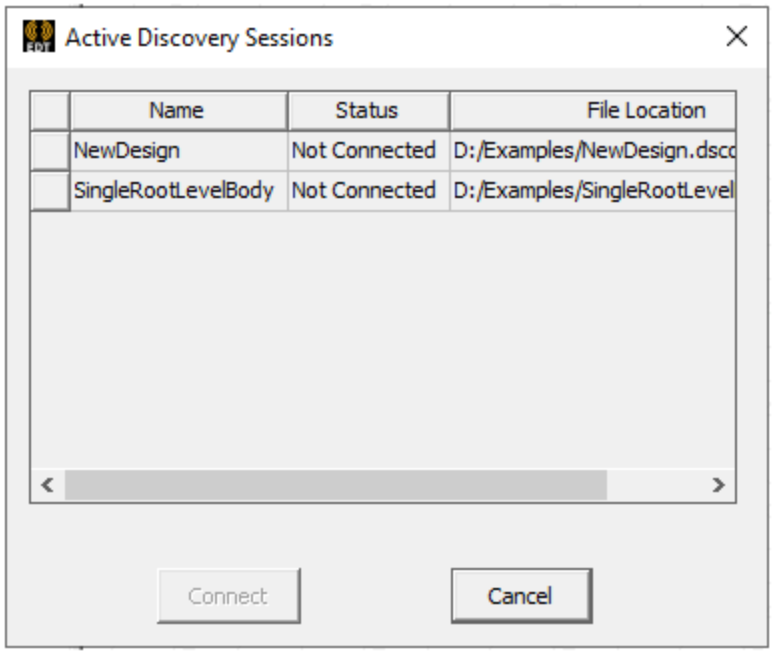
If there is an active Discovery session which is using the selected document, the created UDM will be linked to the active Discovery session. If there is not one, a new non-graphical Discovery session will be used, which makes the creating and refreshing slow. Afterwards, if an active Discovery session is launched and opens the document, the UDM will be linked to the active session.

Connecting to an Active Discovery Session

If one or more Discovery sessions are running on the machine, Discovery link can connect to them to import geometry.

When you select **Modeler** > **Discovery Link** > **Connect to Active Session** or **Connect to Discovery** in the Model ribbon, the Active Discovery Sessions dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



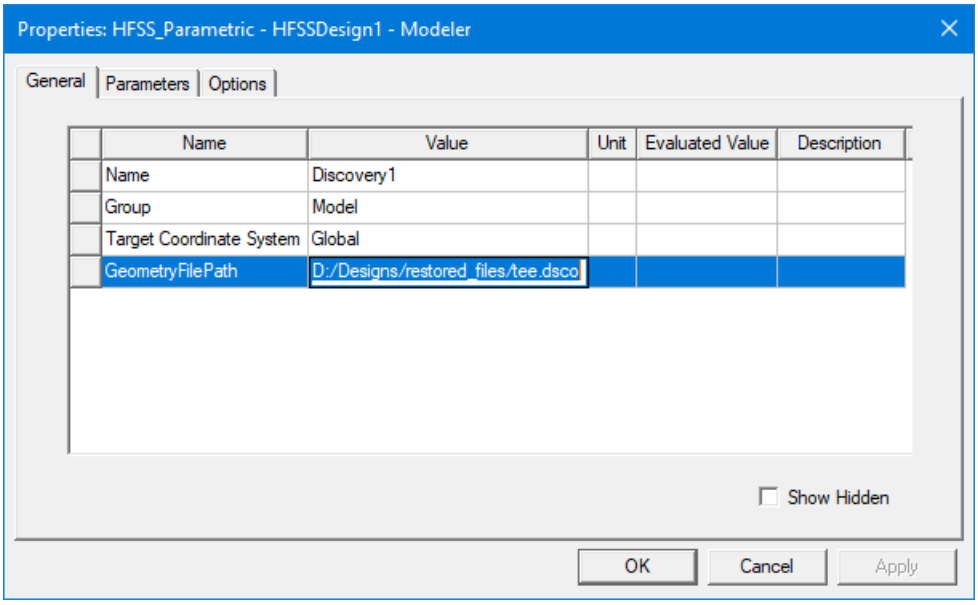
It shows all active Discovery sessions with their document names and connection status. Sessions which are not using a saved document are not included. You can select a session and click **Connect** to import geometry from that session. Once connected, that session will be used to regenerate the corresponding Discovery link UDM.

Property Window of Discovery Link UDM

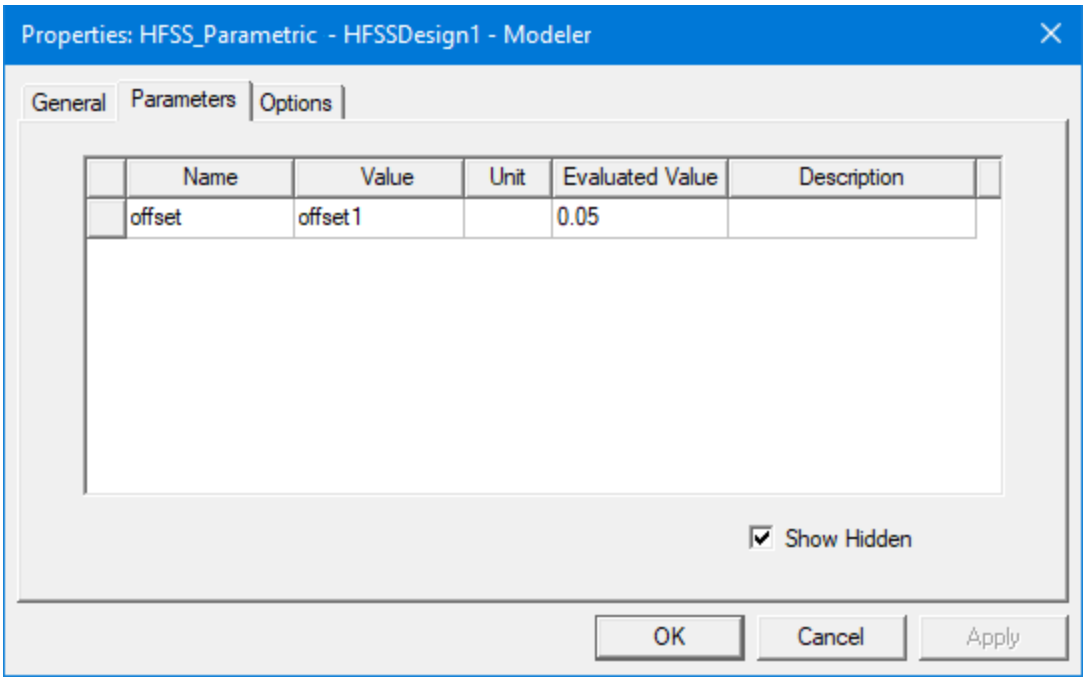
The Property window of a Discovery link UDM has three tabs:

- The **General** tab shows general information such as Discovery document path. Note that the Geometry File Path is editable; users can update the path if the original file is moved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The **Parameters** tab shows all parameters.

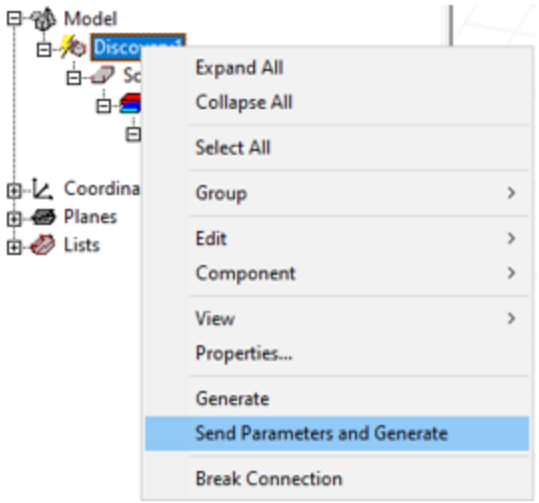


You can edit the parameters and send them back to Discovery to get the geometry updated accordingly.

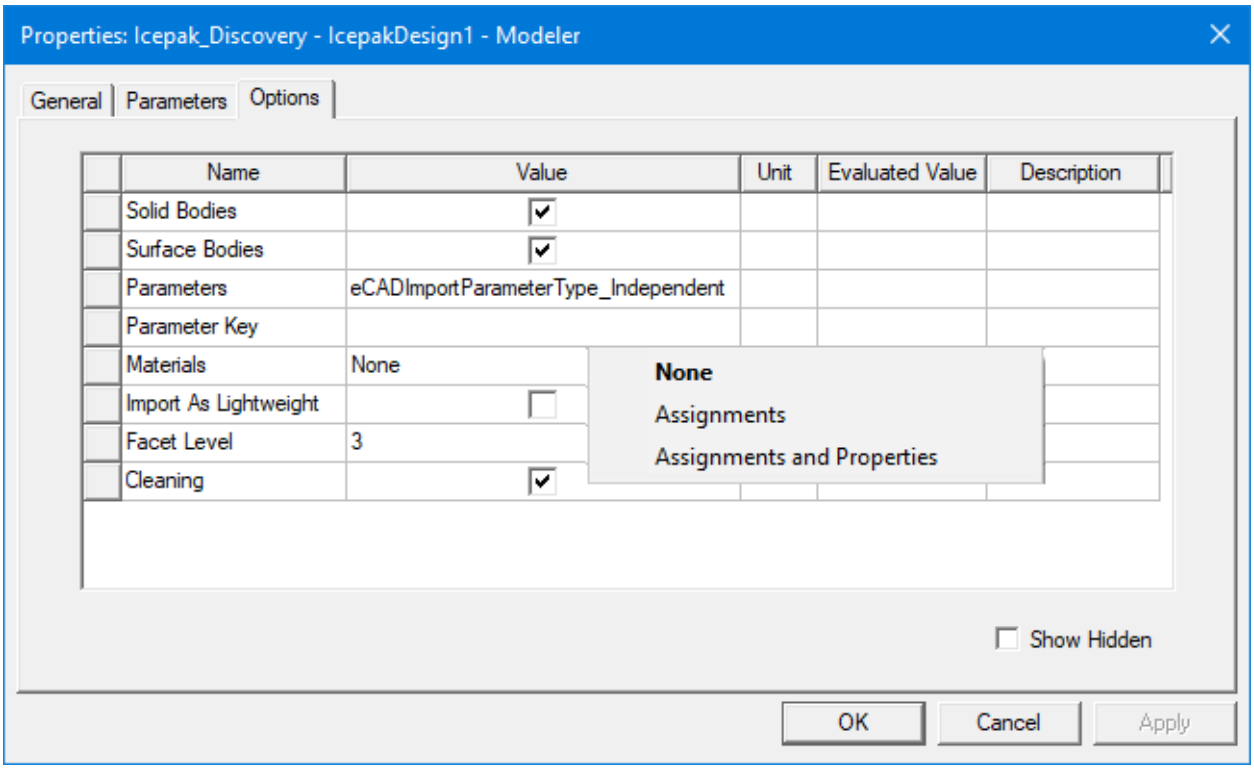
After editing the parameter, you will need to use **Send Parameters and Generate** in the context menu of the Discovery link UDM. If there is an active Discovery session linked, its

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model will be updated but not saved. If there is not an active Discovery session linked, the Discovery document will not be updated.



- The **Options** tab shows supported options. Note that Materials**Assignments and Properties** is only available for Icepak and Mechanical designs. See Discovery Link Options for details on the options in this window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Archive a Project Containing Discovery Link

If you want to share a project with Discovery link with other users, you need to archive the project, because the linked Discovery file is not included in the project file. When archiving, the Discovery documents corresponding to the links will be included in the project archive automatically. Once the archive file is unarchived, referenced Discovery documents will be put in restored_files subfolder in the project directory. All definition paths are updated to refer to the reference files in restored_files folder.

Break Connection with Discovery Link

A **Break Connection** command is available in the context menu of the Discovery link UDM.

- **Break Connection** converts the UDM bodies to imported objects. The UDM will be removed.
- Any operations that you have performed on UDM bodies are preserved.
- Parameters of the UDM are lost. You lose the ability to create parametric variations with Discovery parameters.
- Attributes such as rendering attributes, materials, etc., are preserved.

SpaceClaim Integration

As of 2025R1, the direct link between the Ansys Electronics Desktop software and the SpaceClaim application has been discontinued. Users are encouraged to convert SpaceClaim documents to the Discovery application file format and to use the Discovery link (**Modeler** > **Discovery Link**) to import geometry into AEDT.

Legacy AEDT projects containing SpaceClaim links can be opened. Any regeneration of SpaceClaim geometry will open the SpaceClaim file in background session of the Discovery application. We recommend that users convert source *.scdoc* files to *.disco* files by opening and saving them in the Discovery application. Users can then modify the source file path under the UDM link to point to the Discovery file.

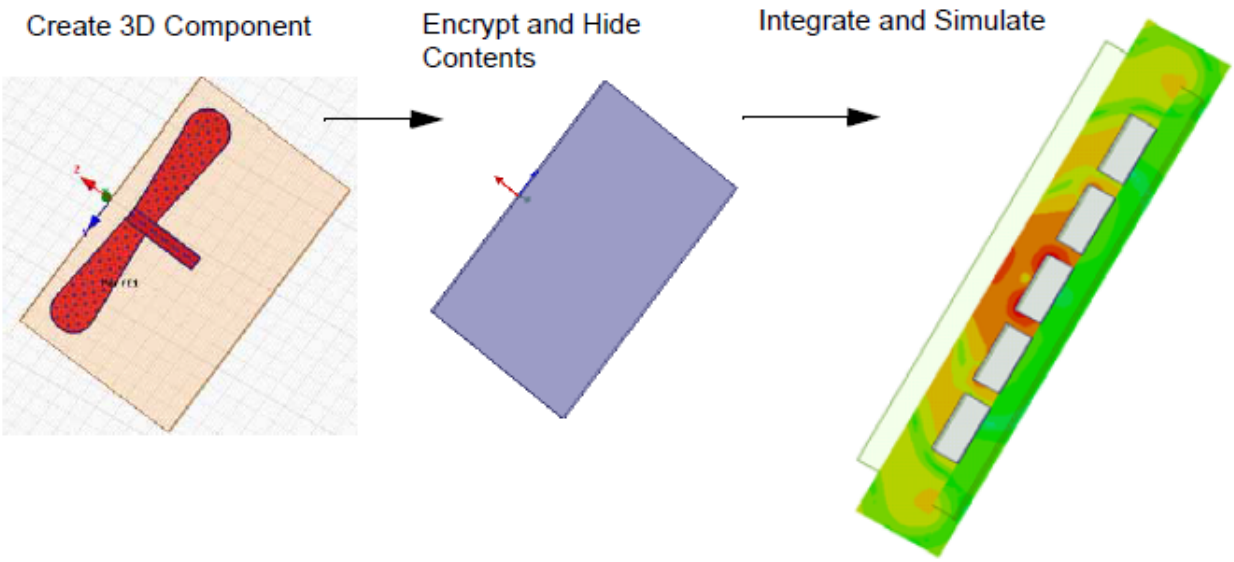
3D Component Library

The 3D Modeler lets you access predefined 3D component libraries and to create 3D components using:

- Geometry (objects and coordinate systems)
- Design data (boundaries, excitations and mesh operations)
- Material assignment and definition for components
- Component parameter, including variables used by geometries, design data, and material properties

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optional logo or icon image to display with components in the modeler window
- Optional encryption with password definition or internal key as a password



3D Component Libraries may be included as part of Electronics Desktop, downloaded from third-party vendors, or created from scratch.

The ability to create components from models lets you easily share and reuse components. You can add components to a library and share components with other users. Once you insert a component into the target design, you can directly manipulate whole components using **Arrange** operations such as **Move**, **Rotate** and **Mirror** or **Duplicate** operations like **Duplicate Along Line**, **Duplicate Around Axis** and **Duplicate Mirror**.

The documentation includes the following sections:

Creating a 3D Component from an Existing Model

Replace Selection with 3D Component

Layout Component in HFSS 3D (Beta)

Inserting a Component into a Design

Using the Component Libraries Window

Viewing Properties of 3D ComponentsViewing

3D Components in the Project Manager Window

3D Components in the History Tree

Editing 3D Component Properties

Editing a 3D Component Definition

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update 3D Component Definition

Additional **Show**, **Show Only**, **Hide** and **Fit** features, helpful in working with 3D components are described here:

Fit All Objects in a View Window.

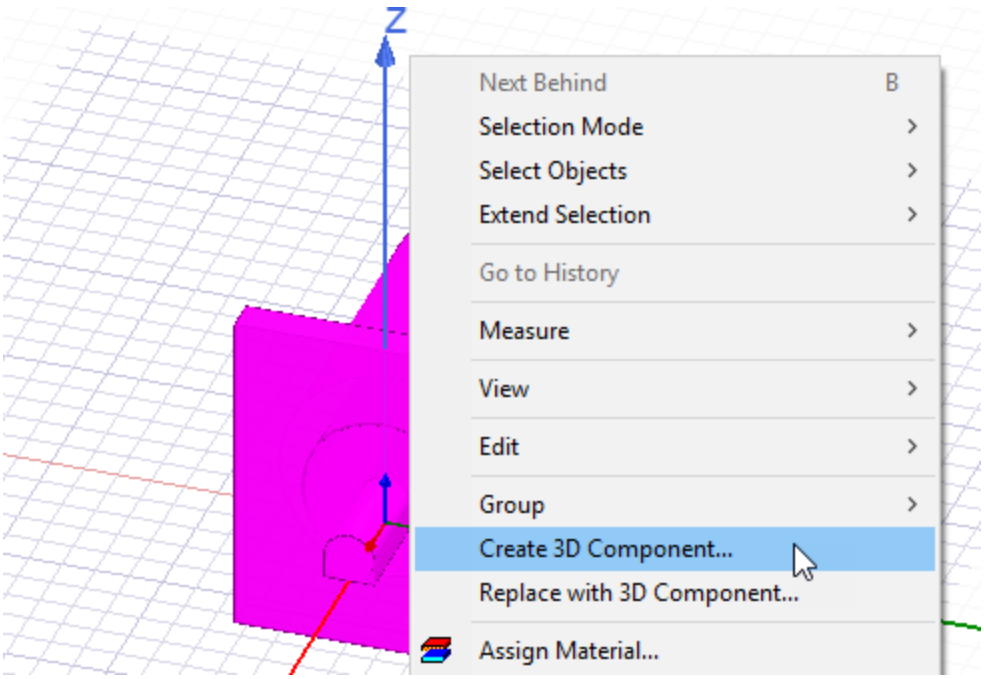
Fit Selected objects in a View Window.

Showing Only Selected Objects in All or Active Views

Hiding Objects from View

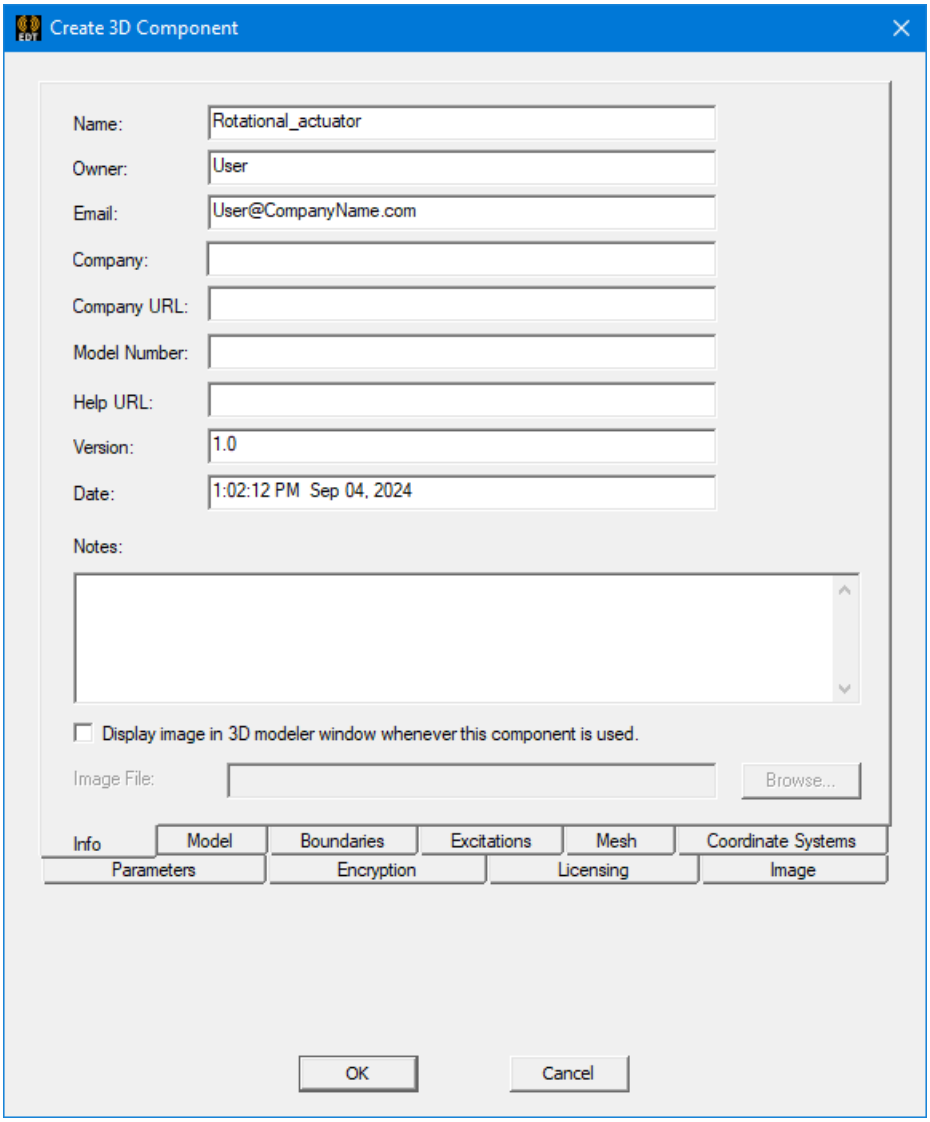
Creating a 3D Component from an Existing Model

To create a 3D component, select an existing object or model, right-click, and select **Create 3D Component**. You can also click **Draw** > **3D Component Library** > **Create 3D Component**. You can also go to the **Model** tab of the ribbon and select **Create 3D Component**.



This opens the *Create 3D Component* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The tabs let you view the component features organized according to geometry, design data, and settings:

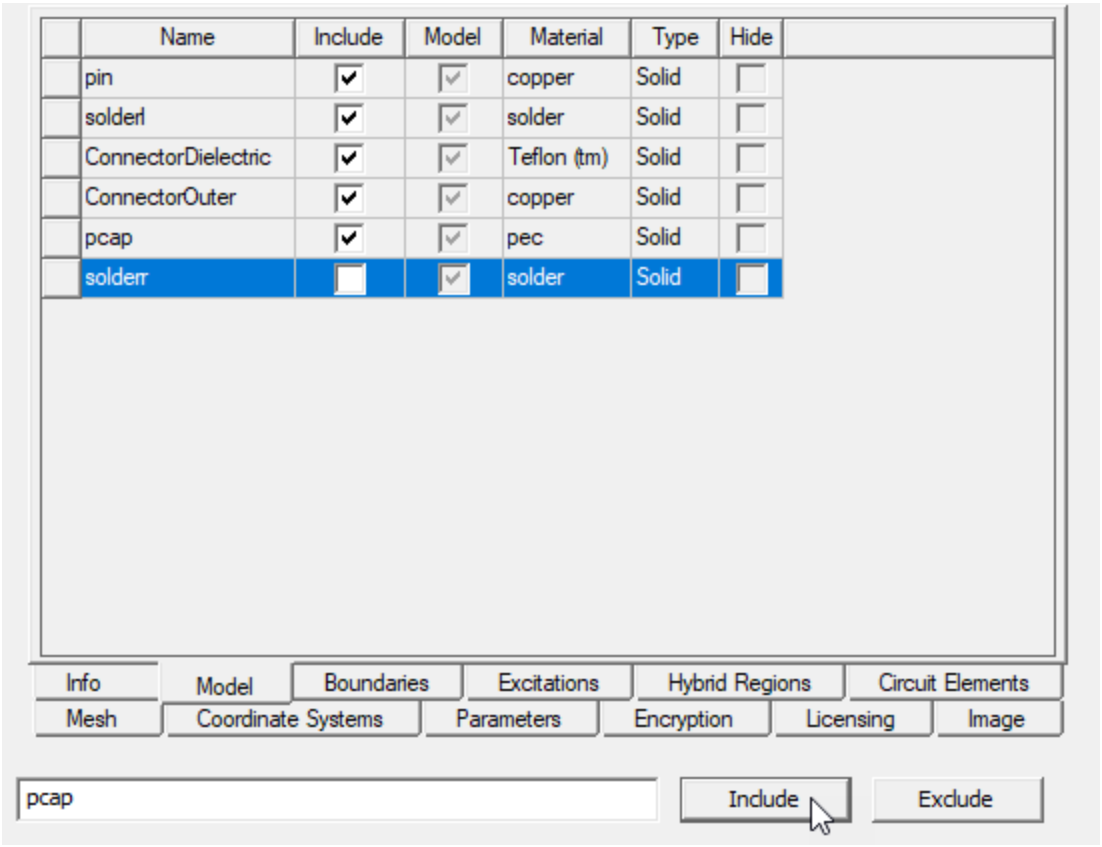
- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Circuit Elements tab
- Mesh tab
- Coordinate Systems tab
- Parameters tab

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Encryption tab
- Licensing Tab
- Image tab

Include or Exclude Feature from Component

You can view the various tabs and select which features to **Include** or **Exclude** in creating the component. You can use the check boxes in the Include column for this purpose. You can also use the text field. Type a feature Name which you can then use the command buttons to Include or Exclude.



Object selection for **Include** or **Exclude** impacts selection in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Data Notes:

- The **Create 3D Component** dialog does not list design data without an assignment.
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Save 3D Component File. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included. After validation, you see a dialog for saving the component to a location.

Create 3D Component: Model Tab

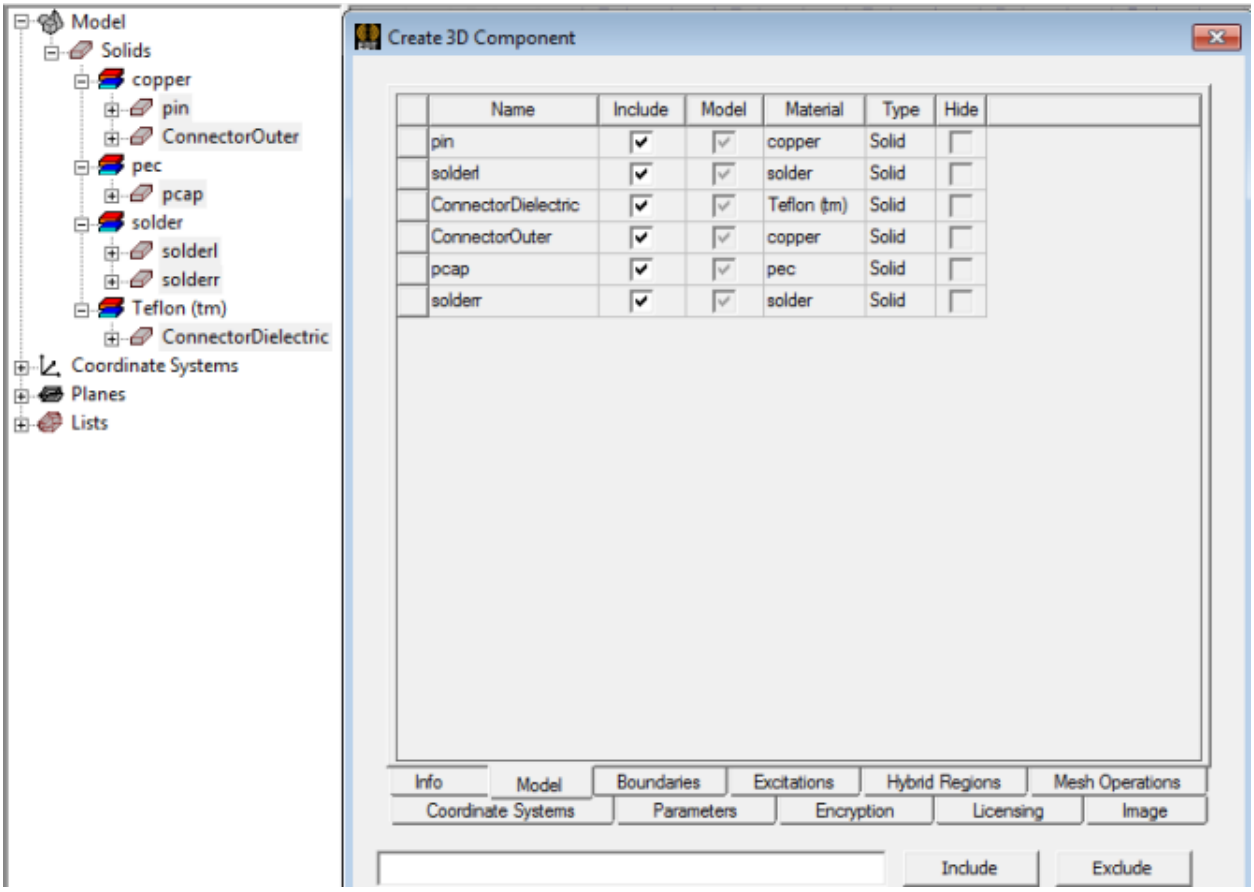
The **Model** tab lists all of objects selected before you invoked **Create 3D Component**. These correspond to the selected objects listed in the history tree.

- Use the **Include** check box to include objects in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an object **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple objects with similar names in a single operation.

- The **Material** and **Type** columns are read only.
- For Encrypted components, you can also chose which objects to hide.
- The **Hide** column is only enabled if you have the **Hide Selected Contents** option selected on **Encryption** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Boundaries Tab

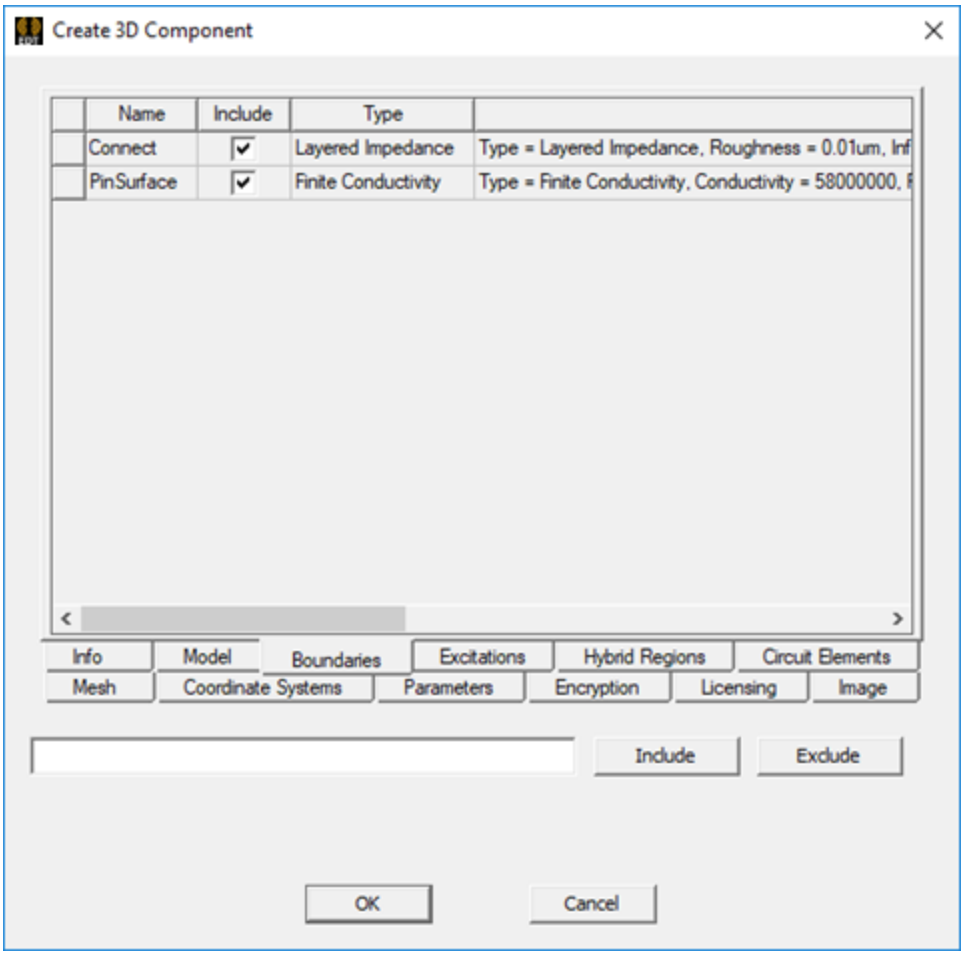
The list is populated with all the boundaries on selected objects.

- Use the **Include** check boxes to include (selected) or exclude (cleared) individual boundaries for the 3D component.

  Alternatively, you can type a boundary *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple boundaries with similar names in a single operation.

- The *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Excitations Tab

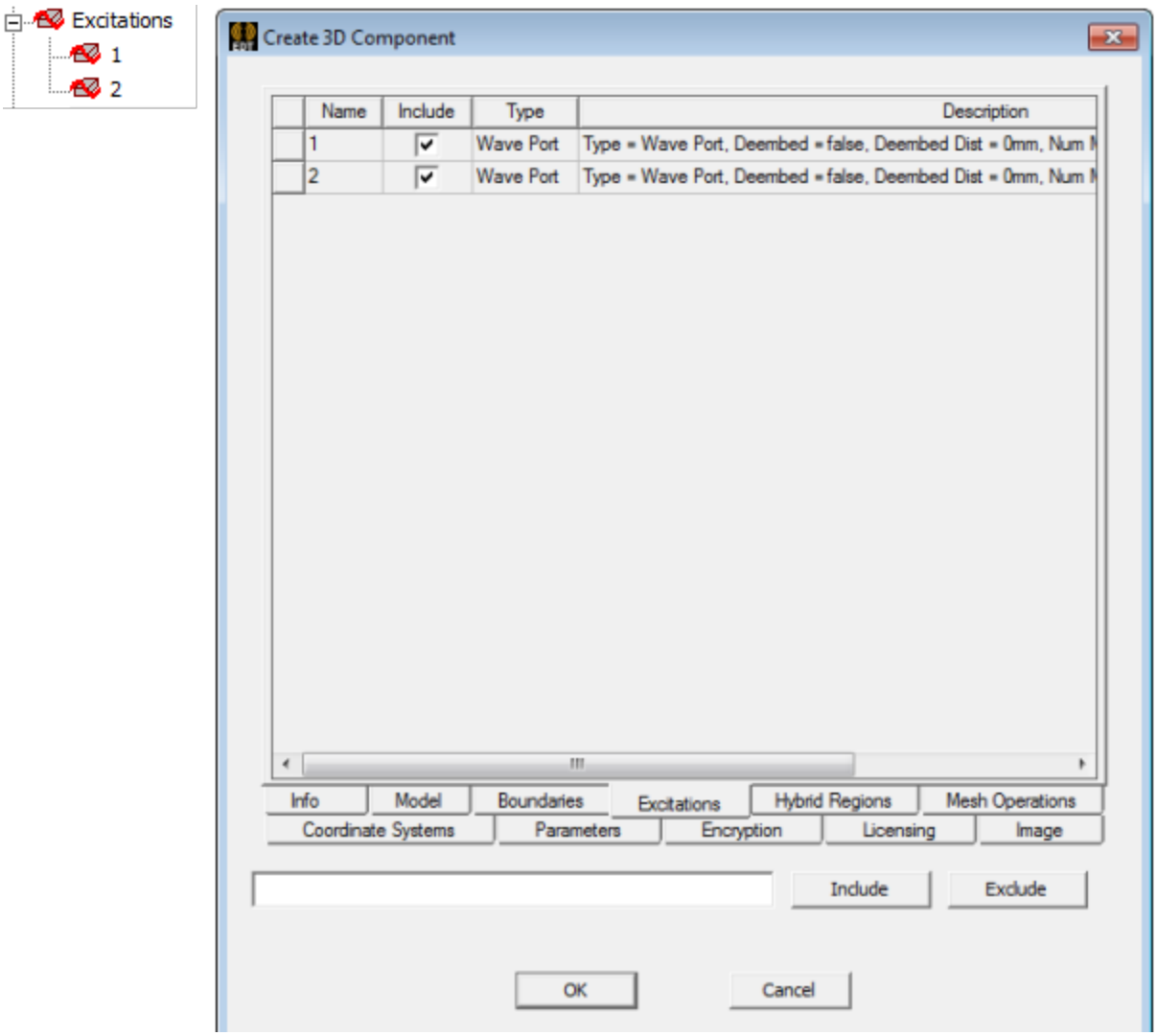
The list is populated with all the excitations on selected objects.

- Use the **Include** check box to include excitations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an excitation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple excitations with similar names in a single operation.

- *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Mesh Tab

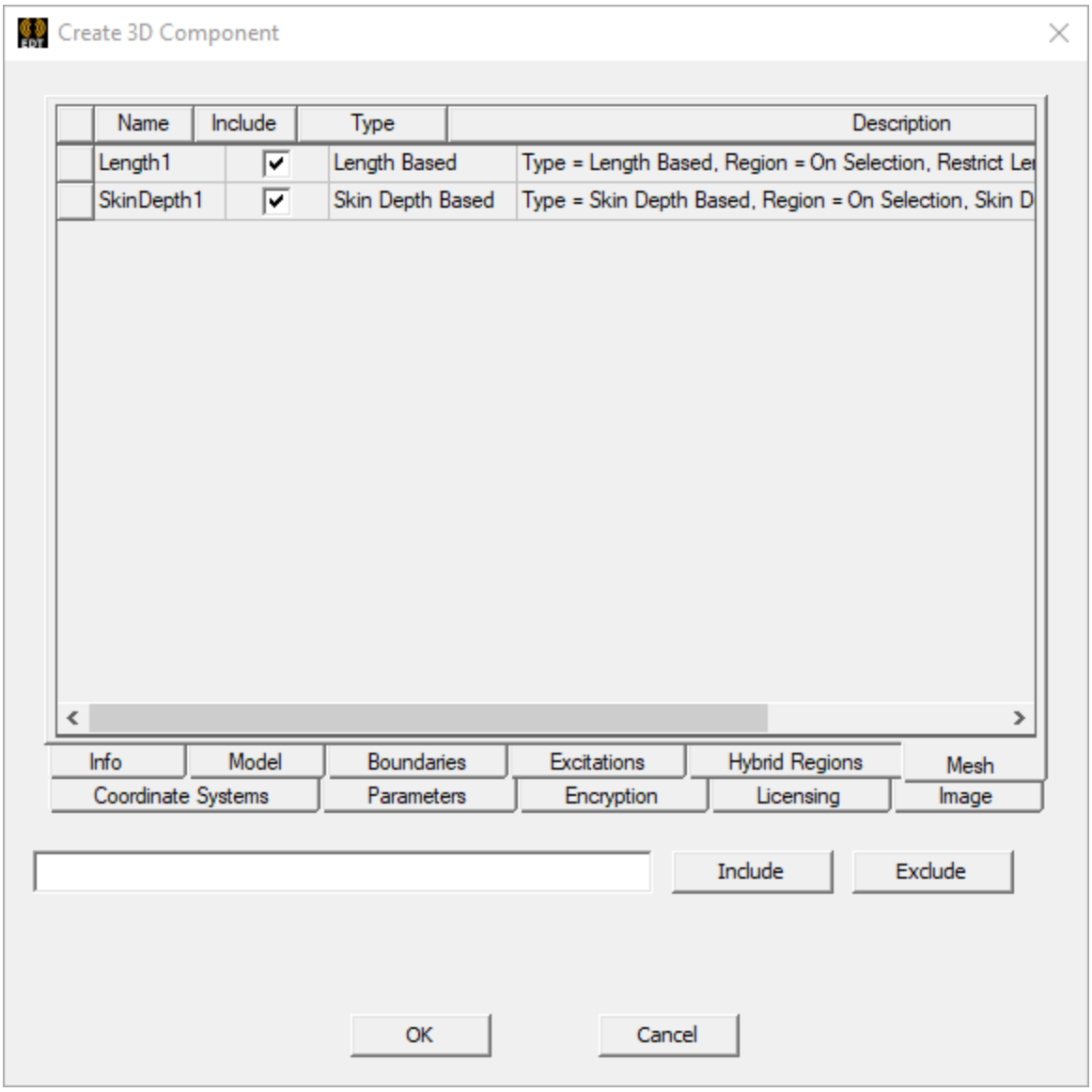
The list is populated with all the mesh operations on selected objects.

- Use the **Include** check box to include mesh operations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type a mesh operation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple mesh operations with similar names in a single operation.

- *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Coordinate Systems Tab

The list is populated with all the Coordinate Systems on selected objects. By default, only the Coordinate Systems used to define the object orientations are included. You can include Coordinate Systems created on specific model parts.

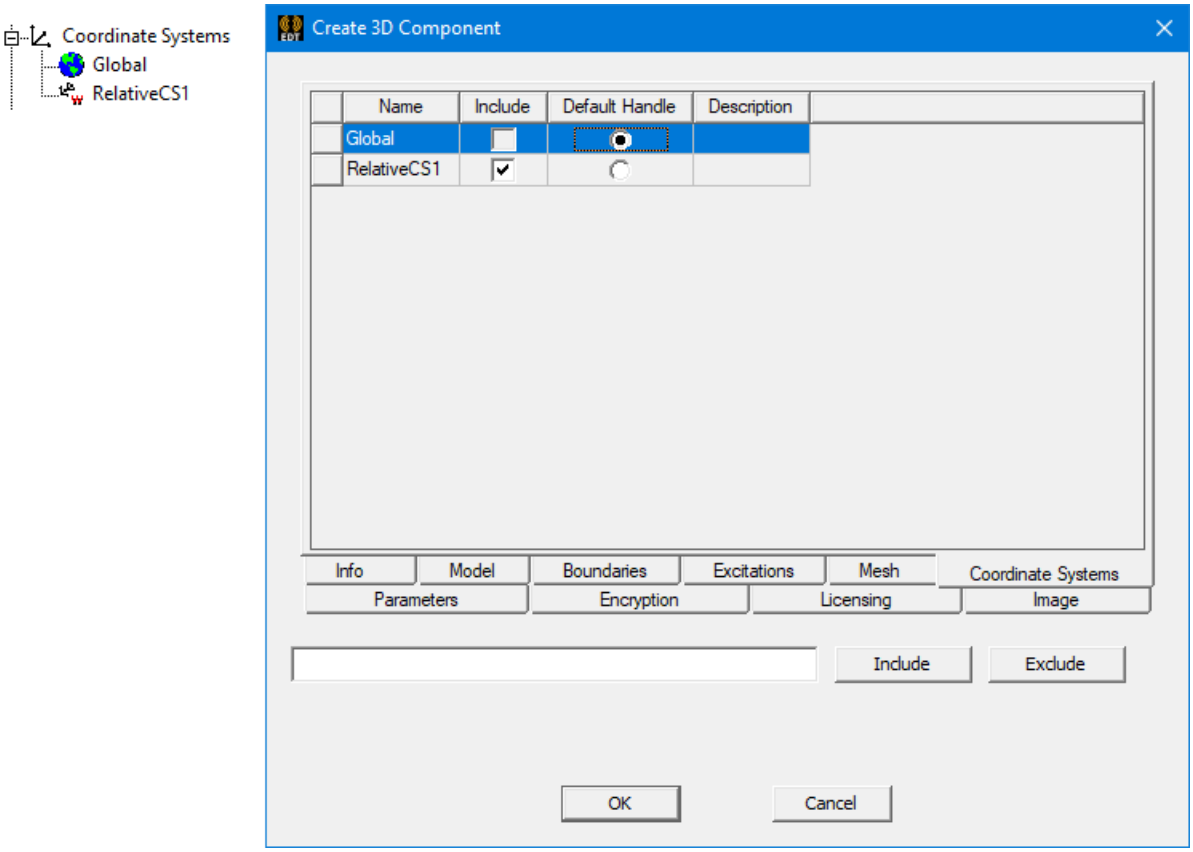
- Use the **Include** check box to include coordinate systems in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type a coordinate system *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

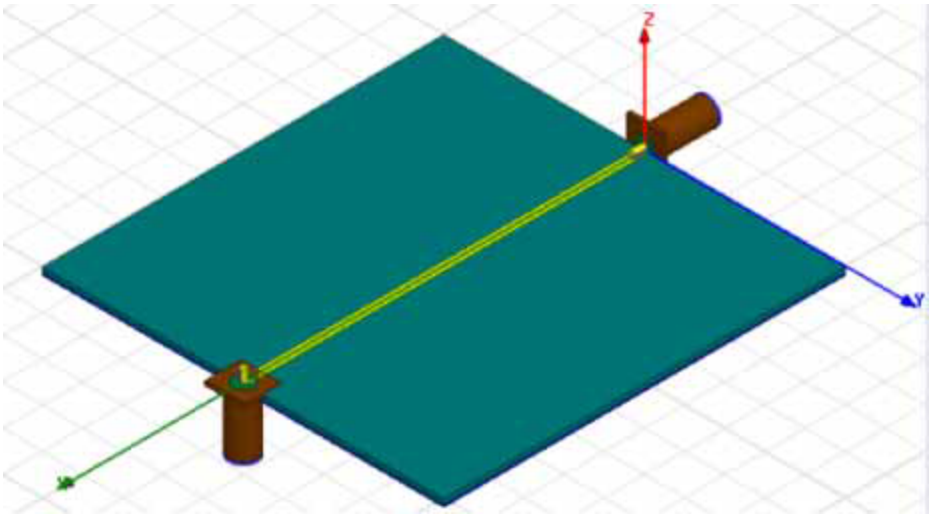
so you can change the inclusion state of multiple coordinate systems with similar names in a single operation.

- You can select any of the CS as the default handle. When this component is placed in assembly design, the handle CS is aligned to target CS in assembly design. Note that the handle CS can be changed after import; see Editing 3D Component Properties for more details.
- By default, the current working CS is designated as the default handle CS.
- The default handle CS must be included in the model (except for global).



The specified default handle affects the orientation of the component upon insertion:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Parameters Tab

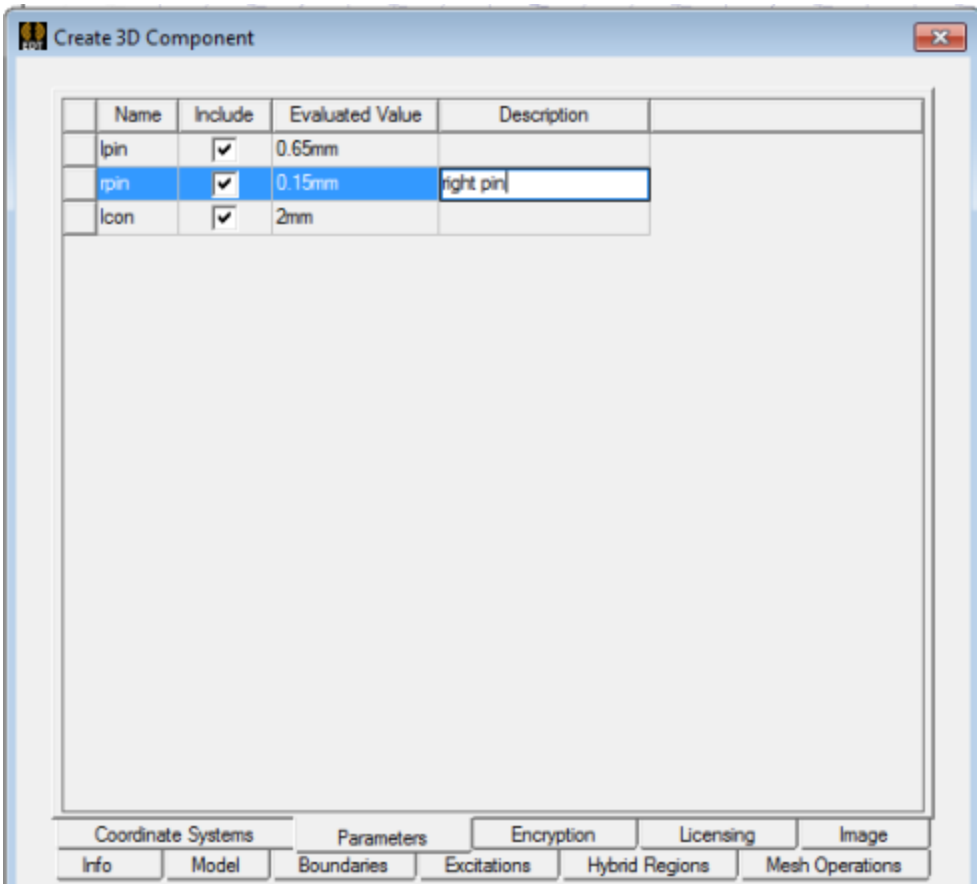
Component properties can be parameterized by assigning variables. The parameters list is populated with all the variables used by objects and coordinate systems, design data, and material properties included in the component.

- Use the **Include** check box to include parameters in the 3D component (selected), or to exclude them from the component (cleared).

  Alternatively, you can type a parameter **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple parameters with similar names in a single operation.

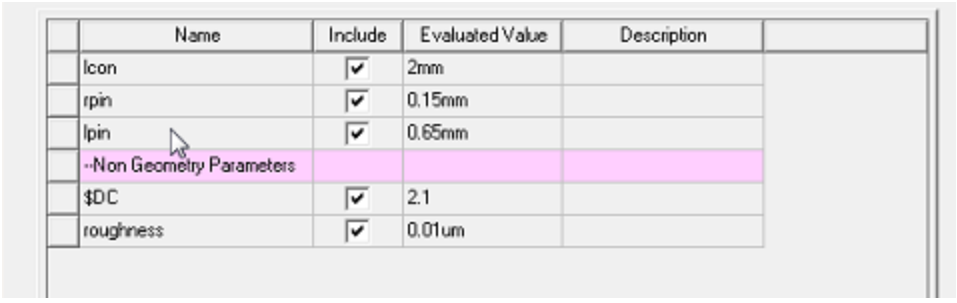
- You can fill in the **Description** field, if desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Non-Geometry Variables in 3D Components

If you create non-geometry variables materials or boundaries, these are listed separately:

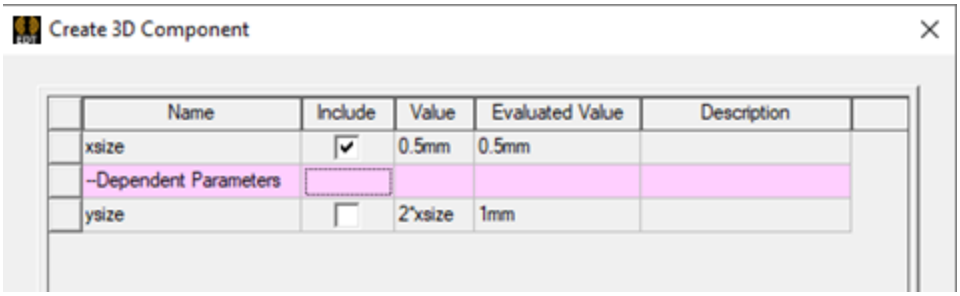


If you exclude any geometry, boundary, or material associated with a variable, the associated variable is automatically excluded on the **Parameters** tab.

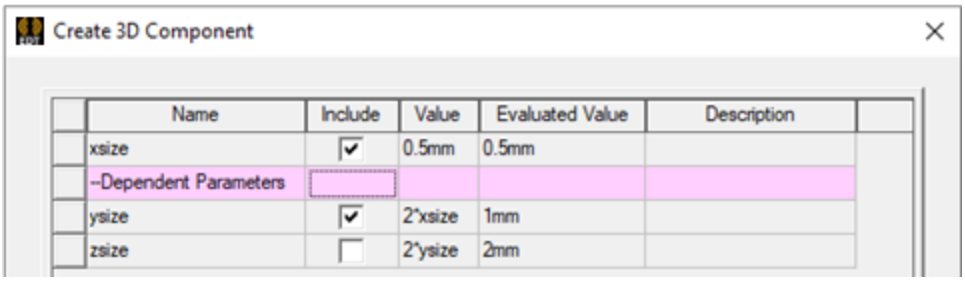
Dependent Variables in 3D Components

If you create dependent variables, you can choose to display the values.

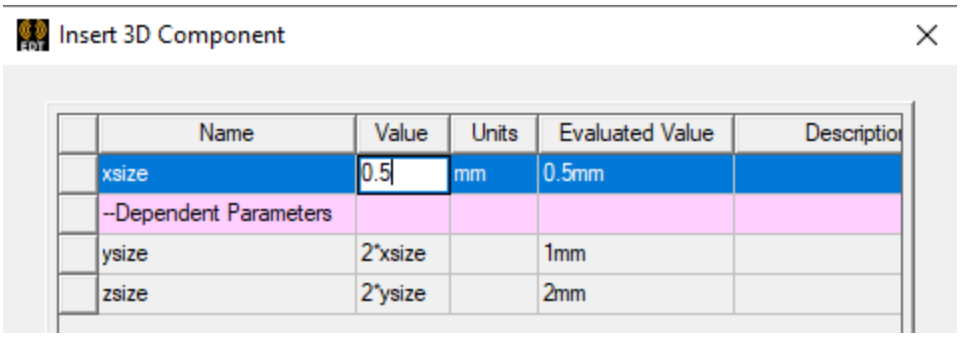
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If more variable dependency occurs for an Included variable, such variables are also displayed with the option to include.



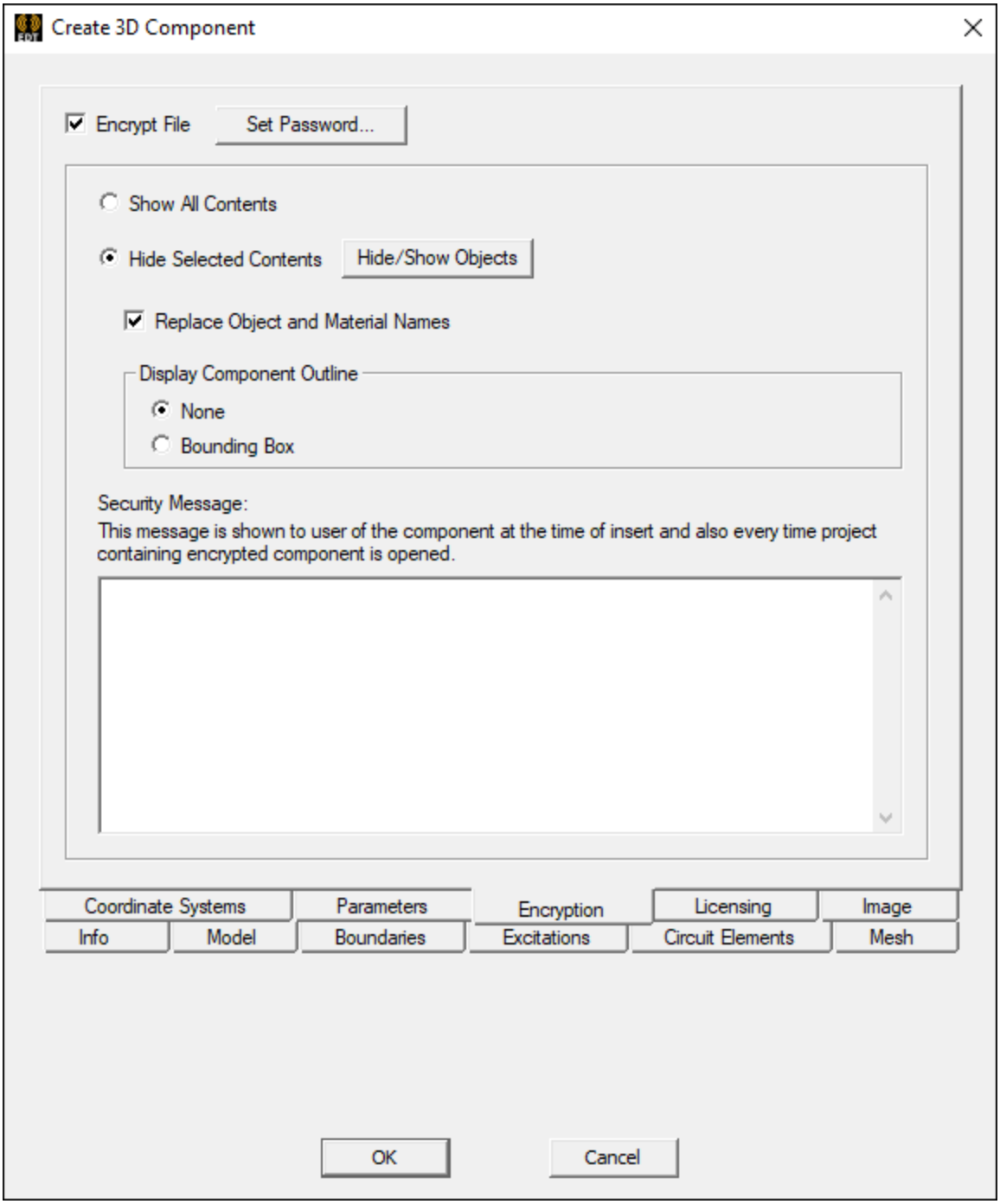
When you Insert a 3D Component with dependent Parameters included, you can edit the Independent variables, but can only view the evaluated values of the Dependent Parameters.



Create 3D Component Encryption Tab

The **Encryption** tab contains options for 3D component encryption.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting the **Encrypt File** check box enables all options and automatically launches the **Set Passwords** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Encryption is a common technique for encoding information in a manner that is very difficult for unauthorized users to decode. When this option is enabled for 3D components, the contents of the component file are encrypted to protect them from external viewing or editing.

- 3D components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.
- If the encrypted component is created with an **Internal Key**, it can be used without additional authorization.
- If the encrypted component is created with a **User Password**, that password must be entered when the component is used in a design.
- It is not possible to edit the definition of an encrypted component within Electronics Desktop unless **Allow Component Edit with User Password** was selected when defining the component.
- If the **Show All Contents** option is selected, the geometry and other contents will be shown when the component is used. Geometry export and links will operate as for general geometry.
- The **Hide Selected Contents** option can be used to further protect access to an encrypted component's details. When this option is selected, hidden contents will not appear in the user interface.
  - All geometry will be hidden, except for objects that the user specifies as visible during component creation.
  - Hidden geometry will not be visible in the model window or included in the history tree.
  - Geometry export and links will not be allowed.
  - Materials, boundaries, and other setup data will also be hidden, with limited exceptions such as excitation settings that must be accessed in order to control the simulation and post-processing of the design.
  - For components with hidden contents, all geometry will be excluded from field calculations and from plots of fields and meshes. Plots in surrounding volumes will extend to, and can include, the outer surfaces of the component, but not the containing volume.

    > **Important:**
    >
    > Such plots may indirectly reveal component content by providing a view of the surfaces where the plots terminate. To prevent such exposure, include additional modeled objects that surround the hidden contents of the component and have material of the surrounding material environment (e.g., air, vacuum).

    See: Hiding Component Content.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using **Hide Selected Contents**, the names of objects, materials, boundaries, and other setup data may appear in progress updates, messages, and other informational text. The **Replace Object and Material Names** option can be used to automatically convert object and material names to generic names in the exported component. Other sensitive names should be changed before exporting.
- The **Security Message** field specifies text that is displayed at component insertion and when the project containing the component is opened.
- When a project contains one or more encrypted 3D components, the project file is encrypted in order to protect its contents. This encryption uses an internal key, so that the file can be opened without entering a password.

Important:

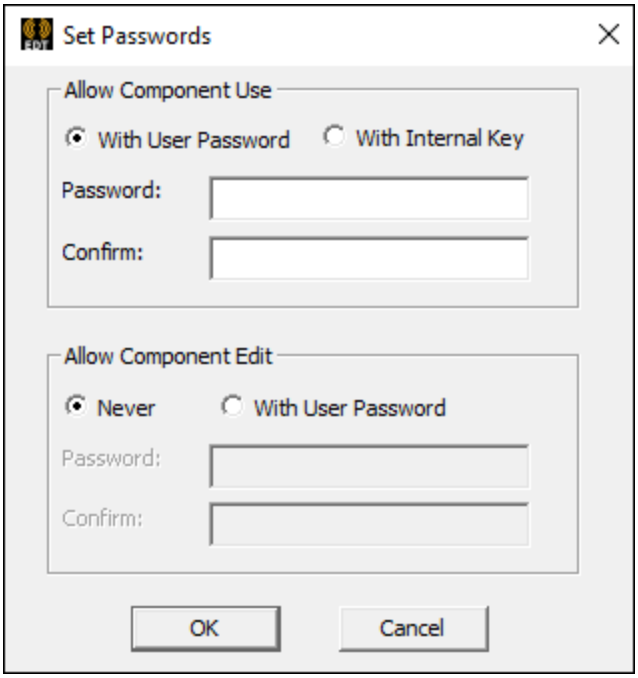
There are some additional considerations with respect to projects using encrypted components:

- The component itself will always be stored in encrypted form.
- When an HFSS design contains an encrypted component with hidden contents, it will also protect solver inputs and other project files containing data that could be used to extract component information. However, note that solver inputs associated with SBR+ regions are not protected, even when encrypted components are in use.
- For a hybrid design that contains a combination of SBR+, FEM, and IE regions: the FEM and IE regions will be fully protected, while the SBR+ regions will not.
- Other design types, such as Maxwell, Q3D, or Icepak, do not include extended protection of solver inputs and other project files.

Set Passwords

Clicking **Set Password** launches a window containing fields for setting component passwords. This is the same window that automatically launches upon enabling **Encrypt File**.

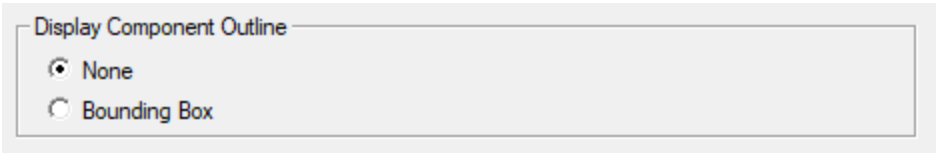
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Allow Component Use** – allows you to select between encrypting with a user password or an internal key (which requires no additional authorization). If setting a password, enter it in both the **Password** and **Confirm** fields and click **OK**.
- **Allow Component Edit** – allows you to choose between never allowing edits to components or allowing edit with a user password. If selecting the latter, enter a password in both the **Password** and **Confirm** fields and **click** OK.
- When enabling a password for both component use and component edit, the password does not have to be the same.

Hiding Component Content

The **Hide Selected Contents** option allows selected parts of the component to be displayed either as a **Bounding Box** or without an outline (**None**).

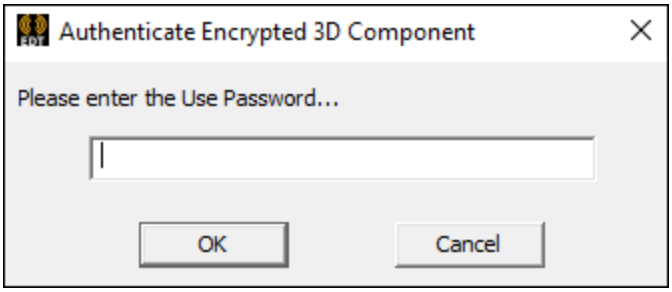


Once encryption is set, visibility can be set on a per-object basis using the **Model** tab of the **Create 3D Component** window. Preview the component using the**Image** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

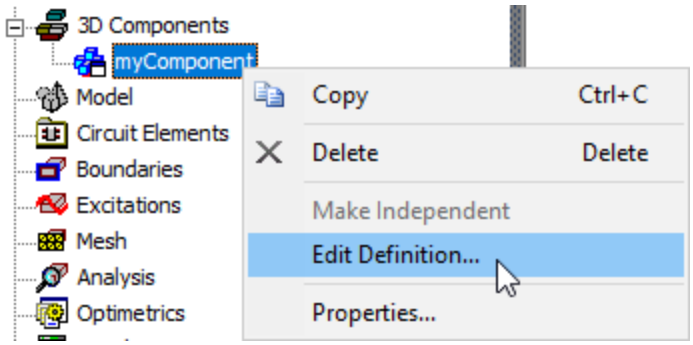
Inserting Encrypted Components

When an encrypted 3D component is added to a design, a prompt will appear asking for the password.

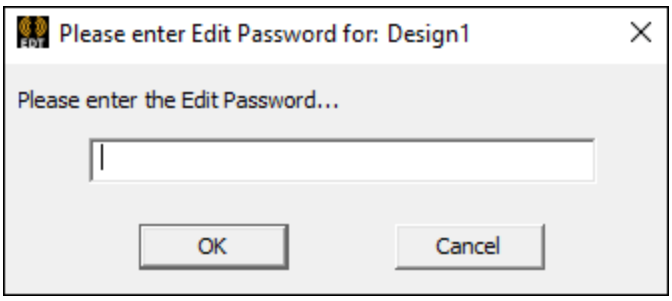


Editing Encrypted Components

3D component definitions can be edited from the **Project Manager**:

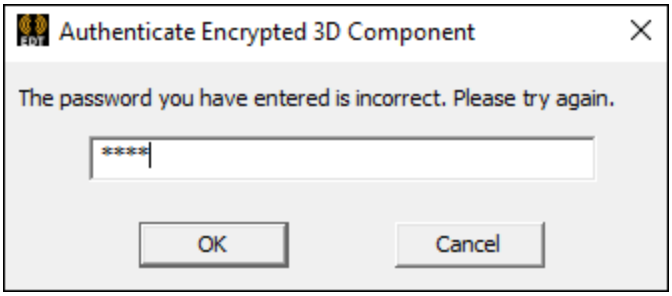


A prompt appears asking for the password:

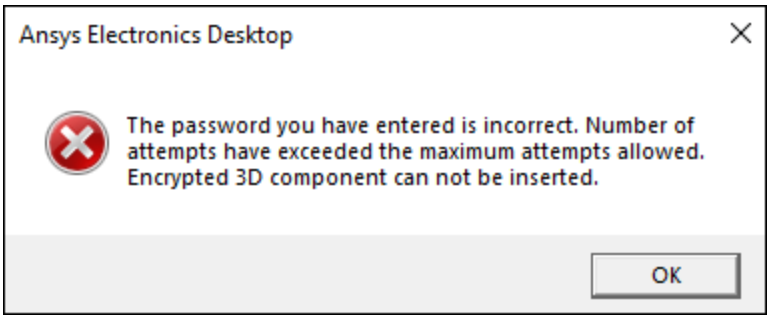


If the password is incorrect, a warning appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

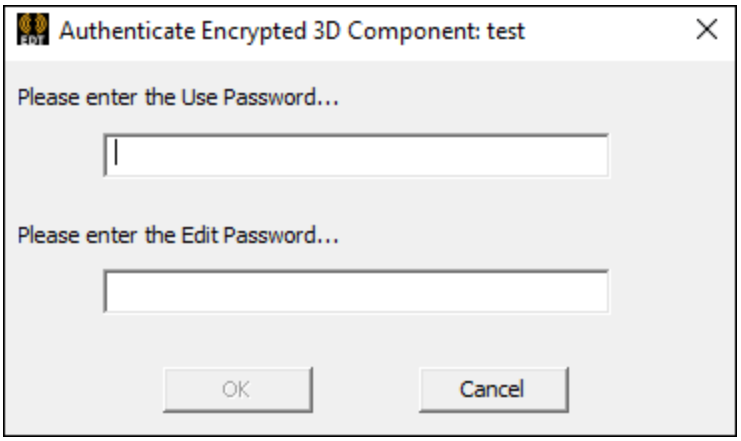


There is a three attempt maximum, after which a warning will appear and no further attempts may be made:



Opening a 3D Component File

When opening an encrypted 3D component file using **File** > **Open**, both the use password and edit password must be entered:



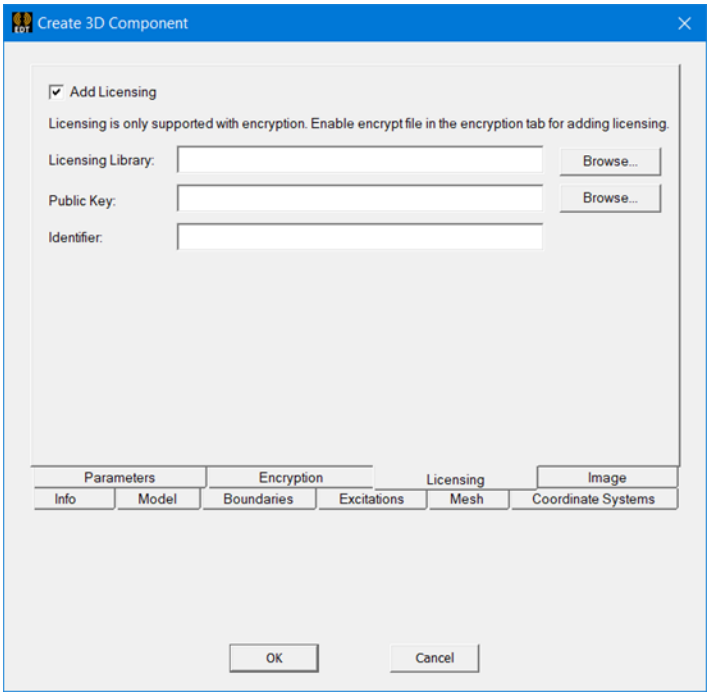
Copying and Pasting Encrypted Components

Encrypted components may be Copied and Pasted. They will be remain encrypted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create 3D Component Licensing Tab

A 3D Component can be licensed using an external licensing library. This feature is for those who want to license and distribute their 3D Component models. On the **Licensing** tab, you specify a 3D Component Licensing dll, a public key and an optional component identifier. You can license a 3D Component only if it is also encrypted.

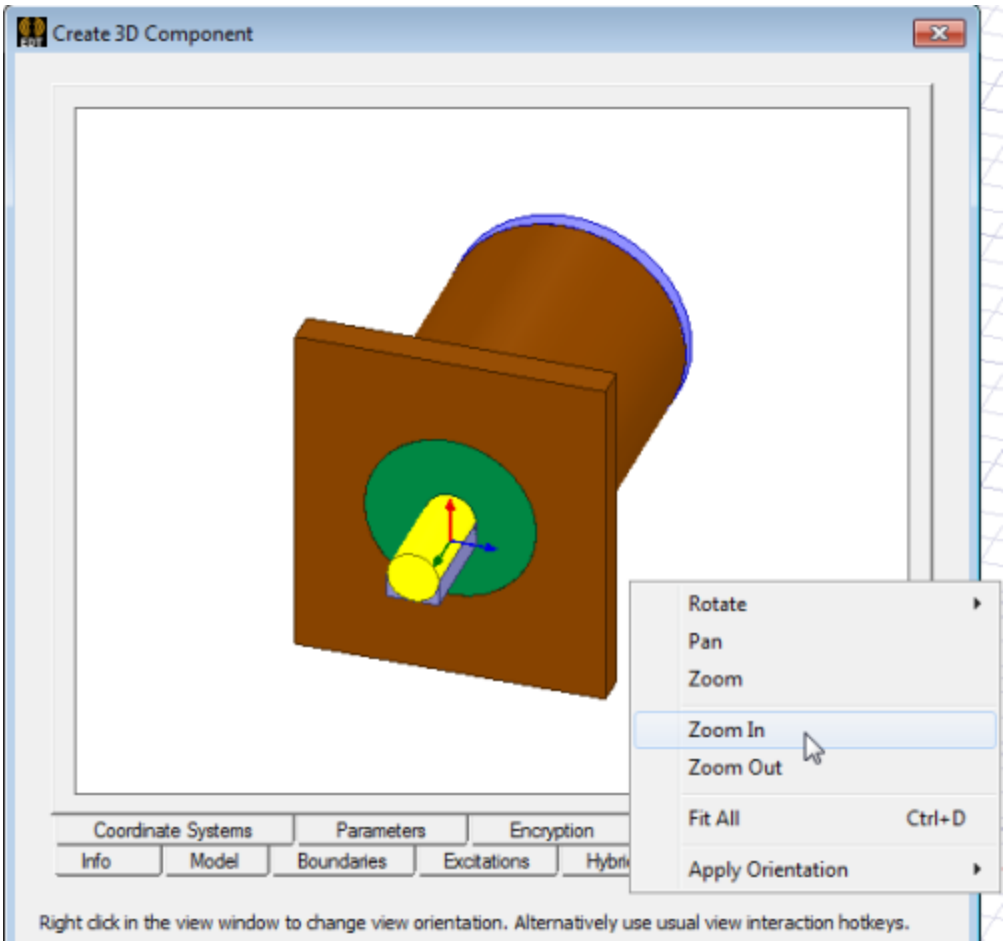


Create 3D Component Image Tab

The **Image** tab shows the preview image for the component, based on the reference Coordinate System. The image also responds when you **Include** or **Exclude** objects from the **Model** tab.

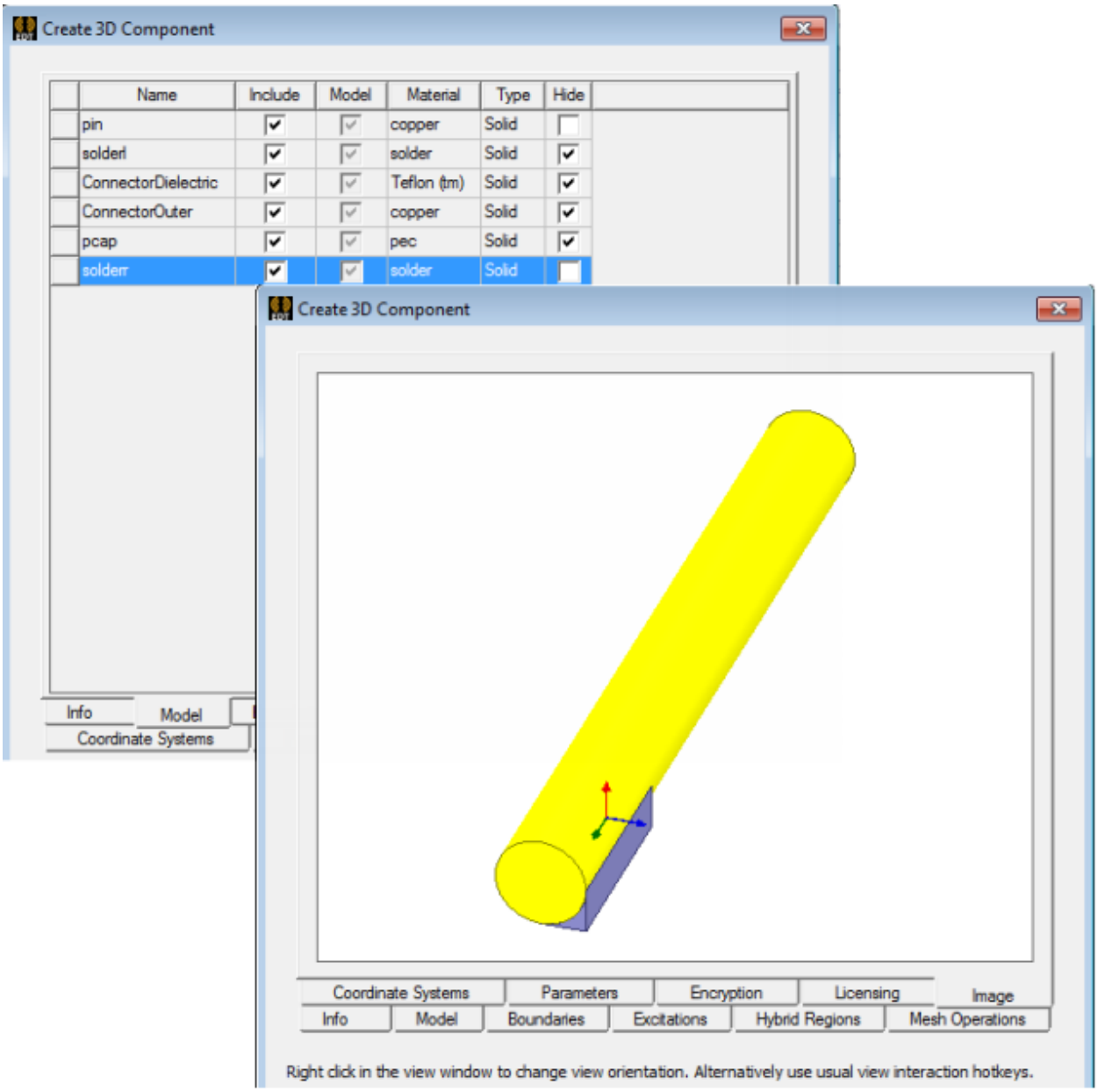
- You can right-click for a shortcut menu to change the view of the preview image.
- Changing the preview orientation does not affect the modeler window view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



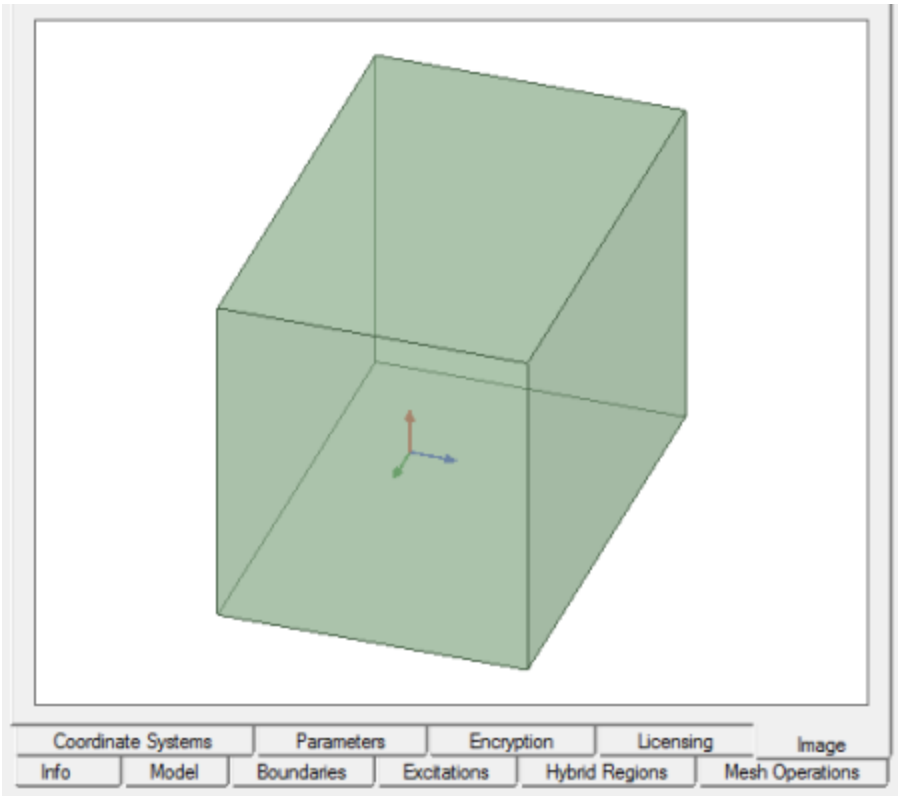
If you use the **Encryption** tab and **Model** tab settings to hide/show the model or specified model objects, the **Image** tab responds. You must use encryption to enable the Hide check boxes on the **Model** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you use the **Encryption** tab to display only an object outline, the **Image** tab responds.

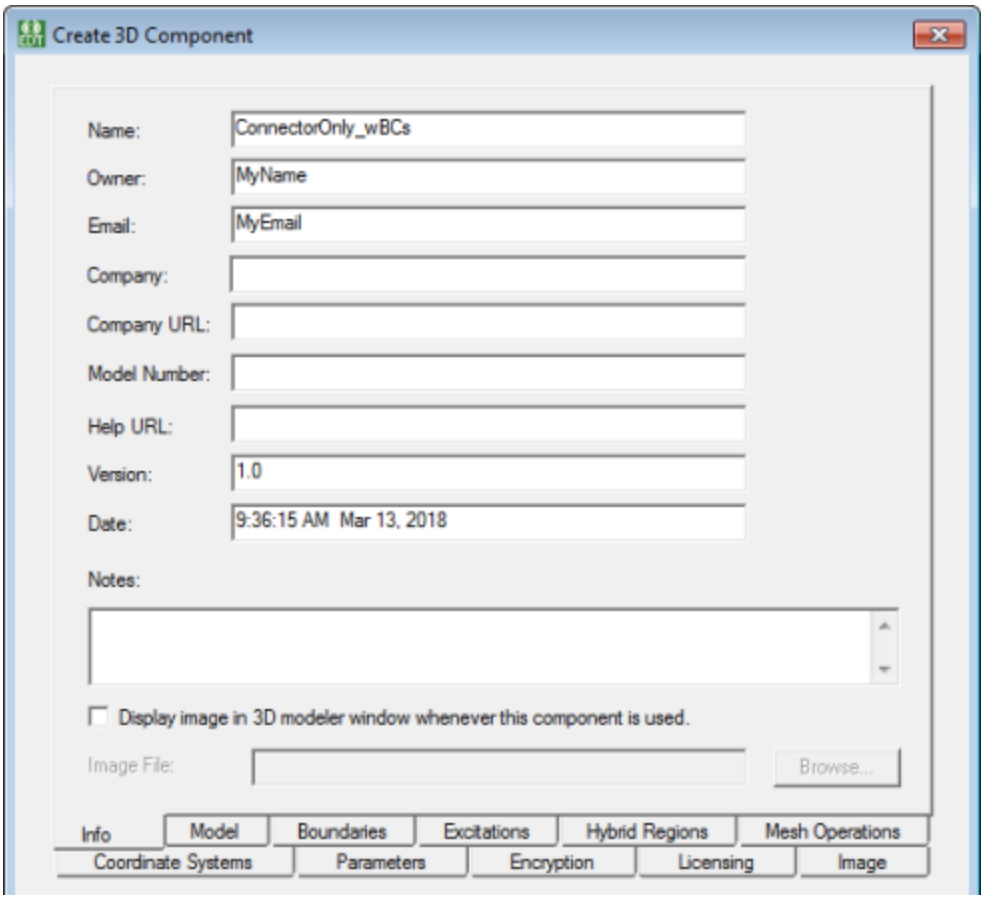
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



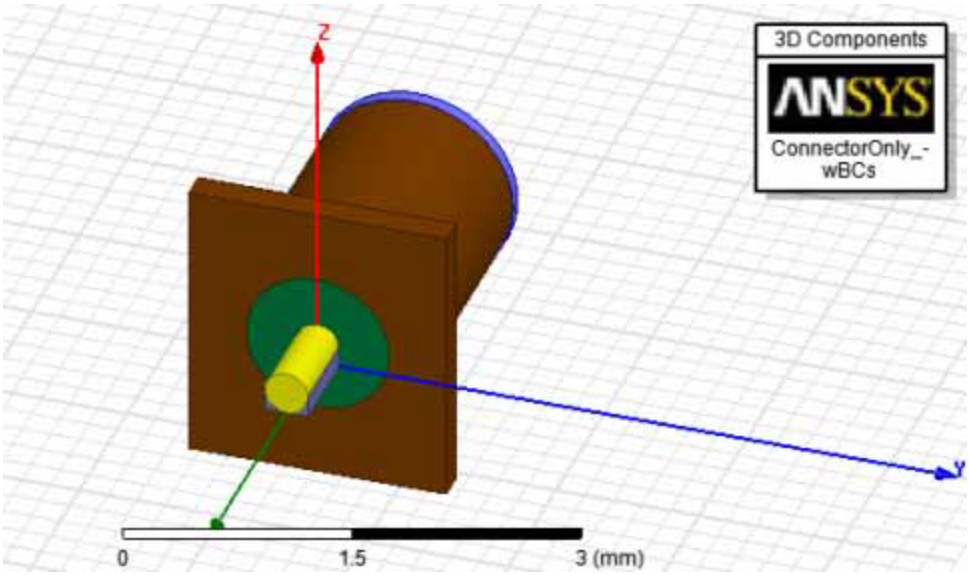
Create 3D Component Info Tab

On the **Info** tab you can view and edit component information, as well as add notes. The *Name* in the first field is used when the component is inserted into a design. Other fields include *Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

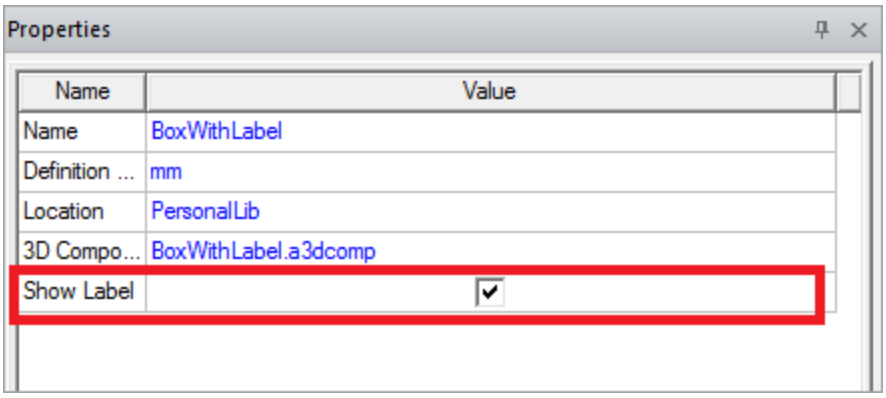


You can also included a .bmp format image file to display in the upper right area of the 3D modeler whenever the component is used. The image file typically contains a company logo. Check the **Display image** box to enable the **Image File** field, and click **Browse** to open a window to navigate your file system select the image.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

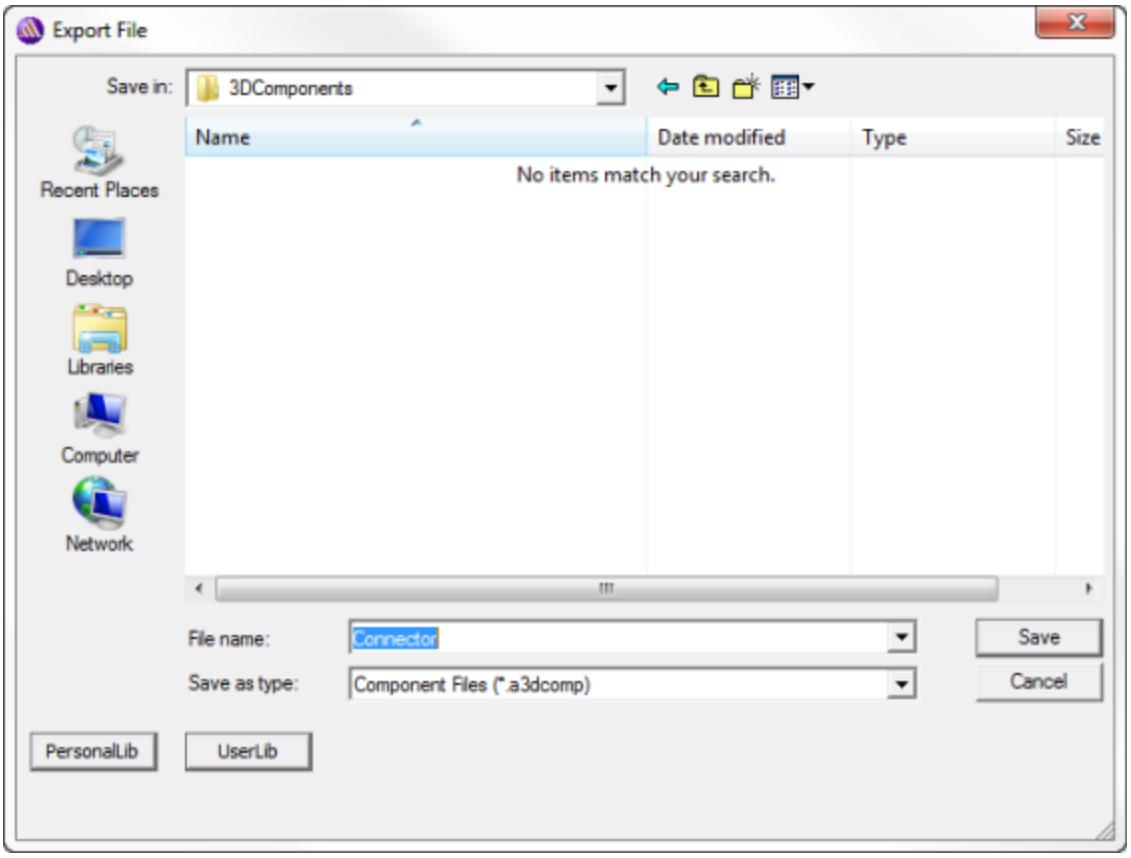
You can move the image in the modeling workspace by clicking the "3D Components" area of the box and dragging it. You can hide the image by editing the 3D component and clearing the **Display image...** check box, or by deselecting the Show Label check box in the Properties window:



Save 3D Component File

When you **OK** the *Create 3D Component* dialog box, an *Export File* dialog box displays. By default the **Save in** field shows the model source folder. Click **PersonalLib** or **UserLib** to display a 3D Components folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



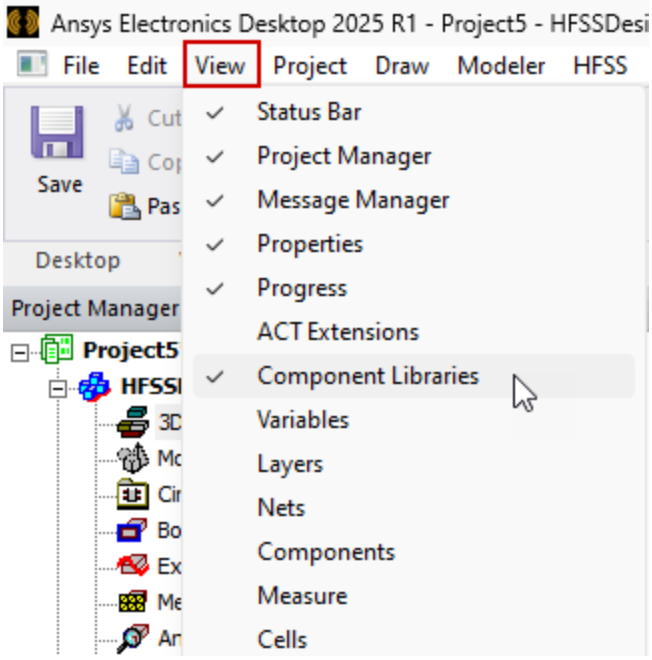
The default **File name** is the component *Name* specified in the *Info* tab.

Using the Component Libraries Window

Once you save one or more components to a library, and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D component icon ( ) in the Project Manager or, using the menu bar, click one of the **Draw** > **3D Component Library** commands to browse your folders or libraries.

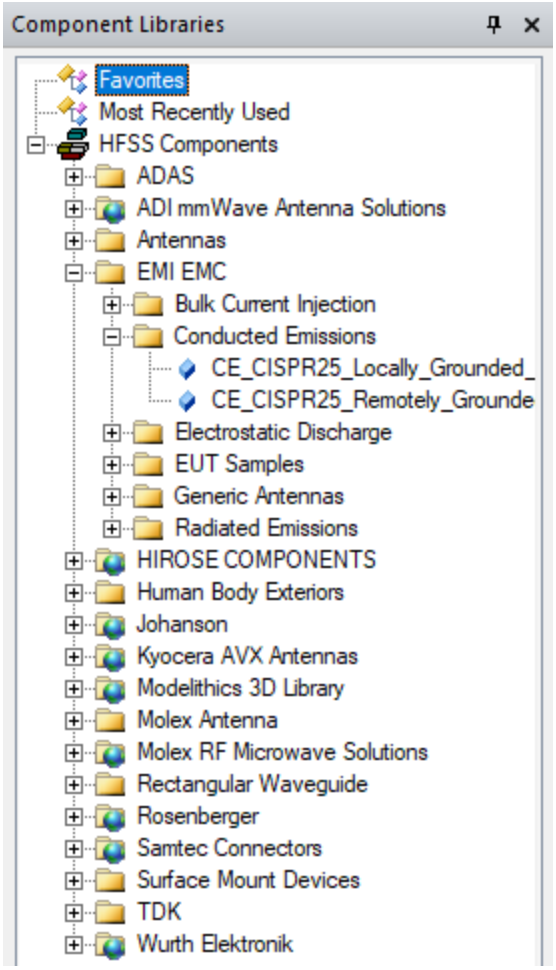
Toggle the visibility of the docked **Component Libraries** window from the menu bar by clicking **View** > **Component Libraries** and navigate the installed libraries from there. Visibility of the **Component Libraries** window is remembered for each design type so that, if context switches between different design types, the window is shown or hidden based on setting for new design type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Component Libraries** window, you can navigate the directory of installed components, as well as any in your Personal Library. The **Component Libraries** window lists all the 3D components available in syslib, UserLib and PersonalLib. It shows a folder corresponding to a library only if there are components for the active design type in that library folder. There are also additional optional folders for managing Favorites and Most Recently Used components. **Component Libraries** window contents, including Most Recently Used and Favorites correspond to the active design. When context is switched between different designs, the *Component Libraries* window will be refreshed so that it is relevant to the active design.

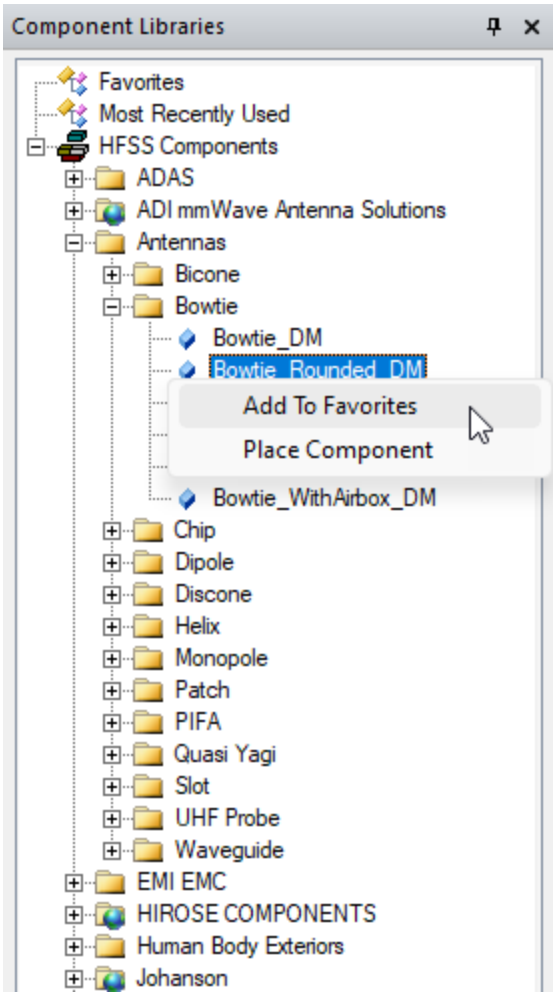
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can control whether to show the Favorites and Most Recently Used folders. Access the *Options* dialog box from the menu bar by clicking **Tools** > **Options** > **General Options**. The folder visibility options are on the General > Component Libraries Options page.

Right-click a component to view the **Add to Favorites** command for including frequently used components in the Favorites folder. Once a model is included in the Favorites folder, you can also right-click it to **Remove from Favorites**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you have added a component to a design, it also appears in the **Most Recently Used** folder. You can also right-click model there to view the **Add to Favorites** and **Place Component** commands.

You can select any component from the library, and drag and drop to insert it in the design. If you insert a component by the drag and drop method, the component is immediately inserted, skipping the **Insert Component** dialog box. The component is inserted with default component parameter values and is placed in the currently active coordinate system.

You can also select **Place Component** from the shortcut menu or double-click any model in the component libraries to view the *Insert 3D Component Instance* dialog box opened to the **Parameters** tab. Using the **Place Component** menu command allows you to review component parameter values, change target coordinate system, and so forth, using the *Insert Component* dialog box. You can also view the **Image** and **Info** tabs.

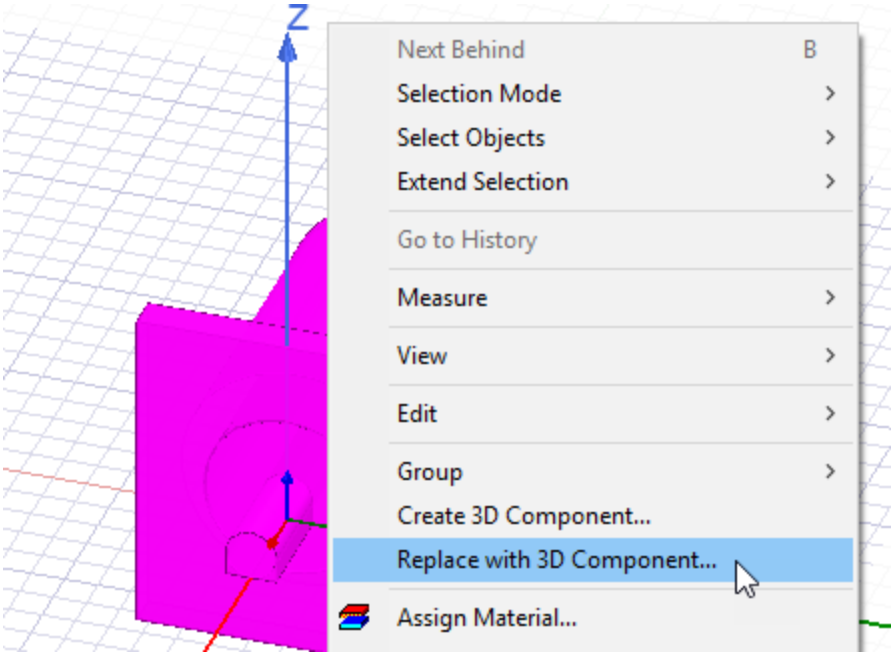
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component file organization on the disk

3D Component files are organized in three libraries (that is, *syslib, UserLib,* and *PersonalLib*). 3D component files must be in a **3DComponents** folder under the appropriate library location. Files must be further organized by design type in the **3DComponents** folder. For example, component files for Mechanical should be organized under 3DComponents\Mechanical, as done for syslib. The 3D Component Libraries window looks for components under a folder corresponding to the active design type, and it ignores component files under folders corresponding to another design type. For example, Mechanical will ignore component files placed directly under the HFSS subfolder.

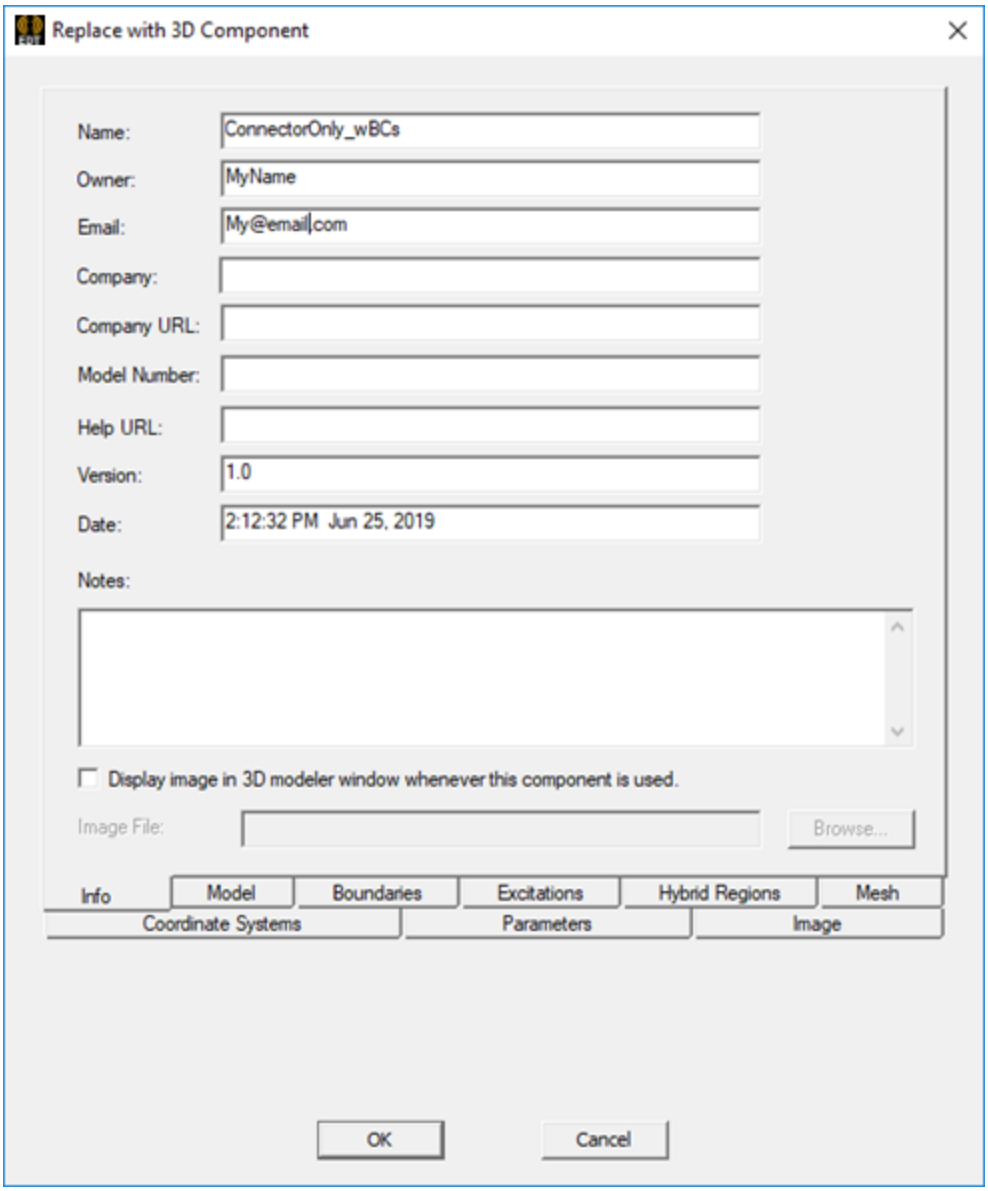
Replace Selection by 3D Component

To replace a selection in the 3D modeler window with a 3D component version of the selection, select an existing object or model, right-click, and select **Replace with 3D Component**. You can also click **Draw> 3D Component Library> Replace with 3D Component**. You cannot replace components from a design with an array setup. However, you create a 3D component from a unit cell and use that to create a Multi-Unit Cell array. When you replace a selection, object names are retained, port names are retained, component parameters map to existing design variables and report, field plot, and edit source are all preserved.



This command opens the **Replace with 3D Component** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The tabs let you view the component features organized according to geometry, design data, and settings. The **Replace with 3D Component** dialog box does not offer tabs for encryption or licensing, compared to Creating a 3D Component. You can use these tabs to change inclusion of objects, boundaries, ports, coordinate systems, or parameters for the 3D component. When you click **OK**, no *Save* dialog box is shown.

- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Mesh tab

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

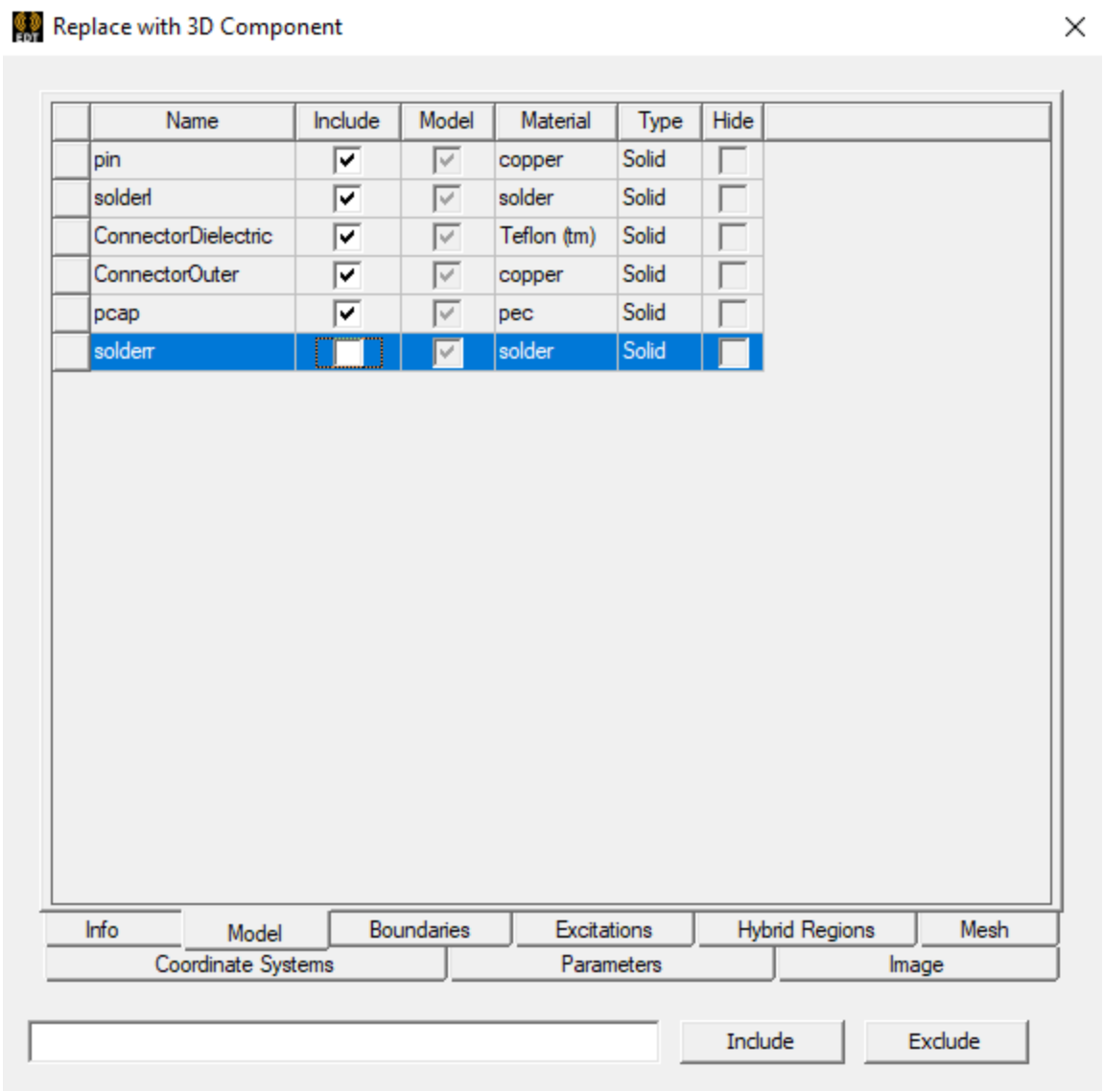
- Coordinate Systems tab
- Parameters tab
- Image tab

When you click OK, no save dialog is shown. The 3D component is saved to a temporary location. Insertion of instances uses the temporary file, but works just as regular 3D component insertion. The Project tree shows the Component under 3D Components, the boundaries are no longer listed under the Boundaries icon. Excitations, Reports and Analysis Setups remain the same. History Tree no longer shows detailed model edits. You can **Undo** and **Redo**.

Include or Exclude Feature from Component

You can view the various tabs and select which features to include or exclude while replacing the selection with a component. You can use the check boxes in the **Include** column for this purpose. You can also type a feature *Name* in the text box and use the **Include** or **Exclude** button. The text field supports wildcard characters (* and ?), so you can toggle the inclusion state for multiple objects with similar names in a single operation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An object's **Include** option under the *Model* tab impacts the inclusion state in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Design Data Notes:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Create 3D Component** dialog does not list design data without an assignment .
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Replace with 3D Component. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included.

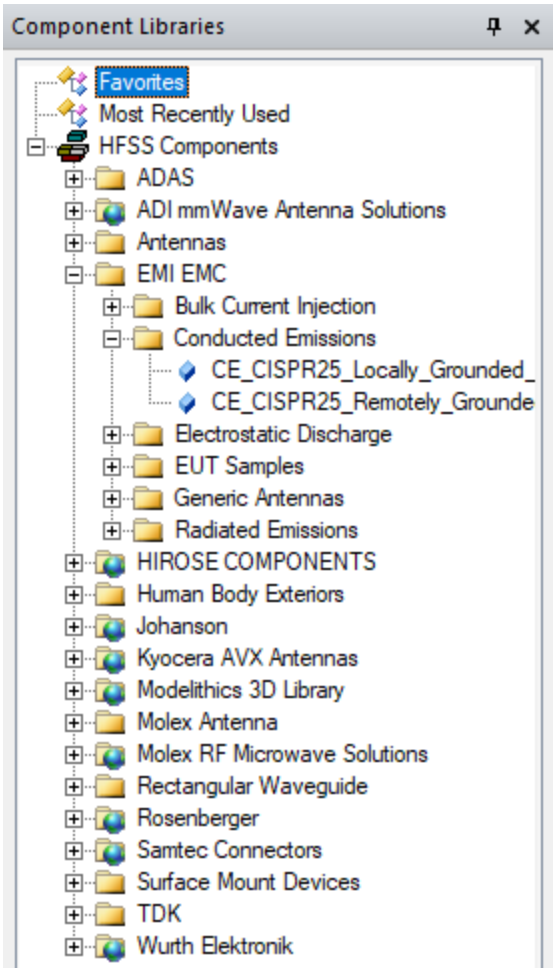
Inserting a 3D Component into a Design

Once you save one or more components to a library, and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D Component icon in the Project Manager or use the **Draw** > **3D Component Library** commands to browse your folders or libraries, or use the **View** > **Component Libraries** to display a Component Libraries window to navigate installed libraries. Alternatively, you can insert an instance of a component by right-clicking the 3D Component icon in the Project Manager and selecting **Insert Instance**.

From the Component Libraries window, you can navigate the directory of installed components, as well as any in your Personal Library. The hierarchical tree display includes **Favorites** and

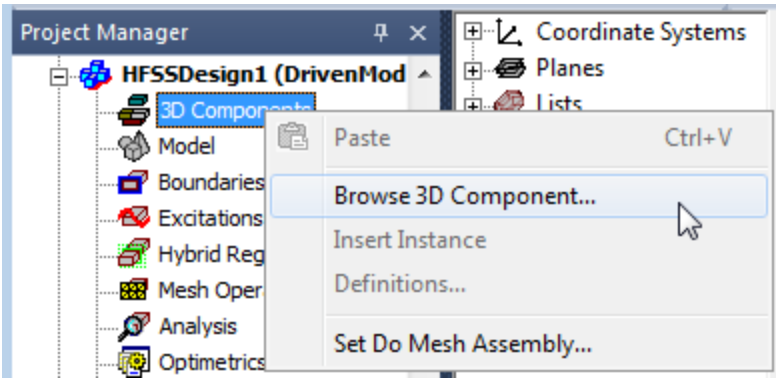
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Most Recently Used branches.



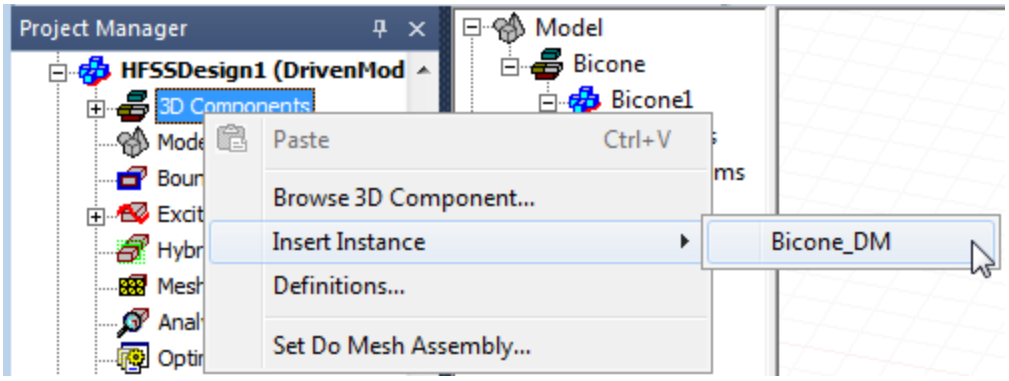
You can select any component from the library, and drag and drop to insert it in the design. Double-click any model in the component libraries to view the **Insert 3D Component Instance** dialog opened to the **Parameters** tab. You can also view the **Image** and **Info** tabs, and select the target coordinate system.

Selecting **Browse 3D Components** lets you navigate directories via a browser window.

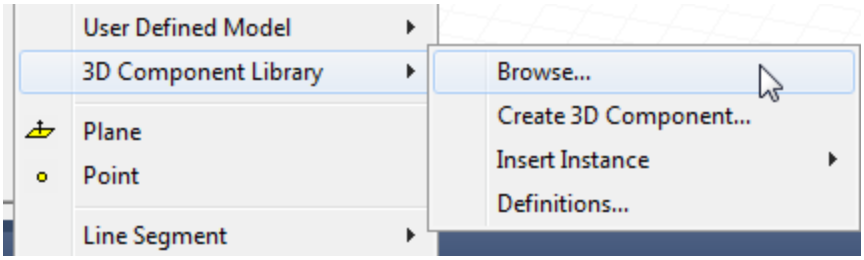


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

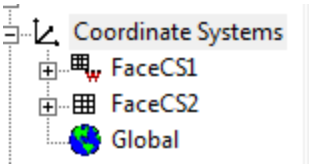
If you have previously inserted a 3D Component, the **Insert Instance** command is enabled on the menu. You can use this command to select from 3D Models in the design to easily insert another instance.



You can also click **Draw> 3D Component Library** to access the **Browse** and **Insert Instance** menus to select a component to insert into a design.

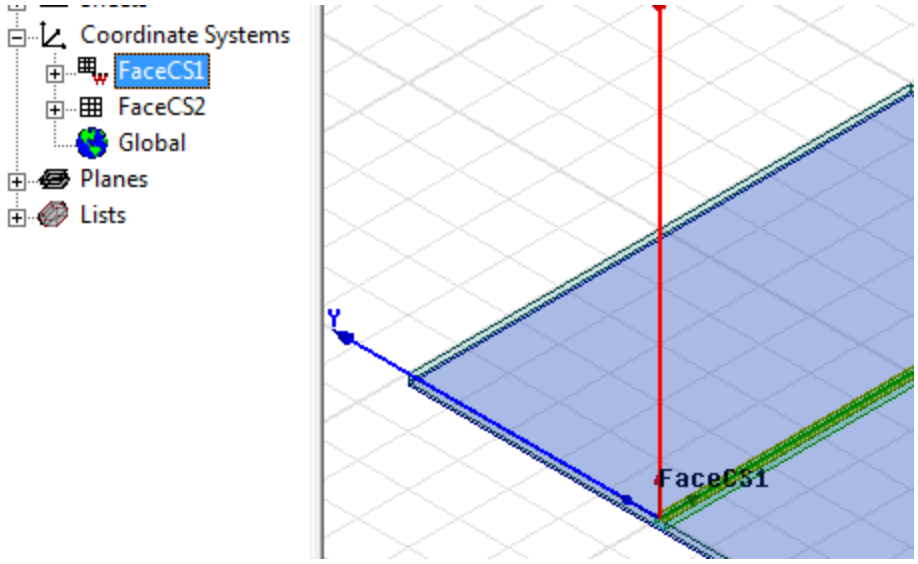


A target coordinate system provides a location for a component. For example, a design includes two additional coordinate systems:



Selection of a coordinate system in the History tree displays a potential location for inserting a component.

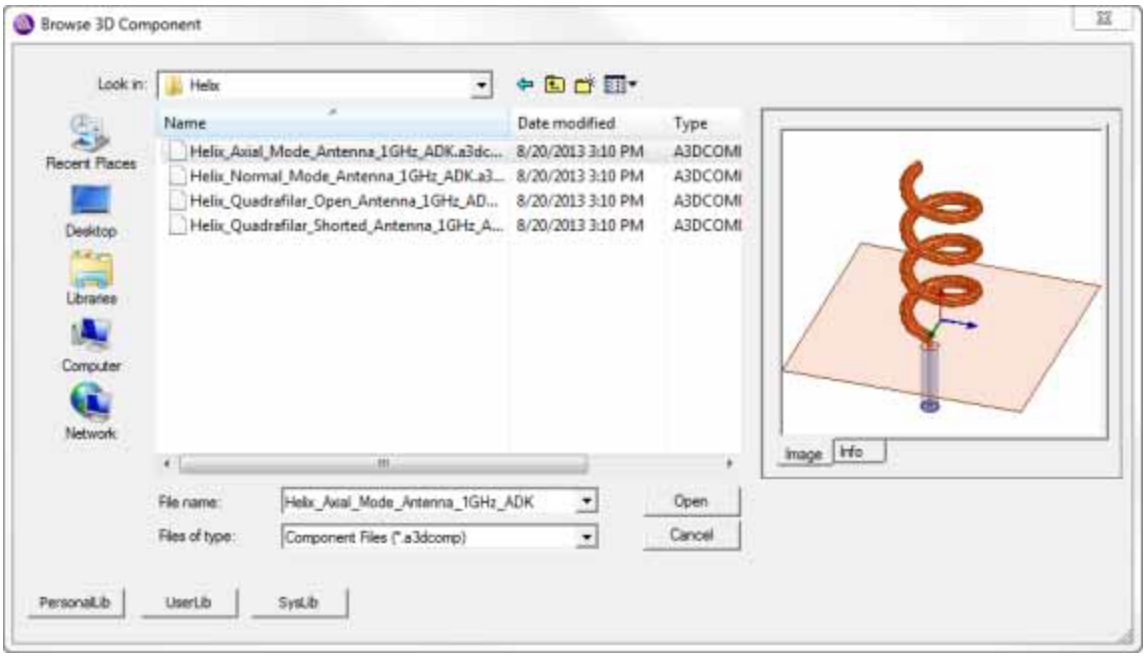
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Insert a 3D Component from the Menus

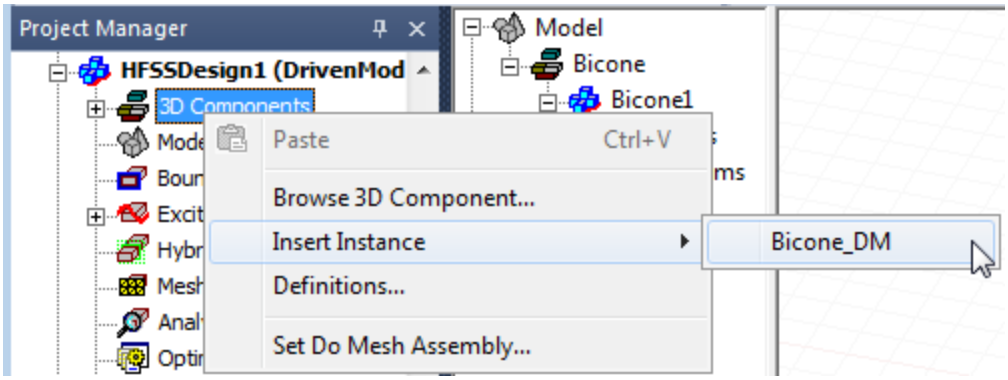
1. Right-click the **3D Component** icon in the **Project tree** to open shortcut menu. You can select **Browse** to use a browser window navigate the file system to the component.

   Selecting a 3D component file causes a display of the component image and the File name.

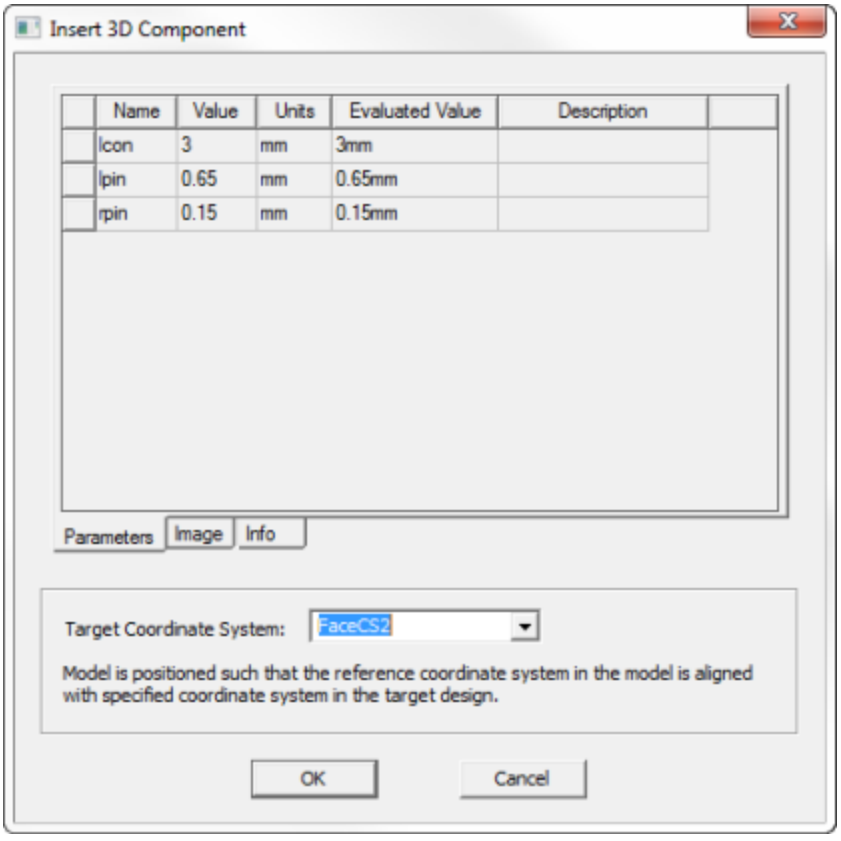


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have previously inserted a component into a Project, the shortcut menu for the 3D Component displays that component so that you can easily insert another instance of the same component.



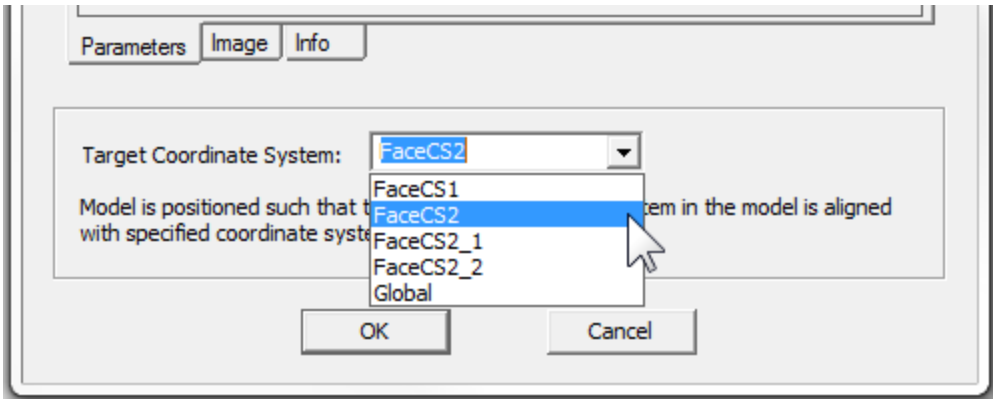
You see the **Insert 3D Component** dialog opened on the Parameters tab.



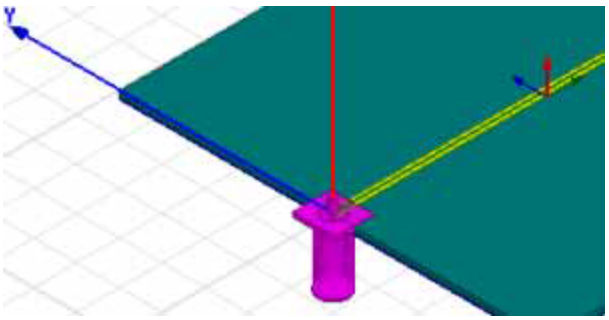
Tabs let you view the Parameters, Image, and Info. You can edit parameter values, and assign variables or expressions for parameters

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

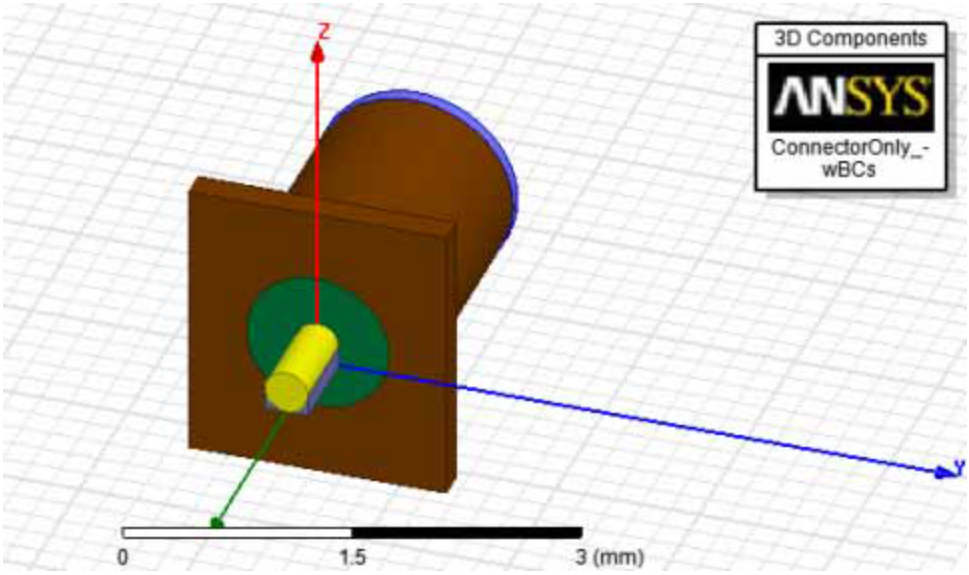
2. Use the menu to select the Target Coordinate System if any have been defined in addition to the Global coordinate system. The target coordinate system that you select is highlighted in the modeler window.



3. When you click **OK** the component is placed at the coordinate system you selected.



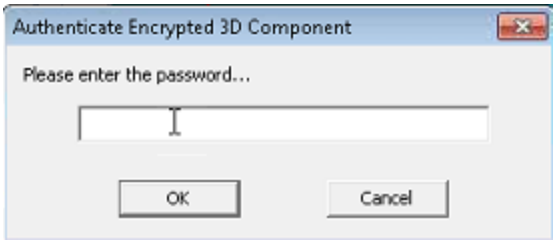
4. If the component has a logo defined, the image is always displayed in the upper right of the modeler window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you have used Browse, select a component and click **Open**. If you use the Insert Instance menu, select the component name.

If the component has been encrypted, you may be prompted to enter a password. If you insert another instance of the same component, you do not need to enter the password again for that component.



You have three tries to enter the correct password. Upon successfully entering the password, the Insert 3D Component dialog opens with the Encryption tab displayed.

> **Note:** There are some additional considerations with respect to projects using encrypted components. When a project contains encrypted components, the project file and other related files are also encrypted. Encrypted component contents may also be hidden from view in the design where the component is used. For more details, see Create 3D Component Encryption Tab.

Making Component Instances Independent

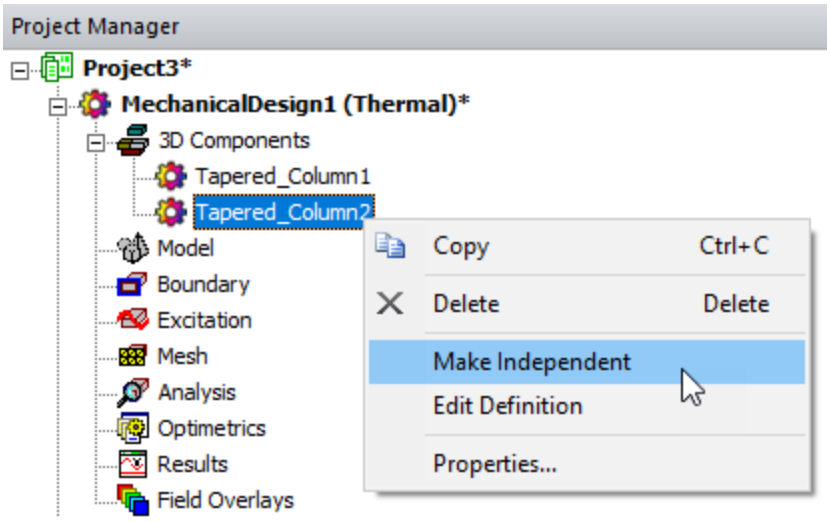
When inserting an instance of a component, it is listed under the original component and assigned the same definition. To create a unique component from a component instance, use the **Make Independent** option in the Project Manager or History Tree. **Make Independent** is only available if multiple instances of the component exist.

> **Note:**
>
> Right-click a component in the history tree to access the **History Tree Layout** menu. You can organize components in the history tree by definition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project Manager

1. In the **Project Manager** under **3D Components**, right-click on a component instance.



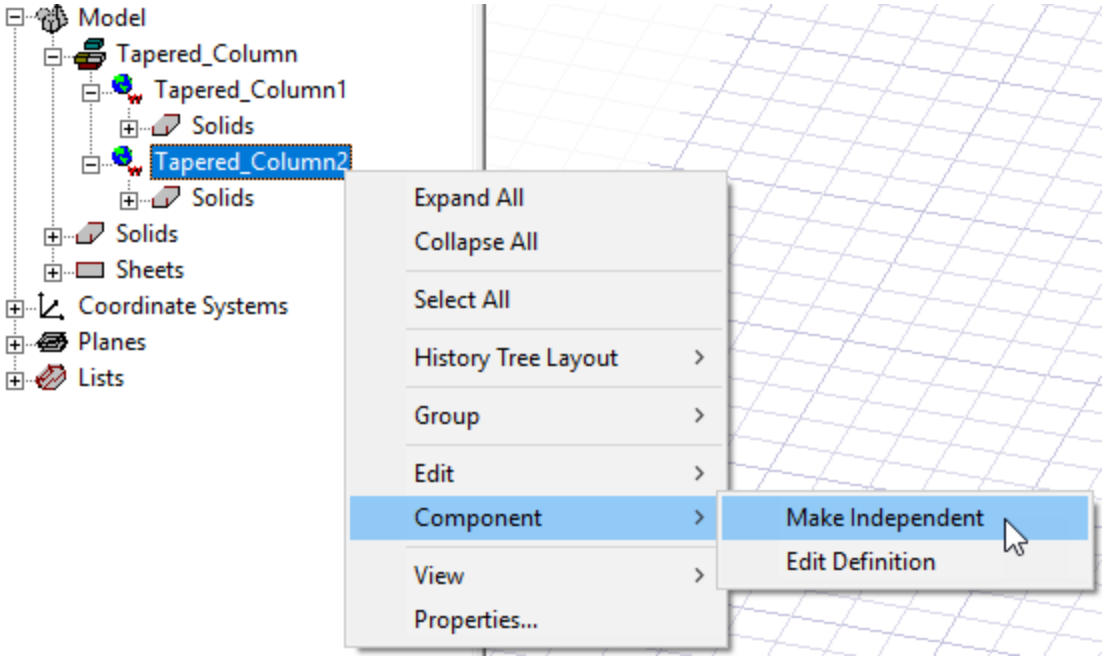
2. Select **Make Independent**. The history tree displays the independent component.

History Tree

> **Note:**
>
> From the history tree, you can select multiple instances and make them independent. If you select all instances of a definition, **Make Independent** is unavailable.

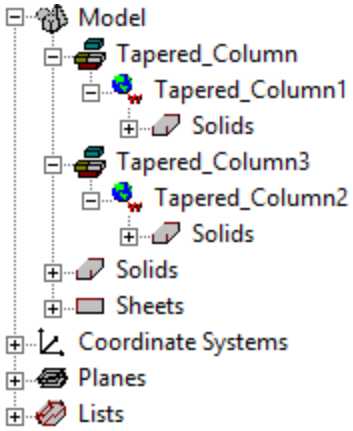
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the history tree, right-click on a component instance.



2. Select **Component** > **Make Independent**.

   In the history tree, the selected component appears independent of the original component and can be assigned a unique definition.

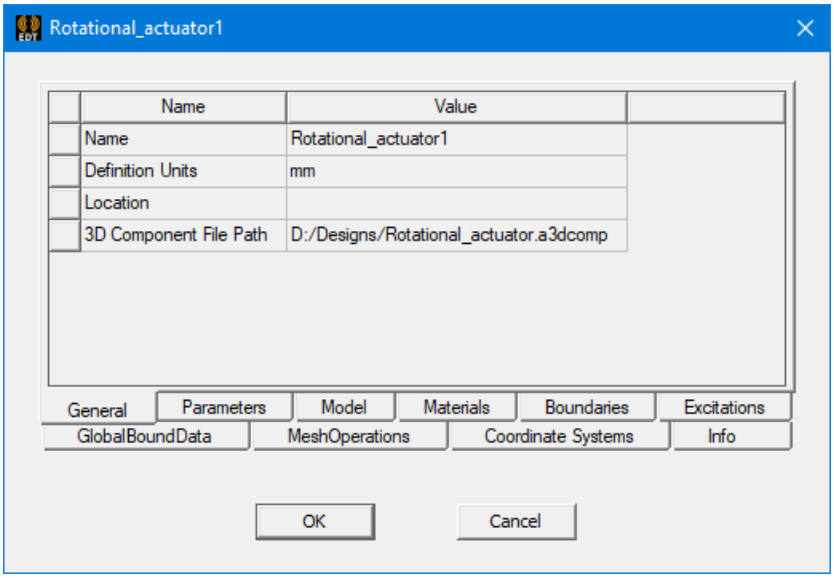


Viewing Properties of 3D Components

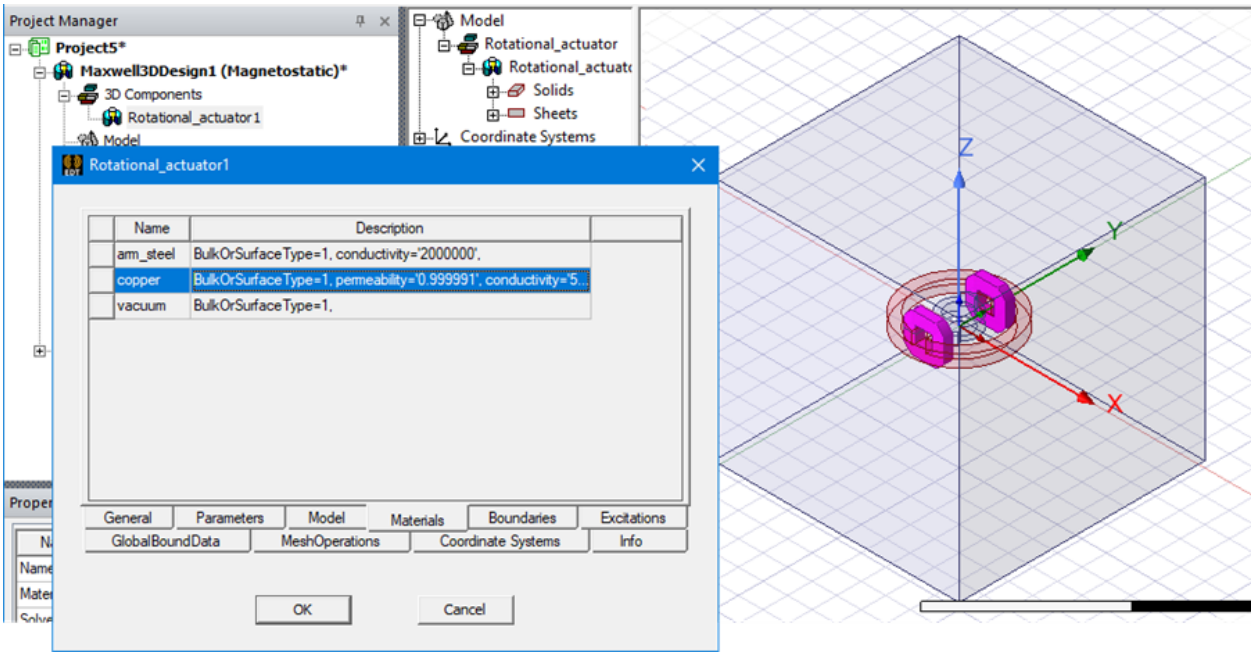
The Properties displayed for a selected 3D component depend on how you select that component. For a complete display of Properties, right-click the selected component (Project Manager, History Tree, or Modeler window) and click **Properties** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, click **Edit** > **Properties** using the menu bar. This action displays a window with tabs for all component properties.

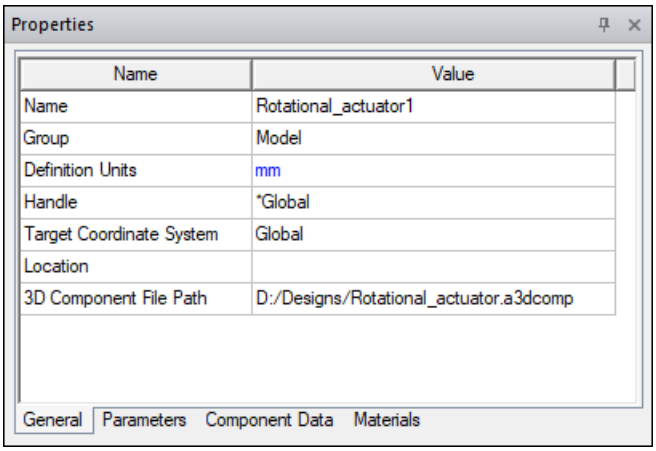


If you choose the *Materials* tab, you can then select any listed material, and the parts of the component made of that material are highlighted in the Modeler window.



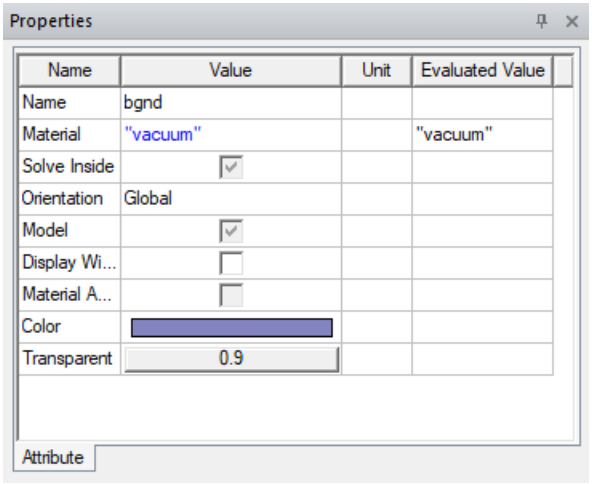
Selecting a 3D component in the Project Manager or the history tree displays tabs for *General, Materials,* and *Component Data* in the docked *Properties* window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



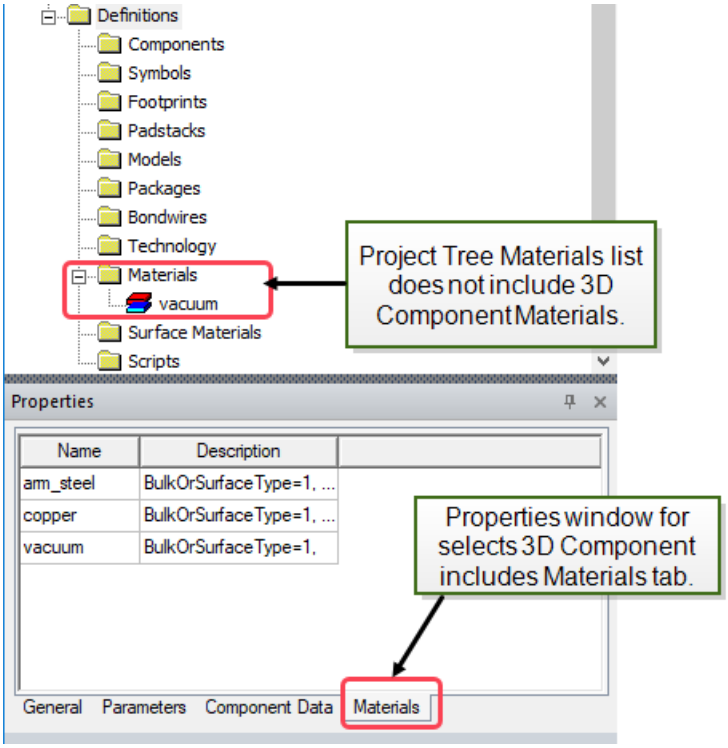
Note that the Handle property lists the handle coordinate system that was selected as reference when the 3D component was created; from this window, it can be changed to any coordinate system in the 3D component.

Selecting a 3D component object in the Modeler window displays the Attributes tab in the docked Properties window:



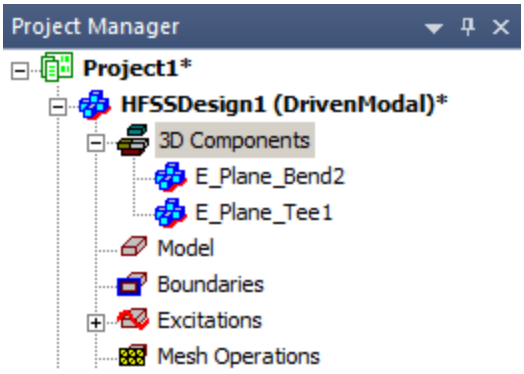
Component materials do not appear in the Project Manager's Definitions list. If you select the icon for an inserted component, the docked *Properties* window contains a *Materials* tab that shows the materials defined for the selected component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3D Components in the Project Manager Window

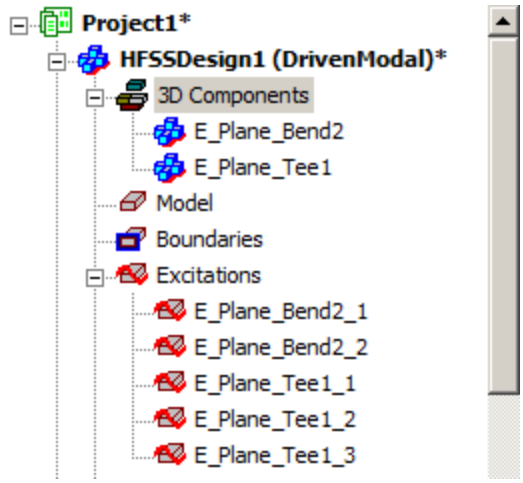
The **Project Manager** window organizes any 3D components that you add to a design under the 3D Components icon in the tree.



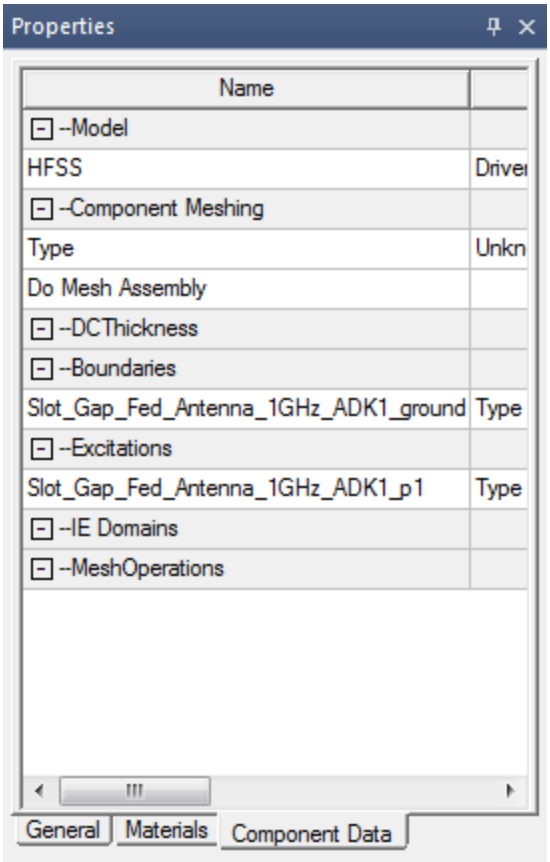
Selecting these inserted components populates the docked **Properties** window, which has tabs that display the instance attributes.

Excitations included in the 3D Components appear in the Project Manager. This gives you the ability to edit Post Processing settings, such as the de-embed distance, renormalization, and power settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



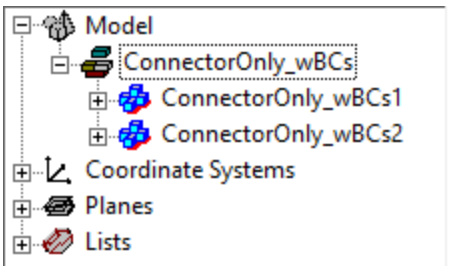
Boundaries defined inside the component do not appear in the Project Manager. However, boundary properties can be seen when you select the component in the Project Manager and view the component properties under the *Component Data* tab of the docked *Properties* window.



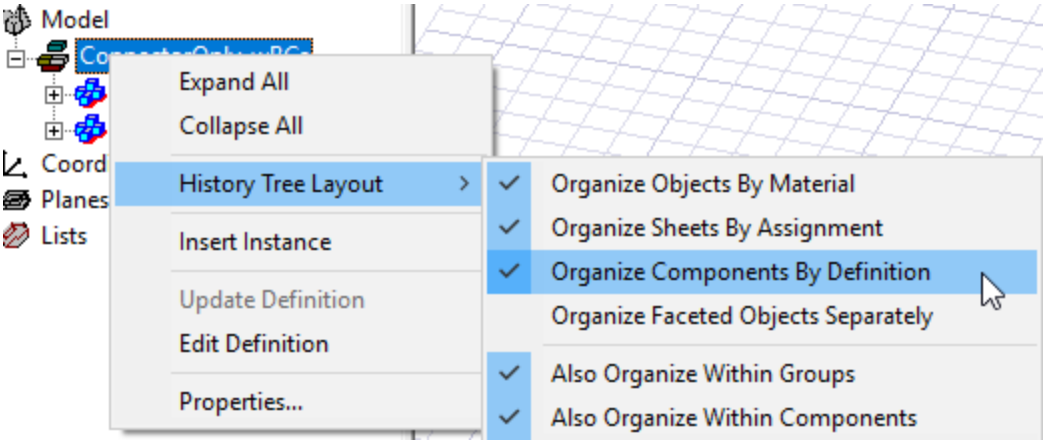
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Components in the History Tree

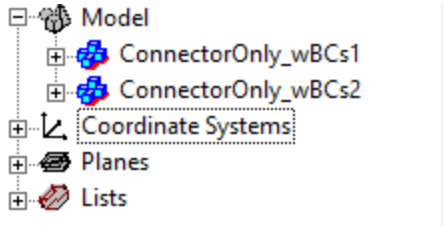
Although Group Commands do not work with 3D components, you can group all instance of a components by the component definition in the History tree.



To disable this grouping by instance, right-click an instance, and unselect **History Tree Layout>Organize Components by Definition**.



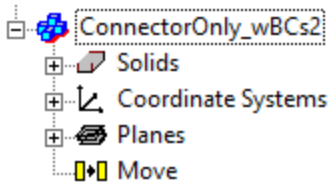
This causes the display to show numbered instances listed without the icon for the primary component.



Component Operations in the History Tree

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Component operations appear under that component folder toward the end.

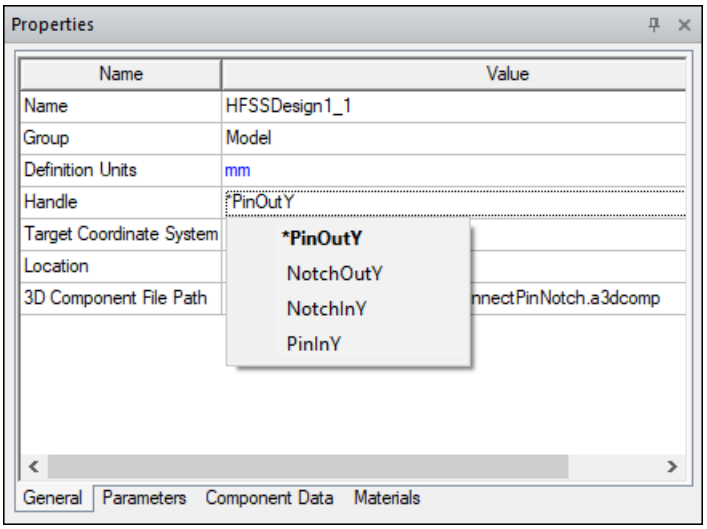


- Component operations acts on parts as well as CS.
- Boundaries, mesh operations are duplicated with duplicate and copy/paste operation.

Editing 3D Component Properties

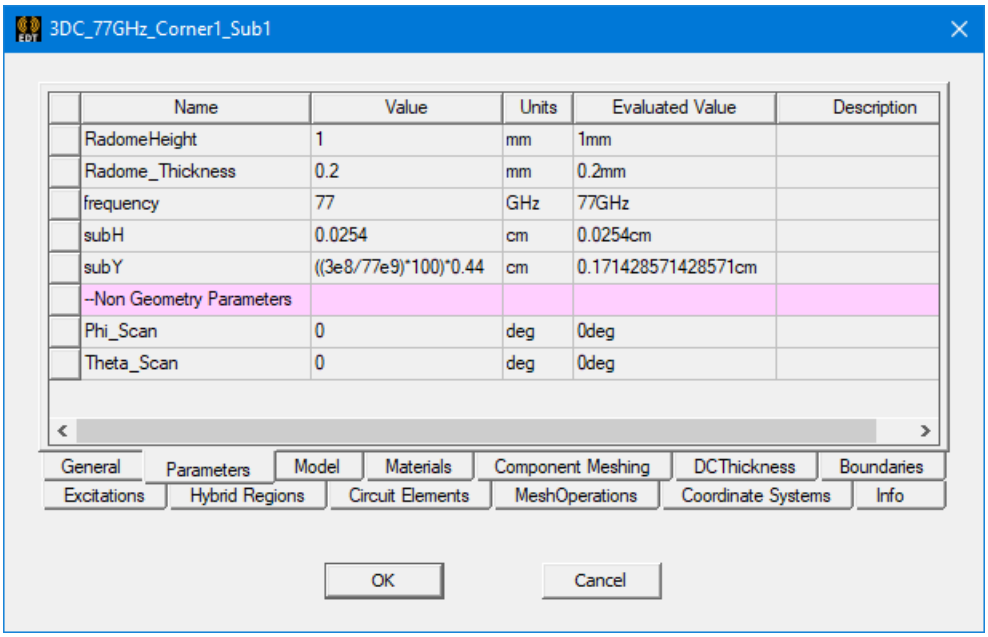
You can change and manipulate 3D components in several ways:

- Edit component handle and target CS in the General tab of component instance properties. All CS that are included in the component are available to be used as the handle. All coordinate systems in the assembly design that are eligible for component placement are available as target CS. Note that the default handle is shown with an asterisk.

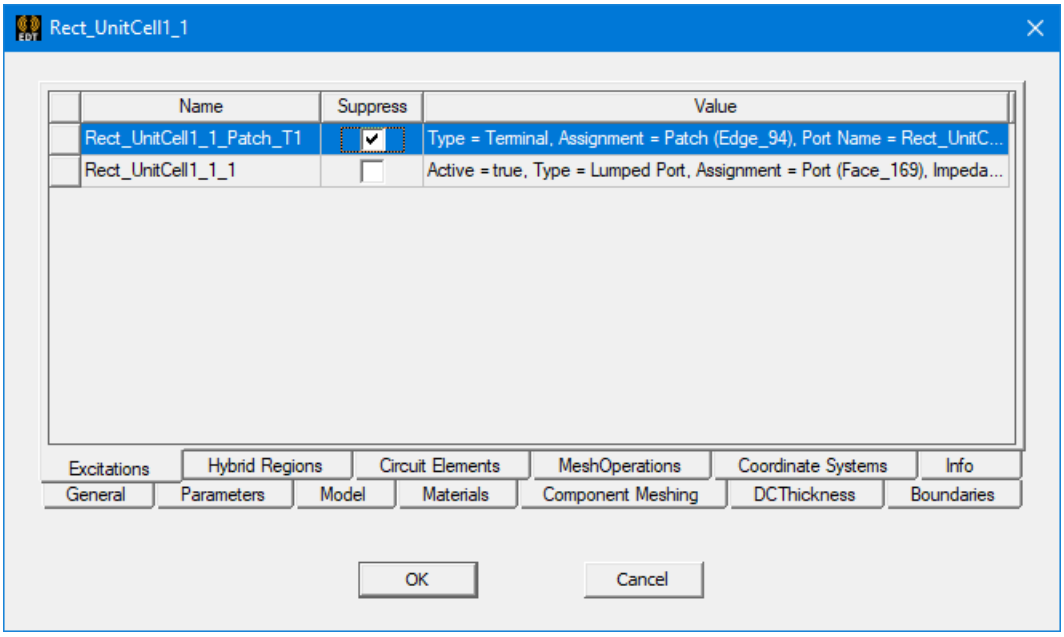


- Edit component parameters, if defined for that component. Editing the parameters of one component instance affects only that instance. If a component includes non-Geometry parameters, these are grouped separately on the Parameters tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



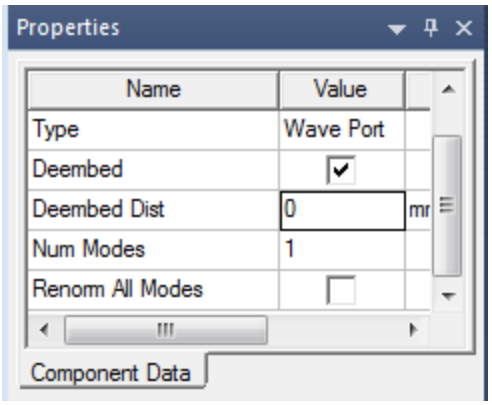
- Edit excitations by suppressing their use by means of the **Excitations** tab in the **Properties** window.



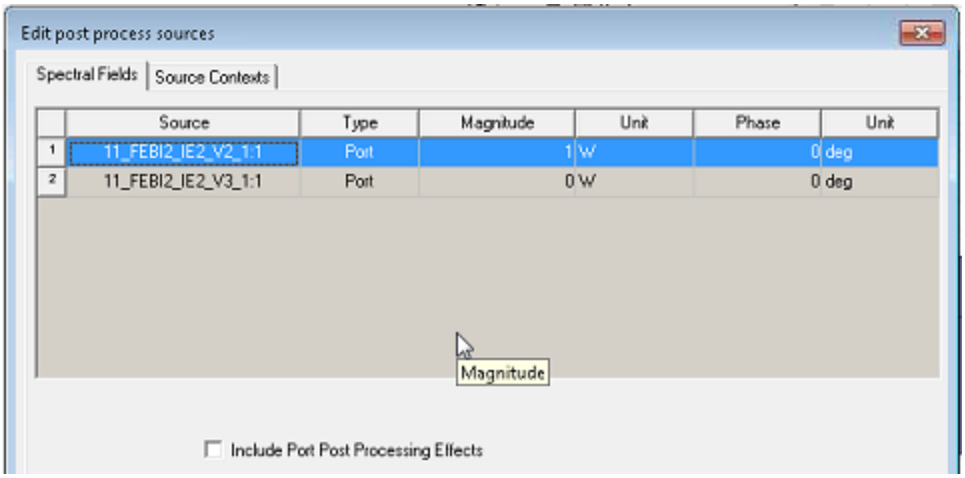
Checking Suppress in the Properties window removes the port on the selected instance from the Excitations list on the Project tree. Unchecking Suppress activates that port for the model and causes the port to appear in the Excitations list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



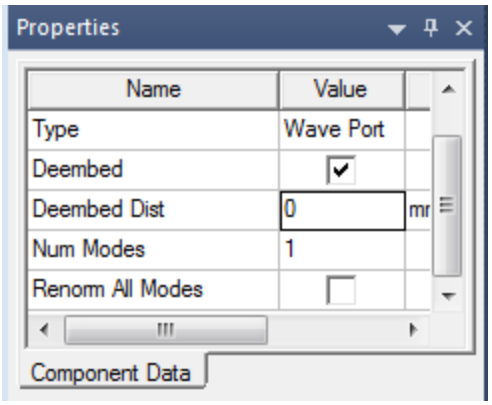
- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of a component are not allowed.

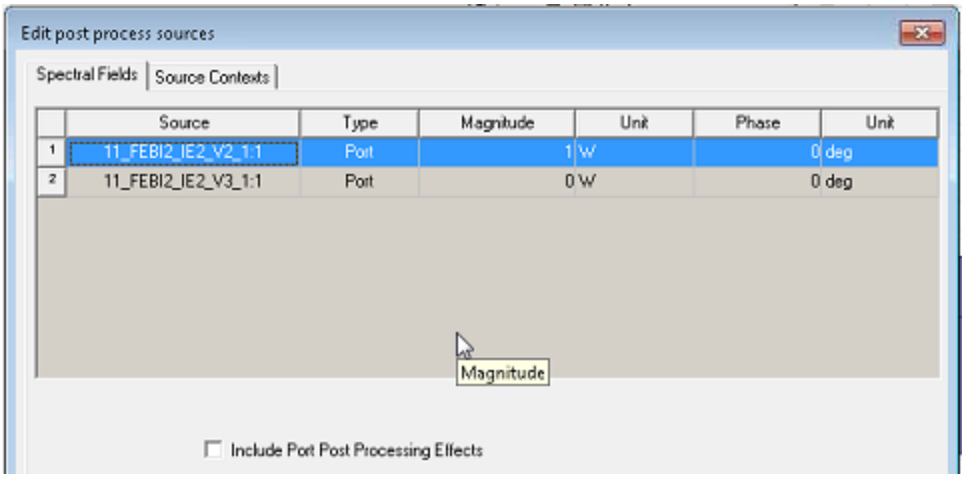
- Copy/Paste component
- Delete component
- Arrange (Move, Rotate, Mirror) component
- Duplicate component (Along Line, Around Axis, or Mirror)
- Edit Attributes for Display Wire frame, Color, and Transparency.
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of component are not allowed.

- Copy/Paste component
- Delete component
- Arrange component: Move, Rotate, and Mirror
- Duplicate component: Along Line, Around Axis, and Mirror
- Edit Attributes for Display Wire frame, Color, and Transparency
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

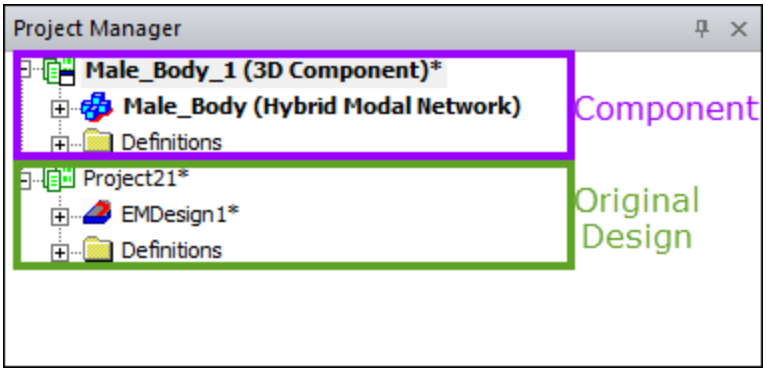
Editing a 3D Component Definition

> **Note:**
>
> If you reached this page looking for help on editing the definition of a **Layout Component**, as opposed to a *3D Component*, go here instead.

The **Edit Definition** feature allows the user to create new component definition using an existing, unencrypted 3D component file. This speeds up the task of creating new versions of components without having to start from the beginning. When **Edit Definition** is initiated, **Electronics Desktop** automatically creates a new project containing all 3D component data from the file in 'exploded' form. Component materials are added at the project level; variables are added at the project and design level; a detailed history is created for all objects in the component; and design data is added to the appropriate folders in the Project Manager.

> **Important:**
>
> Only 3D components that are **not encrypted** can be edited as described in this section.

The **Edit Definition** command creates a new project, named for the component, in the **Project Manager** window.

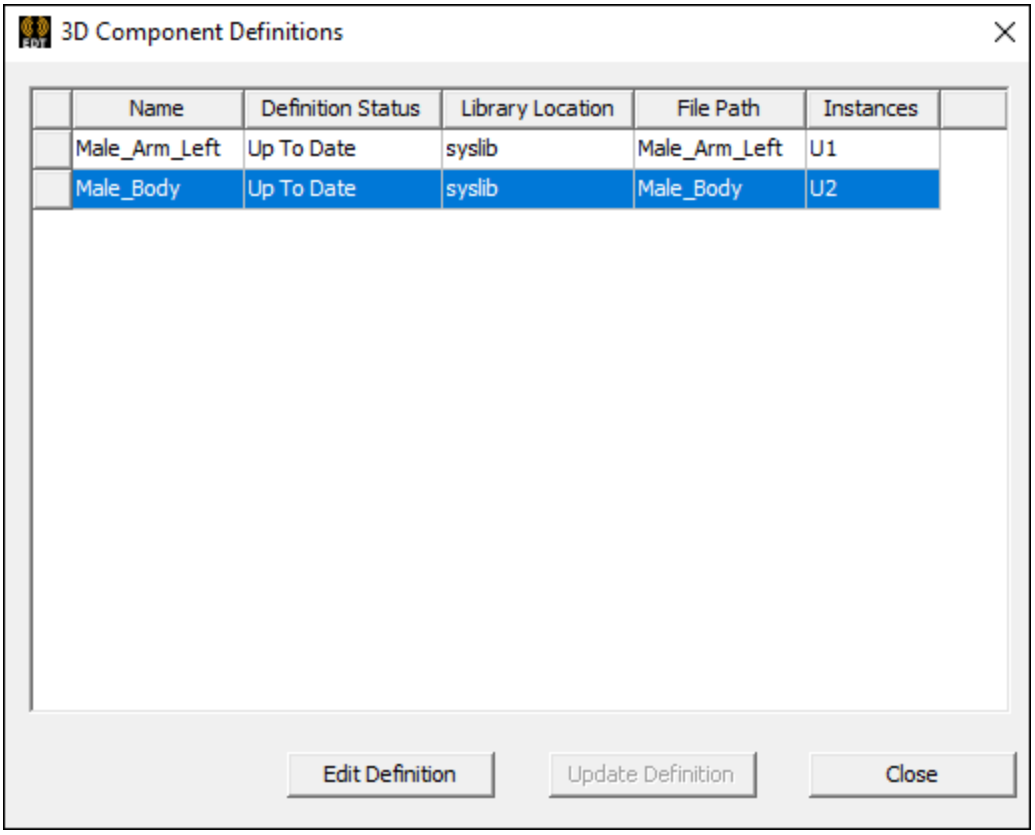


When the edited component is saved, an **Update 3D Component** window appears. From the **Update 3D Component** window, choose whether to save the edited component as a local instance or to the original library. To edit a 3D component definition:

1. Navigate to the **Edit Definition** window by following one of the following paths:
   - From the **History Tree**, right-click the component and select **Edit Definition**.
   - From the **History Tree**, right-click an instance under the component and select **Components** > **Edit Definition**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

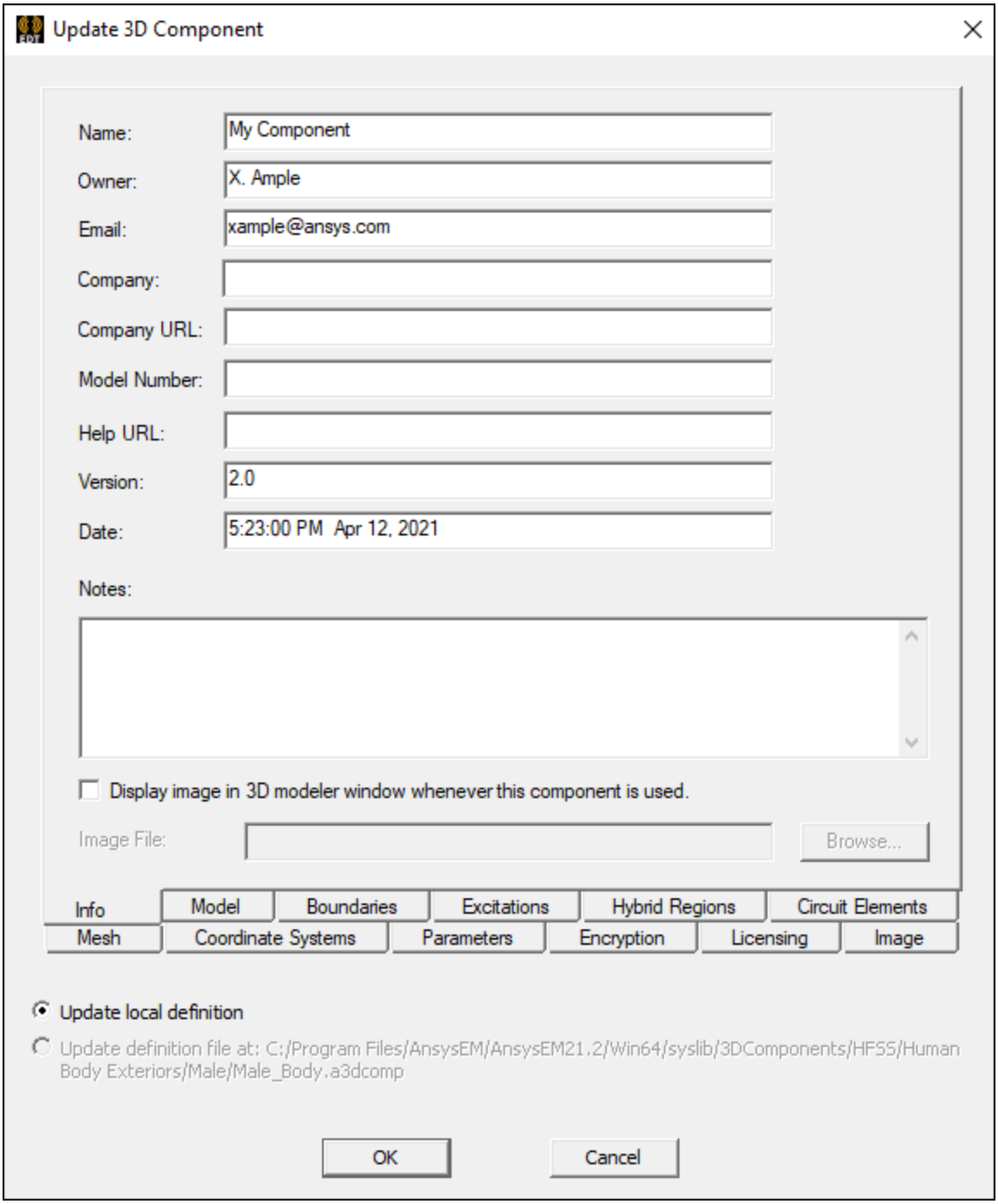
- From the **Project Manager**, right-click **3D Components** and select **Definitions** to open the **3D Component Definitions** window. From the **3D Component Definitions** window, select a component and click **Edit Definition**.



Regardless of which path is taken, a copy of the 3D component opens in the **Layout Editor**, and a new project folder appears in the **Project Manager** window, containing a copy of the original design.

2. Edit the component as desired.
3. To save the component as a new component (*.*a3dcomp*) or as its own **Electronics Desktop** (*.*aedt*) file, click **File** > **Save As**. Specify a file name and destination and click **Save**.
4. To save the component in the current project or for all projects, click **File** > **Save** to open the **Update 3D Component** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window is populated with all entities (objects, boundaries, excitations, etc.) in the component design.

5. Use the radio buttons to select one of the following options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Update Local Definition** – updates the component definition for use in the original design only.
- **Update Definition File At <Path>** – updates the component in the component library, affecting all other instances.

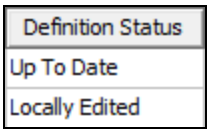
6. Click **OK**.

Verifying Updates to Components

To verify that a component has been edited:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

   The **Definition Status** column shows "Locally Edited" for any altered components.



2. Click **Close** to exit the window.

Updating Definitions

Once the updated component is saved, return to the original design and update all instances of that component. See: Updating 3D Component Definitions.

> **Note:**
>
> Definition update is not automatic even if **Edit Component** was initiated from that design. Instead, when an assembly design becomes active, a dialog box appears, prompting the user to update the 3D component definition. Then choose to update the definition from the prompt.

To update a component's definition:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

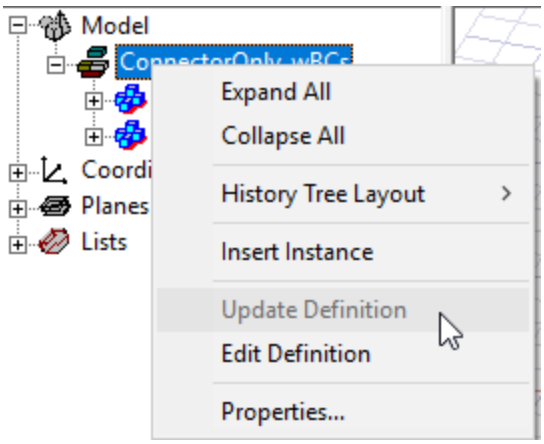
   Components with updates available display "Update Available" in the **Definition Status** column.

2. Select the desired component and click **Update Definition**.
3. Click **Close** to exit the window.

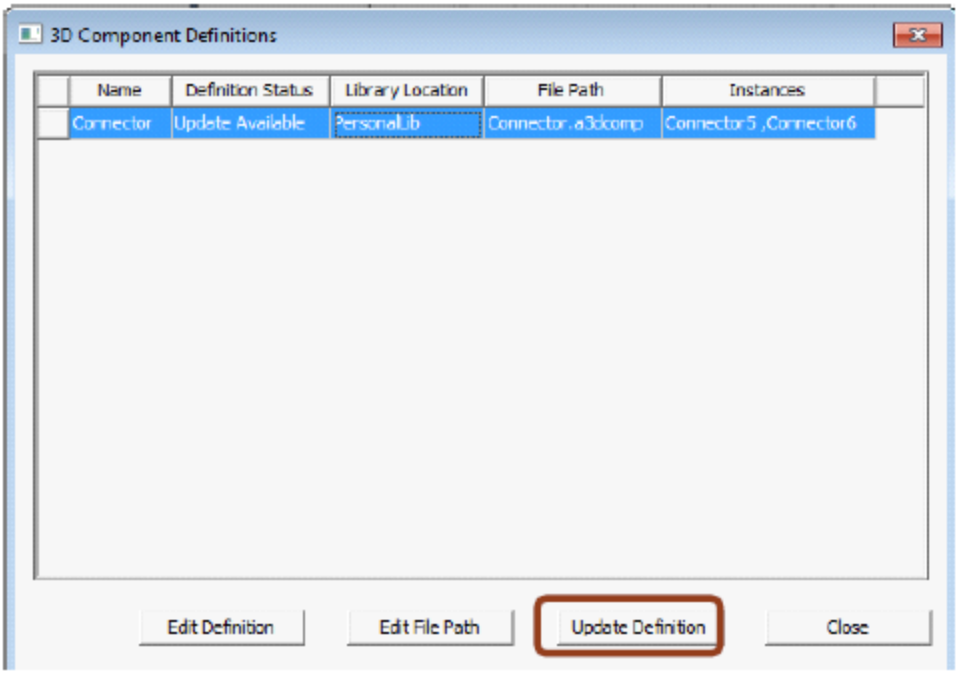
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Updating a 3D Component Definition

If you create or obtain a new version of a component, and then open a project that contained an earlier version of that component, you can right-click the component in the History tree and select **Update Definition** to replace the older version of the component with the newer one This updates all the instances of the component using that definition. Update Definition is unavailable if a newer definition is unavailable.



You can also accomplish this by right-clicking on the 3D Component icon in the Project Tree and selecting **Definitions...** from the shortcut menu. This displays a dialog listing available component definitions. The Definition status column lets you know if an undated definition is available.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All instances and component operations in the design are updated:

- Parameter values are preserved.
- New parts are generated in all instances.
- Deleted parts are deleted from all instances.
- Edited parts are changed in all instances.
- Design data, materials, coordinate systems can also be updated.
- Undo is supported; command is scriptable.
- The Definitions dialog box provides complete list of definitions in the design.
- Definitions can be updated from the dialog box.

Layout Components [Beta] – Topic Index by Product

Support for layout components is a beta feature available in HFSS, Maxwell, and Mechanical designs. It is also a native feature for modeling printed circuit boards (PCBs) in Icepak.

One or more *Layout Components* can be included in a design, each of which is based on a source *HFSS 3D Layout* of a PCB. Availability of the feature may be limited to specific solution types, depending on the product.

Product-specific *Layout Component* topics are provided in the help. Click the appropriate link below for the product and topic of interest:

- **HFSS:**
  - Layout Component in HFSS 3D (Introduction)
  - Layout Component in HFSS 3D Process Flow
  - Using Layout Custom CS for Placement in HFSS 3D
  - Editing a Layout Components Definition
  - Replacing or Editing Layout Components
  - Visualization for Layout Components in HFSS 3D Modeler
  - Flex PCB Component Support in HFSS 3D
- **Icepak:**
  - Layout Components in Icepak (Introduction)
  - Adding a Layout Component
  - Inserting a Layout Component Instance
  - Editing a Layout Component Definition
  - Replacing or Editing Layout Components
  - Layout Component Properties
- **Maxwell:**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Using a 3D Layout Component
    - Using Layout Custom CS for Placement
    - Editing a Layout Component Definition
    - Replacing or Editing Layout Components
    - Visualization for Layout Components in 3D Modeler
    - Flex PCB Component Support
- **Mechanical:**
    - Layout Components in Mechanical (Introduction)
    - Adding a Layout Component
    - Inserting a Layout Component Instance
    - Editing a Layout Component Definition
    - Replacing or Editing Layout Components
    - Layout Component Properties
    - Layout Component Properties: Visualization Tab

Modifying Objects

You can quickly modify the position, dimensions, and other characteristics of objects created in the 3D Modeler window.

What do you want to do?

Object and History Editing	**Modify Object Appearance**
Modify Object Dimensions	**Modify Object Location or Orientation**
Surface or Edge Operations	**Boolean Operations on Objects**

Object and History Editing

- Copy and paste objects.
- Delete objects
- Delete Last Operation
- Cutting Objects
- Duplicate objects
- View and Edit Commands on History Tree Objects
- Purge History
- Generate History

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Object Appearance

- Assign color to an object.
- Assign transparency to an object.
- Also see Modifying the Model View

Modify Object Dimensions

- Assigning a Cross Section and Dimension to a Polyline
- Scale the size of objects
- Connect objects
- Move faces or edges
- Convert polyline segments
- Rounding the edge of an object (Fillet)
- Flattening the edge of an object (Chamfer)

Modify Object Location or Orientation

- Move objects
- Rotate objects
- Change the Orientation of an object
- Mirror objects about a plane.
- Offset an object (move every face of an object).
- Also see Modifying the Model View

Surface or Edge Operations for Objects

- Sweep objects.
- Cover lines.
- Cover faces.
- Uncover faces.
- Detach faces.
- Detach edges.
- Create a new object by taking a cross-section of a 3D object.
- Wrap Command
- Project Sheet
- Thicken Sheet

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boolean Operations on Objects

- Unite objects.
- Subtract objects.
- Create objects from intersections.
- Create an object from a face.
- Create an object from an edge.
- Split objects.
- Separate objects.
- Imprint Projection Commands
- Imprinting an Object

Assigning Color to Objects

1. Select the object to which you want to assign a color.

   > **Note:**
   >
   > If the *Properties* window is not visible on the desktop, click **View> Properties Window** or use **Edit> Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the color sample in the **Value** column of the **Color** row.

   The *Color* palette appears.

4. Select a *Basic* or *Custom* color from the **Color** palette.

   Alternatively, you can specify a color by typing numbers between 0 and 255 to specify the following values:

   - **Hue, Sat** (saturation), and **Lum** (luminance), or
   - **Red, Green,** and **Blue**

   You can also specify a color by clicking within the color spectrum box and using the luminance slider to the right of this box.

5. Optionally, click **Add to Custom Colors** to save the settings as a custom color, which makes in easier to specify the same color for other objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you wish to overwrite a previously defined or default custom color, choose the custom color swatch to overwrite *before* specifying the new color parameters. To clear your customizations and restore default choices, click **Reset Custom Colors to Defaults**.

6. Click **OK**.

   The color is assigned to the selected object.

Setting the Default Color of Objects

The default color you specify for objects is applicable to when the objects are rendered as solids (shaded) or rendered as wireframes (that is, outlines only). The outline color is the same as the object color you specify according to the following procedure:

1. Using the menu bar, click **Tools**> **Options**> **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the *Options* dialog box, select **3D Modeler**> **Display**> **Rendering**.
3. In the *Object Appearance* section, click the color sample button to the right of **Default color**.

   The **Color** palette appears.

4. Select a color from the **Color** palette, as described in the parent topic.

   Any objects you draw after this point will be assigned the default color you selected. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).

5. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Transparency to an Object

1. Select the object to which you want to assign a transparency.

   > **Note:**
   >
   > If the *Properties* window not visible on the desktop, click **View> Properties Window** or use **Edit> Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the *Value* in the **Transparent** row.

   The *Set Transparency* dialog box appears.

4. Move the slider to the right to increase the transparency of the object. Move the slider to the left to decrease the transparency of the object. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.
5. Click **OK**.

Setting the Default Transparency of Objects

1. Using the menu bar, click **Tools**> **Options**> **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the *Options* dialog box, select **3D Modeler**> **Display**> **Rendering**.
3. In the *Object Appearance* section, click and drag the **Default Transparency** slider. Move the slider to the right to increase the default object transparency. Move the slider to the left to decrease the default object transparency. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.

   Any objects you draw after this point will be assigned the default transparency you specified. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

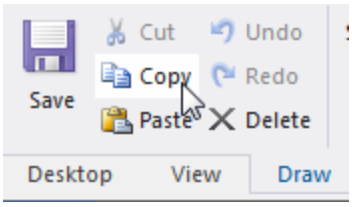
   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).

Copying and Pasting Objects

To copy objects and paste them in the same design or another design, use the **Edit>Copy** and **Edit>Paste** commands. For data link purposes, where you want to include the material assignments as well as the geometry, you can use the Import from Clipboard command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

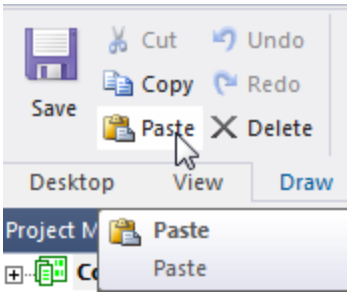
1. Select the objects you want to copy.
2. Click **Edit**>**Copy** or in the **View/Draw/Model** tabs of the ribbon, click the **Copy** icon:



The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted.

To cut an item to the clipboard and deleting the original, use the scissors icon on the toolbar.

3. Select the design into which you want to paste the objects. It can be the same design from which you copied the items.
4. Click in the **3D Modeler** window.
5. Select the working coordinate system. Objects are pasted relative to the current working coordinate system.
6. Click **Edit**>**Paste** or in the **View/Draw/Model** tabs of the ribbon, click the **Paste** icon:



The objects appear in the targeted Modeler window and in the new History tree. The pasted objects become the current active selection. If you undo the paste, the active selection goes back to the previous selection. If you redo, the pasted objects become the current active selection.

Items on the Clipboard can be pasted repeatedly. The items currently stored on the Clipboard are replaced by the next items that are cut or copied. Designs and projects containing encrypted components can be copy pasted for products using 3D Modeler

Import a Model from the Clipboard

You can import a model from the Clipboard in order to use the geometry from a different design. To use geometry with datalink, the geometry ID must be preserved. The **Import from Clipboard** command preserves geometry IDs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To import a model from the Clipboard, the model for the current design must be empty. If the design is not empty, existing geometry is deleted.

The geometry model is imported from the Clipboard with the ID preserved.

1. Select the objects you want to copy from the source design. For selecting all objects, you can use **Edit**> **Select All** or **Ctrl+A**.
2. Click **Edit**> **Copy**.

   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted from the source location.

3. Select the design into which you want to paste the objects. It can be in the same project as the design from which you copied the items or in a different project.
4. Click in the **3D Modeler** window.
5. Click **Modeler**> **Import From Clipboard**.

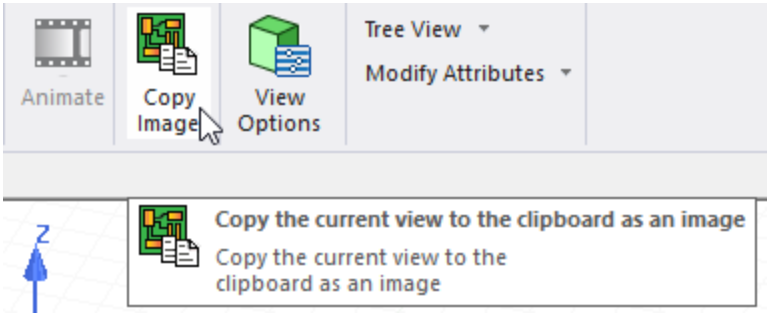
   The geometry is pasted from the Clipboard with the IDs preserved. Additionally, the source design's coordinate systems, materials, appearances, and other attributes are preserved and duplicated in the target design.

Copy Image

You can import images of the **3D Modeler** window or of Reports into any other application. The image has to be copied to the clipboard, so that it can be imported into the other application.

To copy an image of the Modeler window and paste it into another application:

1. Make the Modeler window from which you wish to capture an image active.
2. Copy an image to the clipboard using one of the following three methods:
   - From the menu bar, click **Edit> Copy Image**.
   - Right-click the Modeler window and select **Copy Image** from the shortcut menu.
   - On the **View** ribbon tab, click **Copy Image**:



   A raster image of the contents of the *Modeler* window is copied to the Clipboard.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Open the application into which you want to paste the image of your model and follow that application's image pasting procedure.

To copy an image of a Report to paste into another application:

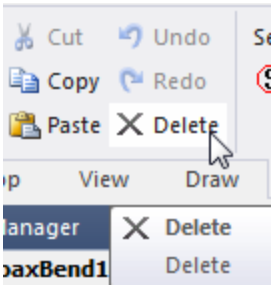
1. Make the report the active window and use the **Copy Image** command from the **Edit** menu, short-cut menu, or **View** ribbon tab.

   An image of the report is copied to the Clipboard.

2. Open the application into which you want to paste an image of the report and follow that application's image pasting procedure.

Deleting Objects

1. Select the objects to delete.
2. Delete the selected objects using *one* of the following three methods:
   - From the menu bar, click **Edit**> **Delete**.
   - From the **View, Draw,** or **Model** ribbon tab, click the **Delete** icon:

   - Press the **Delete** key.

   The objects are deleted.

> **Note:**
>
> Boundaries, excitations, hybrid regions, and mesh operations that were associated only with deleted objects are also deleted along with the objects. However, associated design variables remain, and they can be assigned to other geometry.

Deleting Start Points and Endpoints

For individual segments of polylines, you can delete the start point and/or the end point, with the following conditions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For the *first* segment, you can only delete the *Start Point*, which effectively removes the entire segment.
- For the *last* segment, you can only delete the *End Point*, which effectively removes the entire segment.
- For all remaining (internal) segments, you can delete either the *Start Point* or the End Point. The new point becomes the opposite point of the adjacent segment, and the total number of segments is reduced by one.

To delete *Start Points* or *End Points* of polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to delete and expand the branch to expose the *CreateLine, CreateArc, CreateAngularArc,* or *CreateSpline* entries. These entries represent the individual polyline segments.
2. In the History Tree, right-click the polyline segment you want to edit and choose either:
   - **Delete Start Point**, which removes the preceding segment, or
   - **Delete End Point**, which removes the subsequent segment.

   Alternatively, you can select the segment and choose **Edit**> **Delete Start Point** or **Edit**> **Delete End Point** from the menu bar.

Delete Last Operation

You can delete the last geometry operation performed on an object. For certain situations, this functionality can offer an advantage when compared with the *Undo* command. For example, if you performed several valid operations (such as assigning boundaries, excitations, or mesh operations) after operating on an object's geometry, you could revert the last geometry change without undoing the subsequent assignments.

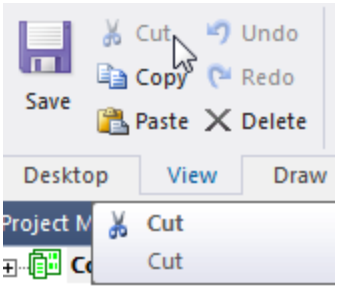
1. Select the object.
2. From the menu bar, click **Modeler**> **Delete Last Operation**.

   This command undoes the last geometry operation performed on the selected object, including removing that operation from the History Tree and updating the context for the *Undo* and *Redo* commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cutting Objects

1. Select the objects to cut.
2. From the menu bar, click **Edit**> **Cut** or, on any ribbon tab, click the **Cut** icon:



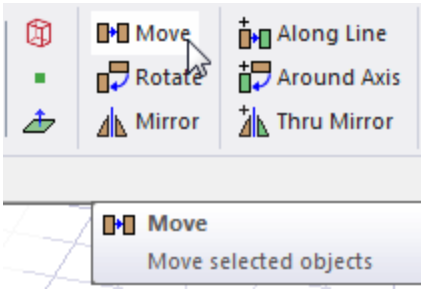
The objects are copied to the Clipboard and removed from the design. They can subsequently be pasted elsewhere, such as into a different design.

Moving Objects

The process for moving objects differs depending on the Drawing Data Entry Mode for the Drawing Options, whether Point mode (F3) or Dialog mode (F4).

For Point mode (click F3 to change to Point Mode):

1. Select the objects to move.
2. Click **Edit**> **Arrange**> **Move** or, on the **Draw** ribbon tab, click the **Move** icon:



3. Select an arbitrary anchor point in one of the following ways:
   - Click the point. (The *Measure Data* window shows the coordinates of the points you click.
   - Enter the point's coordinates in the **X, Y,** and **Z** boxes.
4. Select a target point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

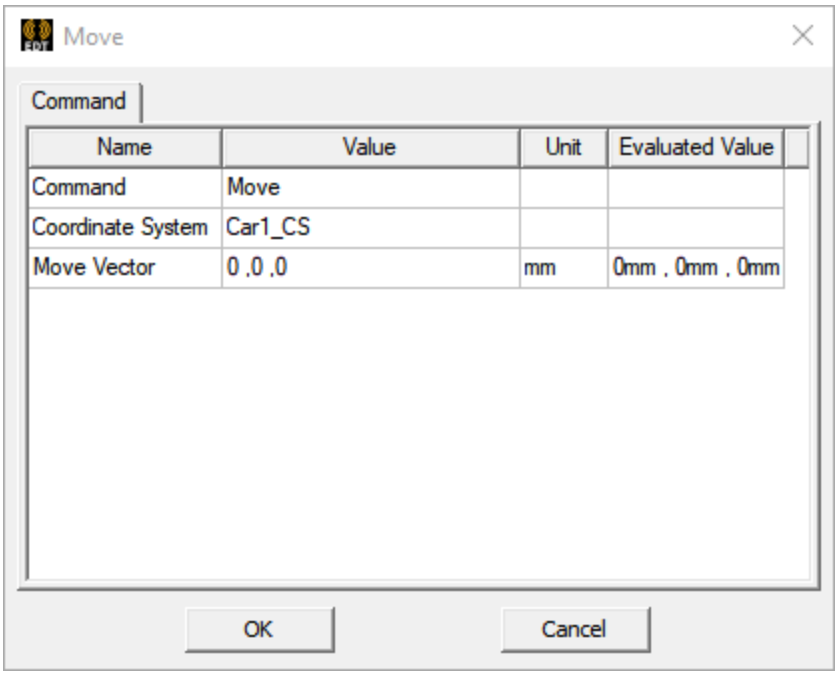
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All selected objects move the distance determined by the offset between the anchor point and the target point.

For Dialog mode (click F4 to change to Dialog mode):

1. Select the objects to move.
2. From the menu bar, click **Edit**> **Arrange**> **Move** or, on the **Drawing** ribbon tab, click the **Move** icon.

   A *Move* command dialog box appears.



3. Enter the coordinates of the location to which the object should move and click **OK**.

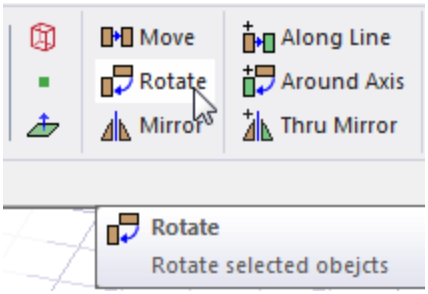
Rotating Objects

Rotate objects about the x-, y-, or z-axis using the **Edit> Arrange> Rotate** command.

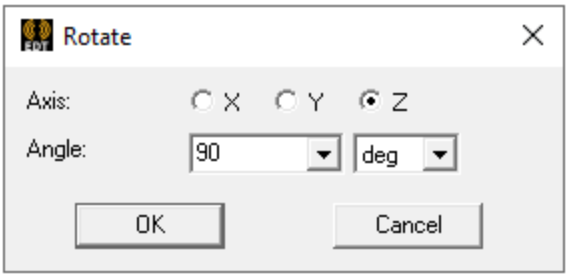
To rotate objects about an axis:

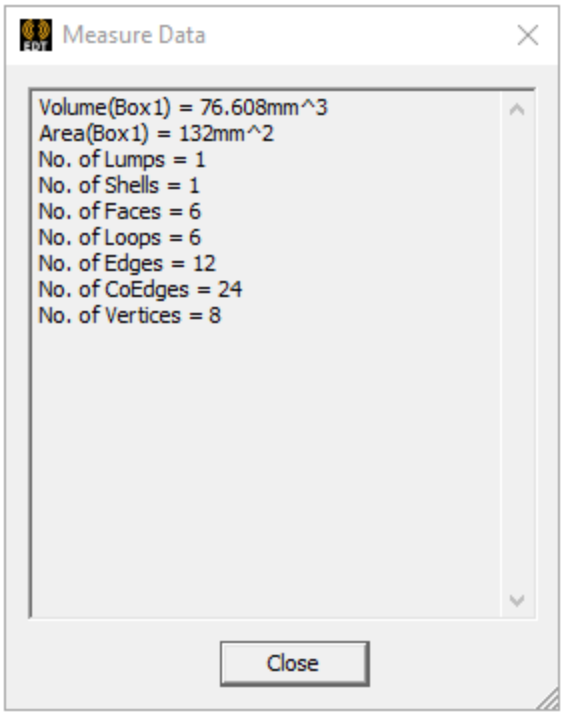
1. Select the objects to rotate.
2. From the menu bar, click **Edit**> **Arrange**> **Rotate** or, on the **Draw** ribbon tab, click the Rotate icon:

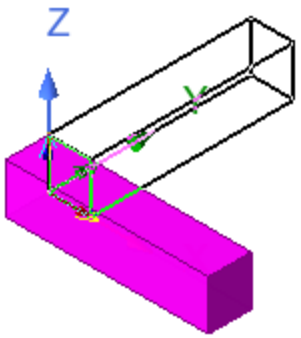
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Rotate** dialog box appears along with the **Measure Data** window. A visualization of the proposed change appears in the Modeler window.



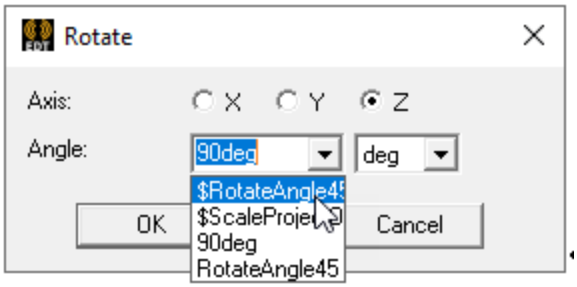




3. Select the axis about which to rotate the objects: **X, Y,** or **Z**.
4. Type the angle to rotate the objects in the **Angle** box. If you have previously specified values, or have created Project or Design Variables, this are listed in the drop-down menus.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A positive angle causes the object to be rotated in the counter-clockwise direction. A negative angle causes the object to be rotated in the clockwise direction.



5. Click **OK**.

   The selected objects are rotated about the axis.

To *rotate* and *copy* objects, use the **Edit> Duplicate> Around Axis** command.

Changing the Orientation of an Object

Each object has an **Orientation** property that specifies whether the coordinate system it uses is Global or a user-defined orientation relative to the Global coordinate system.

This property is useful in dealing with anisotropic materials. The properties of anisotropic materials are specified relative to each object's orientation. Changing the orientation of an object provides a way for objects made of the same material to have the material axes orientated differently.

To change an object's orientation:

1. Define the coordinate systems you want to have available.
2. Open the **Properties** dialog box for the object, or select the object and use the docked **Properties** window.
3. Click on the **Orientation** property and select the coordinate system from the drop-down menu. If no orientations other than **Global** have been defined, none appear on the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** to close the dialog box and apply the changes.

> **Note:**
>
> The object will translate and rotate to make the geometry conform to the new coordinate system. If your intention was to change the material orientation, but the object had already been drawn in the correct physical orientation, you will have to redefine it's properties. The coordinates and dimensions used to define the object geometry must be based on the same coordinate system that is needed for proper material orientation. To avoid this problem, it is best to choose the correct coordinate system for material orientation before drawing the object.

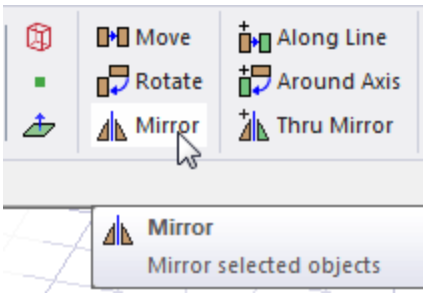
Mirroring Objects

Mirror an object about a plane using the **Edit** > **Arrange** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to move an object and change its orientation.

> **Note:**
>
> The distance between the point on the mirror plane and the point along the normal does not matter — only the vector direction matters.

To mirror an object about a plane:

1. Select the object or objects that you want to mirror. You can select multiple objects.
2. From the menu bar, click **Edit** > **Arrange** > **Mirror** or, on the **Draw** ribbon tab, click the **Mirror** icon:

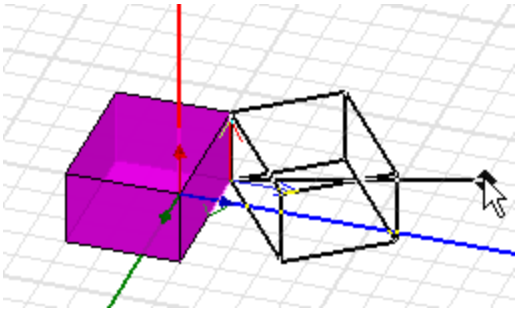


3. Select a point on the plane around which you want to mirror the object.

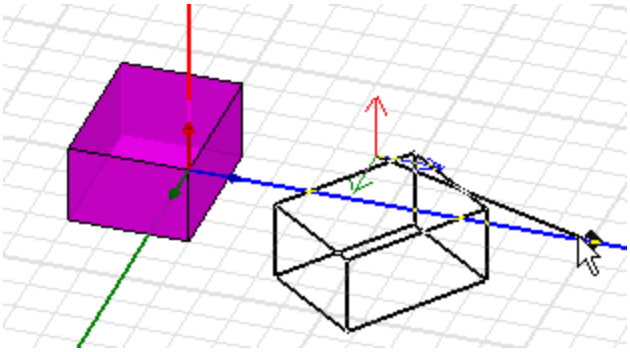
   You can do this by clicking a point, or typing coordinates in the X, Y, and Z boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select a point on the object, the mirroring is relative to that point. In the following example, the first point clicked after selecting **Edit** > **Arrange** > ***Mirror*** was on the right-rear bottom corner of the selected object. So the axis of rotation as you move the cursor is that corner. As you move the cursor, it drags a diamond-shape on a vector extending from the initial point. The distance along the vector does not matter. Moving the mouse rotates an outline of the object to new orientations. Clicking the mouse moves the object to location indicated by the outline.



In this second example, the initial point is at a distance from the original object, designated by the triad from which the handle for rotation extends to the dragging cursor.



4. Select a second point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point. As you type in the values, the outline moves to the coordinates. Press **Enter** to complete the command.

   The selected object is moved to the plane you specified and oriented relative to the normal point you specify.

To *mirror* and *copy* objects about a plane, use the **Edit** > **Duplicate** > **Mirror** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Offsetting Objects

Move every face of a 3D object in a direction normal to its surface using the **Edit** > **Arrange** > **Offset** command. The faces are moved a specified distance normal to their original planes. This command enables you to move every face of a solid object without having to individually select and move each face. Use the **Surfaces** > **Move Faces** > **Along Normal** command if you want to move just one or more faces of an object.

To offset every face of an object:

1. Select the object you want to offset.
2. From the menu bar, click **Edit** > **Arrange** > **Offset**.

   The *Offset* dialog box appears.

3. Type the distance you want to move the object faces from their origins, and then select a unit from the pull-down menu.
4. Click **OK**.

   The selected object's faces are moved the distance you specified.

Duplicating Objects

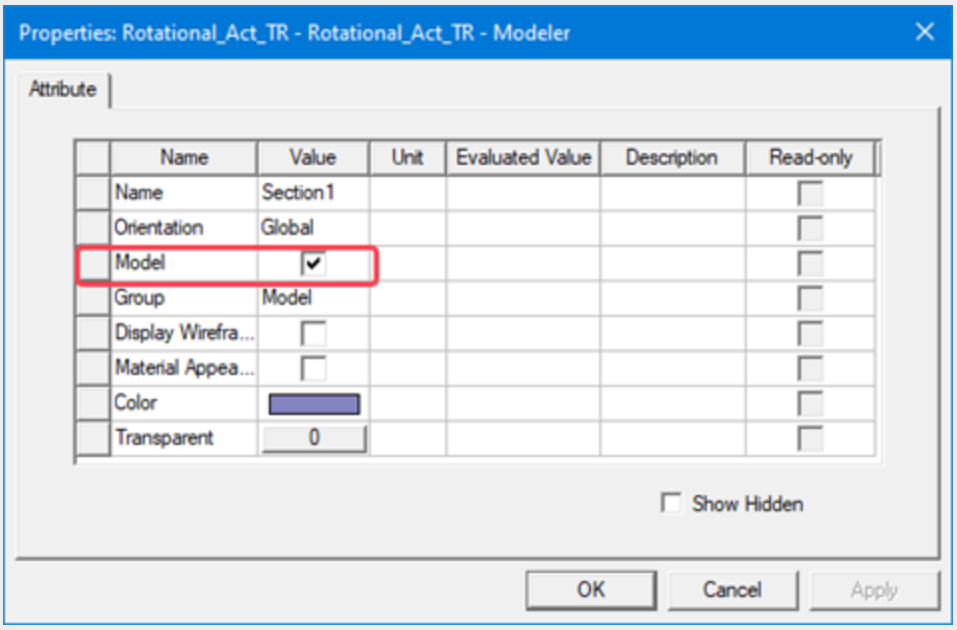
You can duplicate objects within a design using the **Edit** > **Duplicate** commands. Duplicates are dependent upon the parameters of their *parent* object at the time they were created, that is, they share the parent object's history at the time of creation. The command hierarchy in the history tree will show the duplication command, illustrating which commands affect all duplicates (those performed before the duplication) and which commands would not affect the duplicates (those performed after the duplication). For example, if you modify the radius of a parent object's hole, the change is applied to the holes of the object's duplicates because they share the radius specification history, but if you move the faces of the parent object, its duplicates are not affected because this operation took place after the duplicates were created.

Operations performed on duplicates are independent. For example, if you duplicate a cylinder twice, creating a row of three, and then split the second cylinder, the first and third cylinders are not affected by the split.

When creating duplicates, the parent object is duplicated along a line or around an axis the number of times you specify. You can also create a single duplicate that mirrors the parent object about a plane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: When you duplicate an object, the Model\Non-Model status of the new object is not copied from the parent, but is determined by the **Modeler** > **New Object Type** menu selection. You can change the Model\Non-Model status of an existing object from its Properties window:



Choose from the following commands:

Edit > Duplicate > Along Line	Duplicates the parent object along a straight line. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Around Axis	Duplicates the parent object around an axis. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Mirror	Duplicates a mirror image of the parent object about a plane.

To copy objects to another design, use the **Edit** > **Copy** and **Edit** > **Paste** commands.

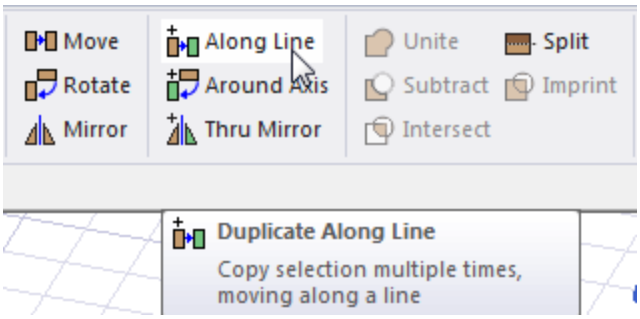
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There is currently no method for dissolving the parent/duplicate relationship once a duplicate has been created.

Duplicating Objects Along a Line

To duplicate an object along a straight line, use the **Edit** > **Duplicate** > **Along Line** command. The line along which the object is duplicated can be vertical, horizontal, or lie at an angle.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Along Line** or, on the **Draw** ribbon tab, click the **Duplicate Along Line** icon:



3. Specify the vector along which the object will be duplicated. The *Measure Data* window appears to help you choose points:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X, Y,** and **Z** boxes.

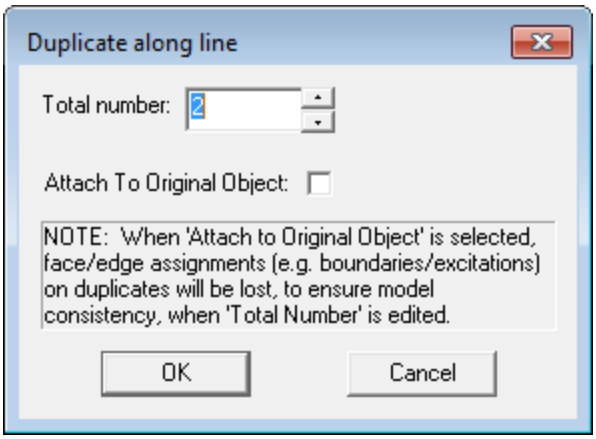
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the duplication line.

   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to duplicate the object.

      The *Duplicate Along Line* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



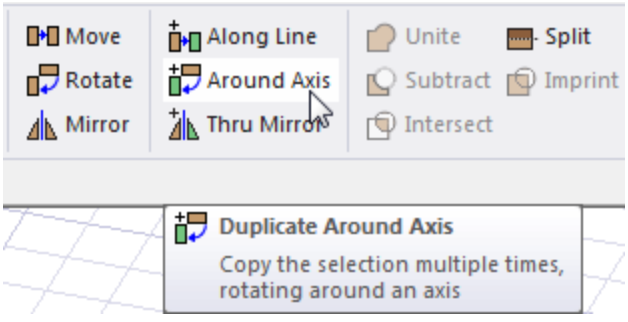
4. Type the total number of objects, including the original, in the **Total Number** box.
5. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
6. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
7. Click **OK**.

   The duplicates are placed along the vector you specified.

Duplicating Objects Around an Axis

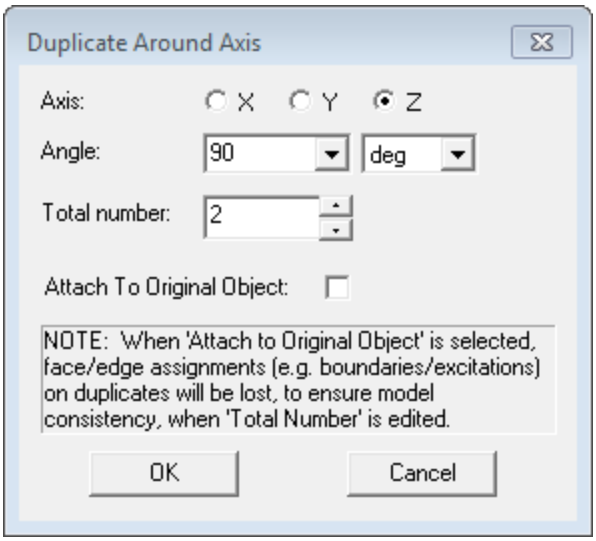
To duplicate an object around the x-, y-, or z-axis, use the **Edit** > **Duplicate** > **Around Axis** command.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Around Axis** or, on the **Draw** ribbon tab, click the **Duplicate Around Axis** icon:



The *Duplicate Around Axis* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select the axis around which you want to duplicate the object: **X, Y,** or **Z**.
4. Type the angle between duplicates in the **Angle** box.

   A positive angle causes the object to be pasted in the counter-clockwise direction.

   A negative angle causes the object to be pasted in the clockwise direction.
5. Type the total number of objects, including the original, in the **Total Number** box.
6. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
7. Click **OK**.

   The object is duplicated around the axis at the angle you specified.

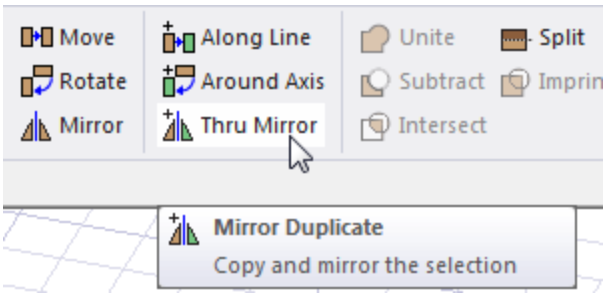
Duplicating and Mirroring Objects

To duplicate and mirror an object about a plane, use the **Edit** > **Duplicate** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to duplicate an object and specify the duplicate's position.

This command is similar to **Edit** > **Arrange** > **Mirror**, except that this command duplicates an object, rather than moves it.

1. Select the object you want to mirror.
2. From the menu bar, click **Edit** > **Duplicate** > **Mirror** or, on the **Draw** ribbon tab, click the **Thru Mirror** (Mirror Duplicate) icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select a point on the plane on which you want to mirror the object.

   A line drawn from this point to the mirror plane will be perpendicular to the plane. The distance between the point on mirror plane and point along the normal does not matter; only the vector direction matters

4. Select a normal point on the plane in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   A duplicate of the object appears on the plane you specified, oriented according to the normal point you specified.

Scaling Objects

Scale an object's dimensions in one or more directions using the **Edit** > **Scale** command.

The scale of an object is determined by the distance of each of its vertices from the origin of the model coordinate system. When an object is scaled, the distance of each vertex from the origin is multiplied by the scaling factor, causing the object to be resized and/or moved.

For example, if you specify a scaling factor of 2 in the X direction, each vertex in the model will be moved so that the distance to its origin is doubled. Note that a vertex located at the origin will not move. You can alter an object's proportions by scaling it in one direction.

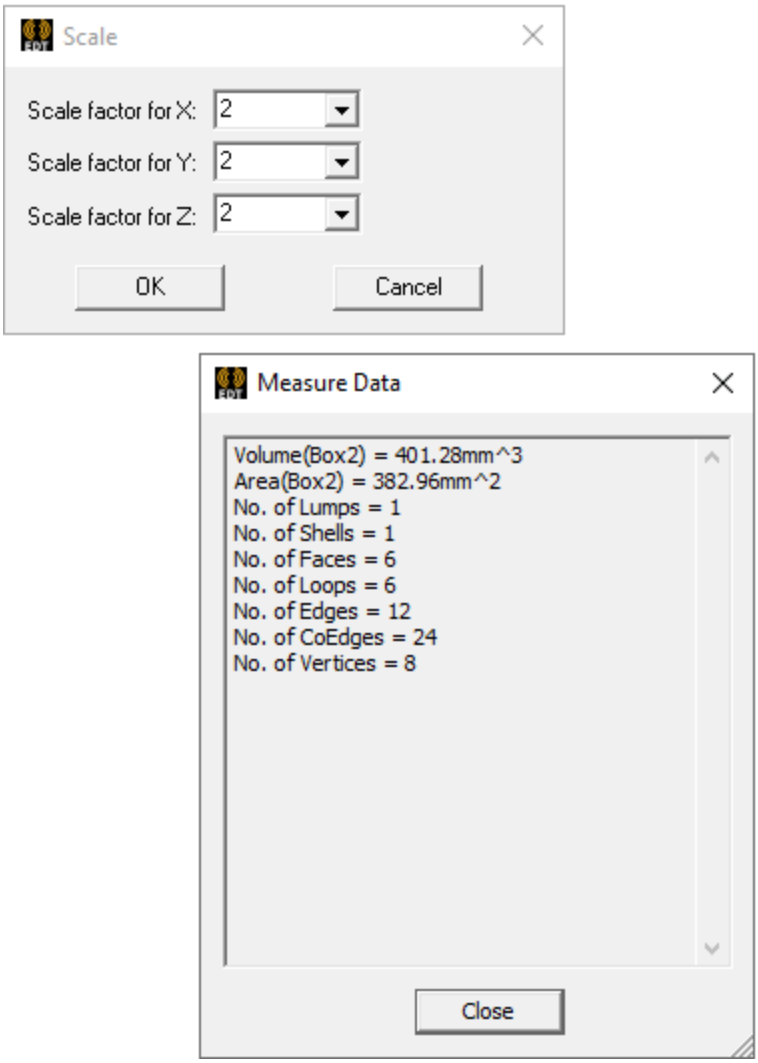
To scale an object's dimensions in one or more directions:

1. If necessary, set a different working coordinate system to achieve the desired scaling.
2. Select the object to scale.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Edit** > **Scale**.

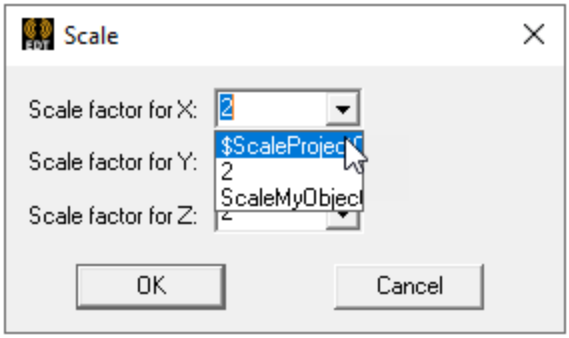
   The **Scale** dialog box appears with the **Measure Data** window.



4. Type the **Scale factor** for each axis. If you have created Project or Design Variables, or have previously used a relevant value, these may be listed for selection in the drop-down

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu.



5. Click **OK**.

   The object is scaled about the working coordinate system's origin.

Sweeping Objects

You can sweep a 2D object around an axis, along a vector, or along a path to create a 3D solid object. Objects that can be swept include circles, arcs, rectangles, trapezoids, polylines, or any 2D object created in the **3D Modeler** window. The 2D object need not be orthogonal to the sweep path.

You can also:

- Thicken sheets to make a 3D object.
- Sweep open 1D objects, such as polylines, which results in open 2D sheet objects.
- Sweep one or more faces of a 3D object to create a new 3D object. (See Sweep Faces Along Normal).

Sweeping Around an Axis

Sweep a 1D or 2D object around the X, Y, or Z axis using the **Draw** > **Sweep** > **Around Axis** command. Sweeping circles around an axis is a convenient way to create an open coil loop.

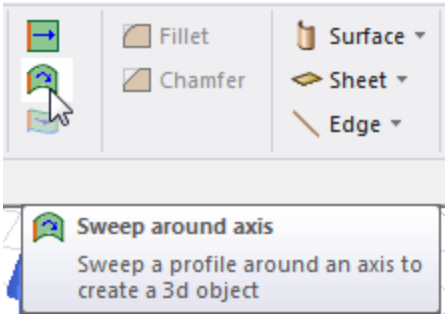
Before using this command, keep the following guidelines in mind:

- The object and the axis you are sweeping around must lie in the same plane. For example, if you are sweeping an object around the Z-axis, the object must lie in a plane that includes the Z-axis, such as XZ or YZ.
- The normal of the object's plane faces must be perpendicular to the axis around which you are sweeping.
- The object may not cross the axis around which it is being swept.

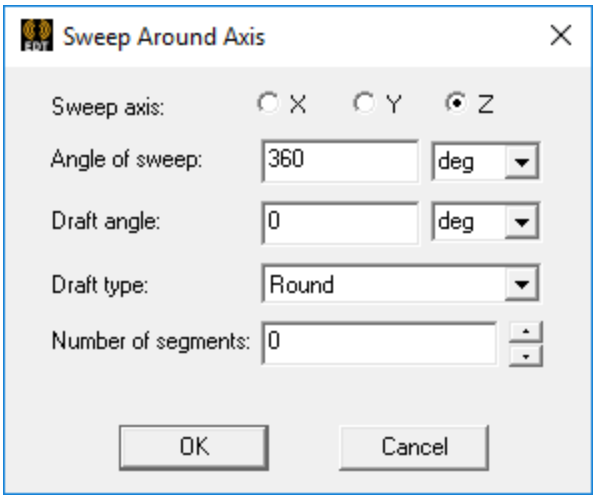
To sweep an object around an axis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to sweep.
2. From the menu bar, click **Draw** > **Sweep** > **Around Axis** or, in the **Draw** ribbon tab, click **Sweep around axis**:



The *Sweep Around Axis* dialog box appears.



3. Select the **Sweep axis** (that is, the axis you want to sweep the object around: **X, Y,** or **Z**).
4. Type an angle into the **Angle of sweep** text box and select the desired unit of measure.

   This is the angle about the specified sweep axis through which you want the object to be twisted or rotated as it is swept. The value must be between **-360** and **360** degrees.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle progressively alters the sweep radius but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept around a parallel axis with a nonzero draft angle would produce a spiral surface with a fixed width of 10 mm. When the sweep angle is *positive*, a positive draft angle produces a spiral surface with a *decreasing* radius (that is, the distance from the generated surface to the sweep

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis decrease along the sweep angle. A negative angle produces a spiral surface with an *increasing* radius along the sweep angle. When the sweep angle is *negative*, the spiral behavior is the opposite of that just described.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

   For a complete description of the behavior of each of these three options, see the Draft Types page.

7. Type the number of segments in the **Number of segments** text box and click **OK**.

   > **Note:**
   >
   > The default number of segments is zero, which creates a true path. A positive value results in a segmented sweep, while a negative value results in an error.
   >
   > If the sweep angle is 360 degrees, the number of segments is equal to the value specified. If the sweep angle is less than 360 degrees, half segments appear at the ends.
   >
   > Projects and scripts from previous software versions are treated as if the number of segments were zero.

   The object is swept around the axis. The new object has the properties of the original object. The *Properties* dialog box appears, enabling you to modify the object's properties.

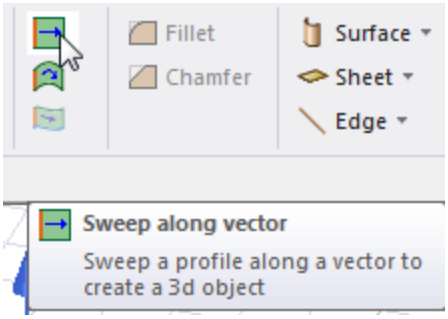
8. Click **OK**.

Sweeping Along a Vector

Sweep a 1D or 2D object along a vector using the **Draw** > **Sweep** > **Along Vector** command.

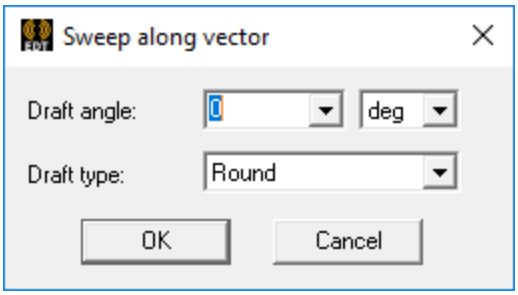
1. Select the object you want to sweep.
2. Click **Draw** > **Sweep** > **Along Vector** or, in the **Draw** ribbon tab, click **Sweep along vector**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Draw the vector you want to sweep the object along:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** boxes.
   b. Select the endpoint in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the start point in the **dX, dY,** and **dZ** boxes, where **d** is the relative distance from the previously selected point.

   The *Sweep along vector* dialog box appears.



4. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep vector.

5. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Natural**

For a complete description of the behavior of each of these three options, see the Draft Types page.

6. Click **OK**.

   The object is swept along the vector. The new object has the name and color of the original profile. The *Properties* dialog box appears, enabling you to modify the object's properties.

Sweeping Along a Path

Sweep a 1D or 2D object along a path that is defined by an open or closed polyline using the **Draw** > **Sweep** > **Along Path** command.

Best Practices

Adhere to the following best practices for good sweep outcomes using the Parasolid geometry kernel:

- When sweeping along a path that is not closed (i.e., the start and end point are not the same), ensure that the start point of the path is on the profile being swept.
- When sweeping a profile around a closed path, ensure that the path intersects the profile.
- Parasolid sweep operator preserves the angle between the profile and path. Ensure that the profile is perpendicular to the path if it is supposed to be perpendicular.
- If the sweep profile is rotationally symmetric (e.g., circle, rectangle, square), position the profile so that the path intersects the profile at the center of the profile. If the profile does not have a center, allow the path to go through or start at the vertex of a profile.

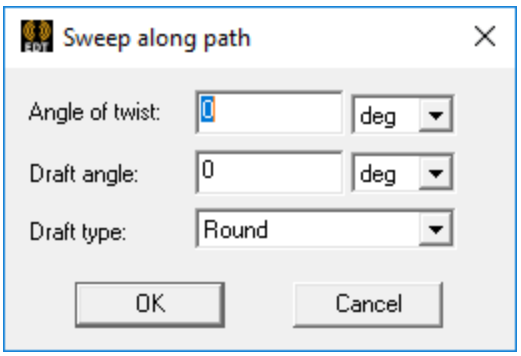
> **Note:**
>
> If the path curve is tangential to the sweep profile, then the sweep will fail.

To sweep an object along a path:

1. Create the polyline to be used as a path.
2. Select the object you want to sweep and then select the new polyline.
3. Click **Draw** > **Sweep** > **Along Path** or, from the **Draw** ribbon tab, click the **Sweep along path** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Sweep along path** dialog box appears.



4. Type an angle in the **Angle of twist** text box and choose the desired unit of measure.

   This is the angle you want the profile to rotate about the sweep path vector as it is swept through the complete path.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want the object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep path.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a complete description of the behavior of each of these three options, see: Draft Types.

7. Click **OK**.

   The object is swept along the path. The polyline object used as the path is deleted. The new object has the properties of the original object. The **Properties** dialog box appears, enabling you to modify the object's properties.

Sweeping Faces Along Normal

To create a new object by sweeping select 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler** > **Surface** > **Sweep Faces Along Normal** command. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners.

To sweep selected object faces in a normal direction:

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.

   Alternatively, with the Modeler window active, press **F** to switch to the *Face* selection mode.

2. Select the faces of the object you want to sweep.
3. From the menu bar, click **Modeler** > **Surface** > **Sweep Faces Along Normal**.

   The *Sweep Faces Along Normal* dialog box appears.

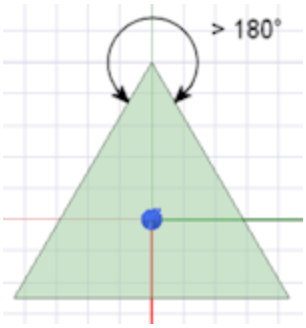
4. Type the distance you want to sweep the object face from its origin.
5. Click **OK**.

   The face is swept the distance you specified to create a new 3D object.

Draft Type Options

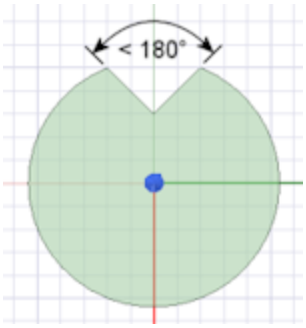
In many cases, the **Draft type** setting may have little to no effect on the resultant geometry of a swept object unless a *positive, non-zero draft angle* is specified. Consider a *sharp corner* feature. For the purpose of this discussion let's define a sharp corner as any protruding feature where the *external* angle between the adjacent faces is greater than 180-degrees:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When this type of feature is expanding, the corner can gradually become rounded. However, if the same corner is contracting, the radius is already zero at the start of the sweep and cannot contract or decrease further. Therefore, no rounding occurs in the latter case.

Now, consider a *groove* in the perimeter of a 2D object to be swept. Let's define a groove as any indented feature, where the *external* angle between the adjacent faces is less than 180-degrees:



A groove will grow in size as the object is swept with a *negative* draft angle specified. In this case, progressive rounding of the corners is possible.

You may need to experiment with the three draft type options to determine what the effect will be for nontrivial object shapes. The behavior of each option can differ according to the shape of the 2D object, the draft angle, and whether it is swept along a vector, a path, or about an axis.

The following three **Draft type** options are available:

- **Natural:** This draft type generally maintains the shape of the original object as it is swept along the specified direction. That is, all sharp corners remain sharp, producing sharp edges in the sweep direction. There are exceptions to this general behavior. Rounding of certain corners may occur if required to accommodate sides with complex shapes (for example, a complex side might be split into a combination of planar, cylindrical, helical, or conic-section faces).
- **Extended:** This option maintains the shape of the original object as it is swept along the specified direction. Sharp corners of the original object remain sharp corners along the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

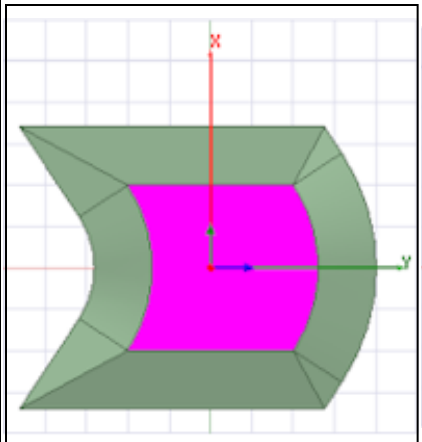
entire swept direction. However, the side faces may be split into three separate tangential faces, as needed and as described in the preceding bullet.

- **Round:** This option rounds all sharp corners of the original object, when possible, as it is swept along the specified direction.

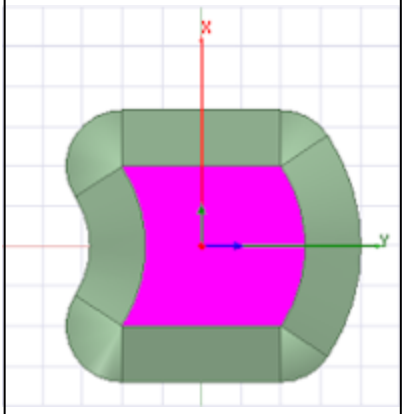
Examples

The images below compare the results of sweeping an object using each of the available **Draft type** options. The first six images are of an object swept with a positive draft angle defined. The final three images are of a different object swept with a negative draft angle defined. For all examples, the original 2D object is highlighted in magenta.

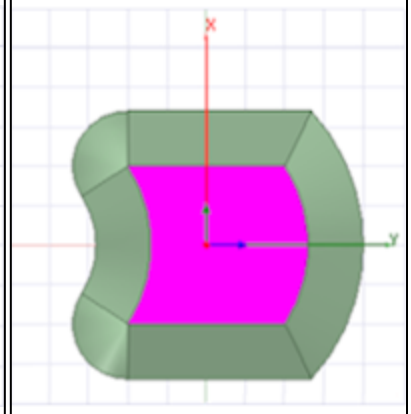
Sweep Along Vector (Positive Draft Angle):



1a: Extended

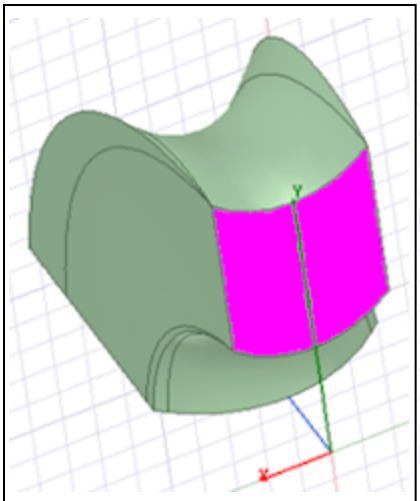


1b: Round

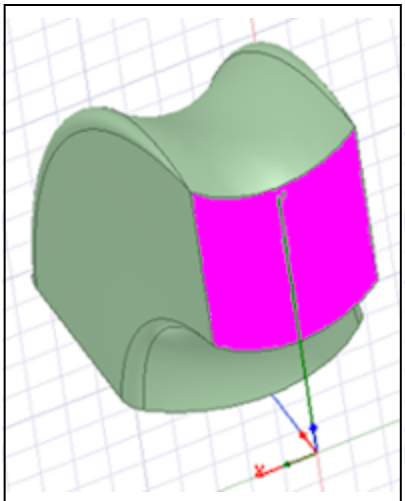


1c: Natural

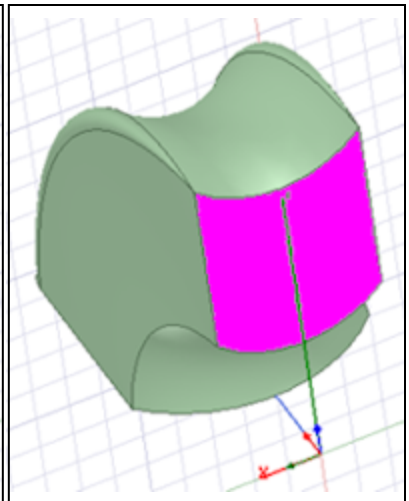
2. **Sweep About Axis** (Positive Draft Angle):



2a: Extended



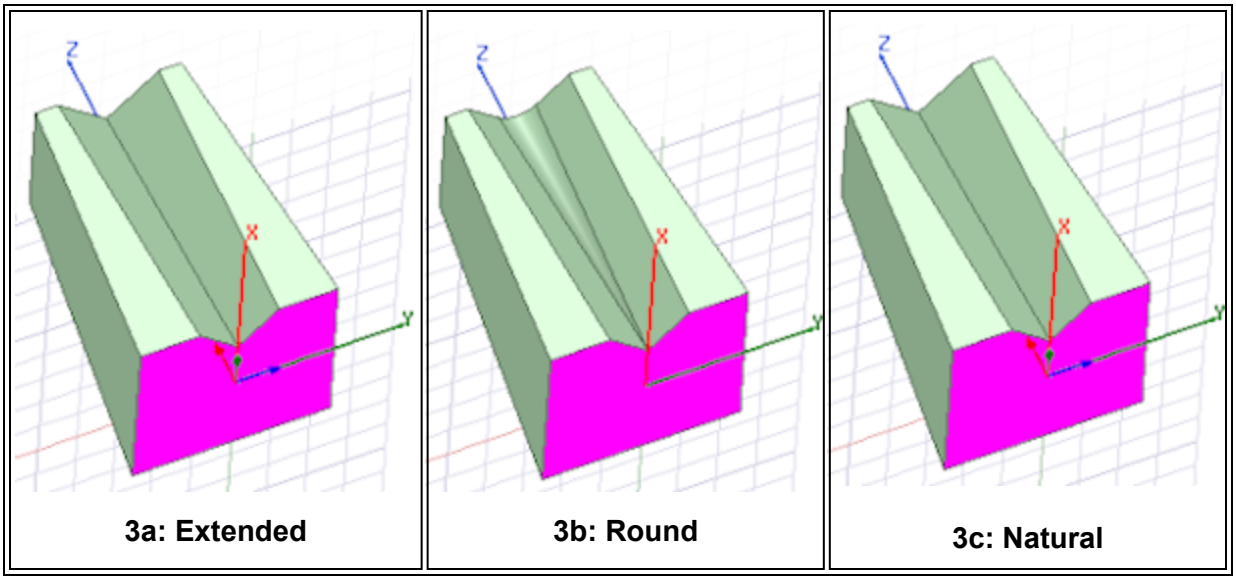
2b: Round



2c: Natural

3. **Sweep Along Vector** (Negative Draft Angle):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Observations:

- **Extended Option:** Notice how two of the four sides of example **1a** have been divided into three faces each (the sides generated by sweeping curved edges). However, the other two sides (those generated from straight edges) have not been divided.

  In example **2a**, all four sides have been divided into three faces each. Even the straight sides generate a complex surface due to the combination of a positive draft angle and a circular sweep path.

  In example **3a** (negative draft angle), no face is split and no edge is rounded.

- **Round Option:** In both examples **1b** and **2b**, all corners of the original 2D object have been progressively rounded (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

  In example **3b** (negative draft angle), only the corner of the groove has been rounded. The indented feature is the only one increasing is size along the sweep direction.

- **Natural Option:** In both examples **1c** and **2c**, the corners adjacent to the convex curved face remain sharp. However, the corners adjacent to the concave curved face are progressively rounded in both cases (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

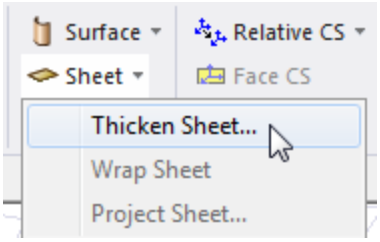
  In example **3c** (negative draft angle), no face is split, and no edge is rounded.

Thicken Sheet

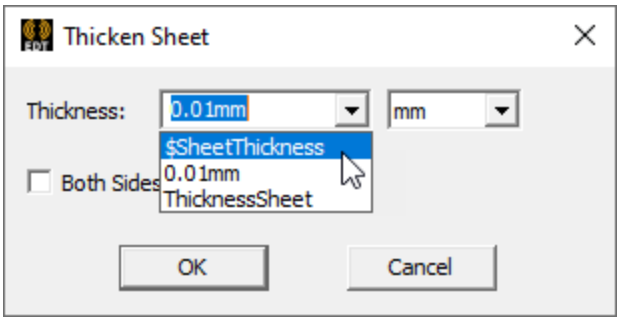
To thicken one or more sheet objects to make 3D objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the sheet or sheets you want to thicken into 3D objects.
2. From the menu bar, click **Modeler** > **Surface** > **Thicken Sheet** or, on the **Draw** ribbon tab, choose **Sheet** > **Thicken sheet**:



The *Thicken Sheet* dialog box appears:



3. Specify the **Thickness**. If you have set Project or Design variables with relevant values, or have previously assigned values, this are listed in the dropdown.
4. If necessary, specify the units by selecting from the drop-down menu.
5. If you want to thicken symmetrically (half of the thickness on each side of the sheet or sheets), select the **Both Sides** check box.

   The resulting solid is the requested thickness, but the original sheet location corresponds to the middle of the resulting solid's thickness.

6. Click **OK**.

   The dialog closes and the sheets are changed into 3D objects of the desired thickness.

Wrap Sheet Command

You can use **Modeler** > **Surface** > **Wrap Sheet** command to wrap a sheet object around a suitable 3D object. The sheet object does not have to be in contact with the 3D object. It should have smaller dimensions than the 3D object.

When the sheet to be wrapped and the target body do not touch, the wrap sheet command chooses among several different methods of bringing them into contact.

- If an edge from the sheet body and the target body are close enough and at a small angle (10 degrees), the edges are brought into alignment before the wrap.

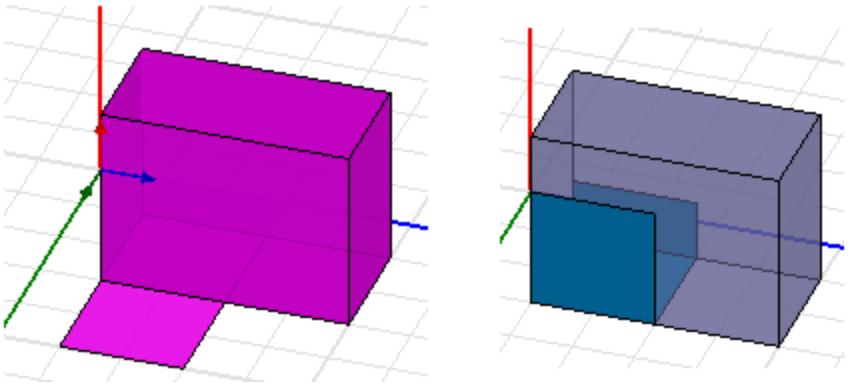
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a vertex from the sheet body and an edge on the target body are close enough, the vertex is moved to the target edge.
- If the previous criterion are not matched, a vertex from the sheet body is moved to the closest face of the target body.

If the sheet to be wrapped and the target face are both planar and have the same or opposite normal, the closest vertex from the sheet body is moved to the target face. No moving to match or align edges is performed.

If the sheet body has multiple lumps, a single rectangle is fitted around all of the lumps. The rectangle is moved to the target surface using the above procedure.

If the sheet object does not overlap the corners of the 3D object, the wrap is straightforward, as shown in the following figure:



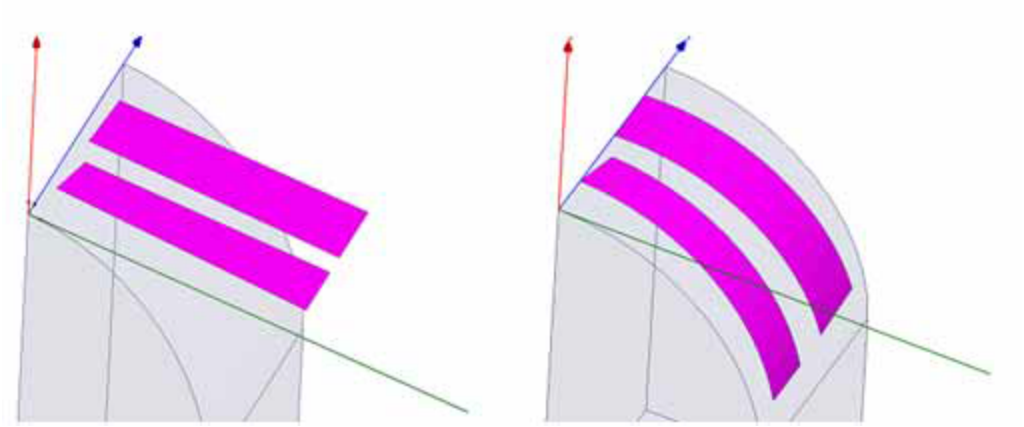
If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

A sheet object that overlaps corners may not wrap in straightforward fashion, depending on both the angle(s) involved, and the sheet object. While it is possible, it is not recommended.

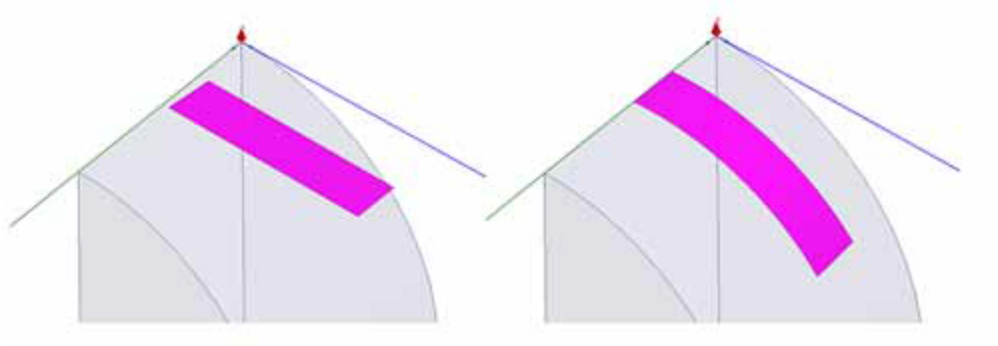
Examples:

The following figure illustrates wrapping on a target surface with the snap to an edge. The surface will snap to an edge if the distance is close relative to the length of the edge. If the angle between the edge is less than 10 degrees, the edge to be wrapped will be aligned to the target body. In the example on the left, the lower sheet has an edge angle more than 10 degrees. The vertex will be snapped to the edge. The top sheet in the picture has an edge angle of less than 10 degrees. The vertex is snapped to the target edge and the sheet edge is aligned to the target. To avoid snapping to an edge or edge alignment, the wrap sheet can be put into contact with the target surface. In that case the sheet vertex will not be moved to the target edge unless it is within modeling tolerances.

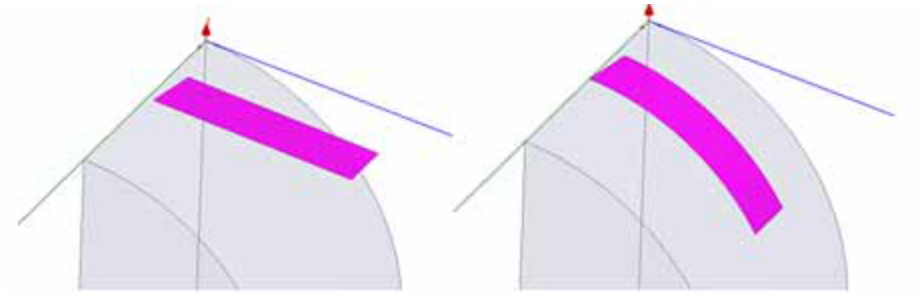
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move the edge to the edge before wrapping.

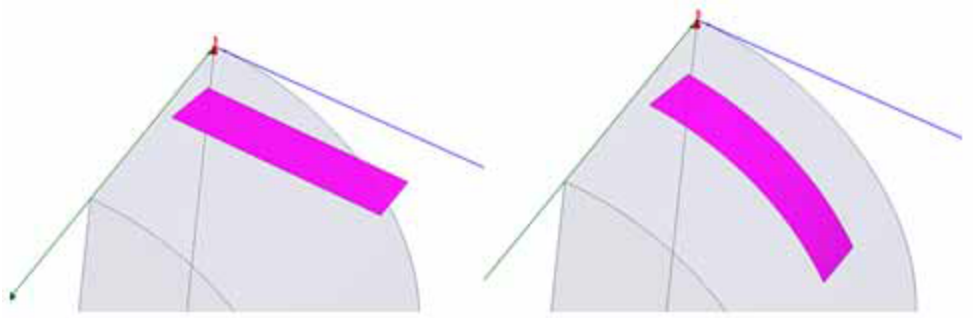


The following figure shows the situation when you move a vertex to an edge before wrapping.

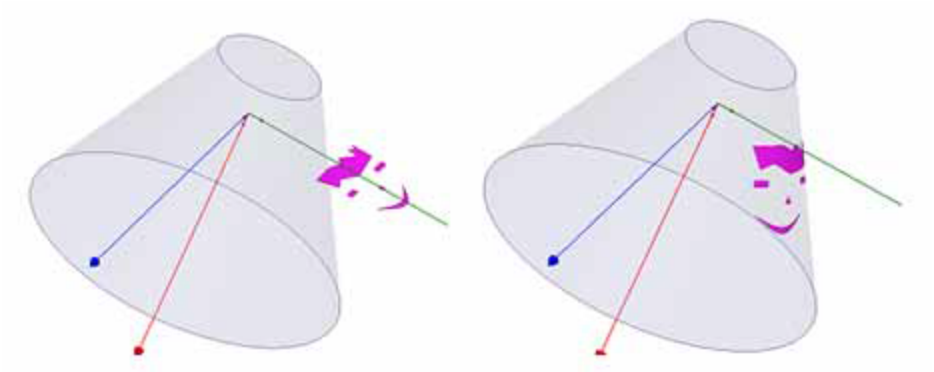


The following figure shows the situation where you move a vertex to a face before wrapping.

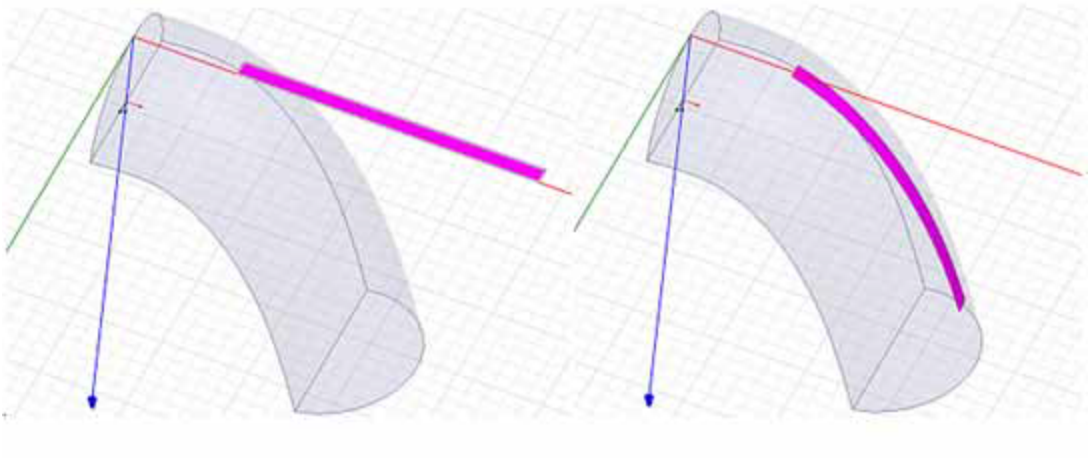
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move a multilump sheet to touch the target body. A rectangle that encloses all lumps is used to position the sheet on the target.

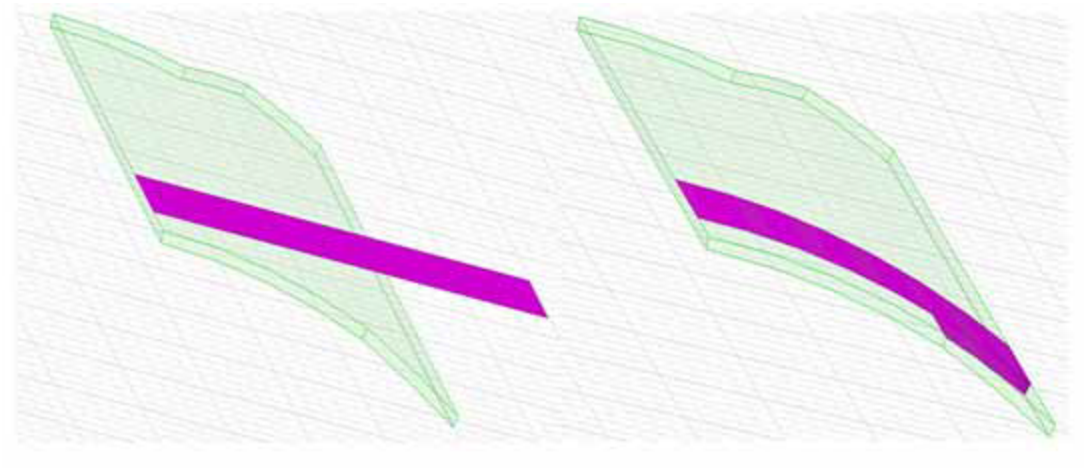


The next figure illustrates wrapping to a non-developable surface. The area of the sheet changed from 0.7000 mm^2 before wrapping to 0.6986 mm^2 after wrapping.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure illustrates wrapping on a target surface which has tolerant edges. The tolerant edge is at the beginning of the wrap. The tolerant edge is not evident visually.



Limitations/Error Conditions:

- Wrapping on non-developable surfaces creates distortion in lengths.
- Wrapping is not allowed across curved boundaries between faces.
- Working with tolerant edges is successful only if the gaps can be tightened sufficiently for the operation.
- Wrapping on the pole of a surface (such as sphere or cone) is not supported. A warning is given:

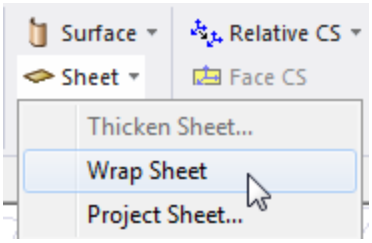
  WrapSheet: Wrapping sheet contacted pole of target surface. Recommend moving sheet and splitting out pole from target surface.

- The choice of method for how the sheet body is moved to touch the target face is not directly selectable by the user. If the sheet to be wrapped and target body touch, then the sheet body is only rotated at point of contact.

To wrap a sheet object:

1. Create a sheet close to an appropriate 3D object.
2. Select both objects.
3. From the menu bar, click **Modeler** > **Surface** > **Wrap Sheet** or, on the **Draw** ribbon tab, select **Sheet** > **Rap Sheet**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The sheet object wraps around the 3D object. You can select the wrapped sheet object and the 3D object separately, and assign properties separately.

If the object cannot wrap, the Message window contains a warning and description.

You can wrap multiple sheets on the same 3D object.

If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

You can use the **Tools** > **Options** > **General Options**, 3D Modeler options to automatically perform a **Modeler** > **Boolean** > **Imprint** command after performing a **Wrap.**

Covering Lines

To cover a closed 1D polyline object with a face, use the **Modeler** > **Surface** > **Cover Lines** command. The polyline object becomes a 2D sheet object.

To convert a closed polyline object to a sheet object:

1. Select the closed polyline object you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Lines**.

   The object is now covered. It is now a 2D sheet object that can be swept to form a 3D solid object.

> **Note:**
>
> If you want the modeler to automatically cover all closed polyline objects you draw, including circles, ellipses, rectangles, and regular polygons, select the **Automatically cover closed polylines** option in the *3D Modeler > Drawing* section of the *General Options*. A closed polyline object can also be created by using boolean unite operations on two or more polylines.

Covering Faces

To cover object faces, the faces must be united into a 3D sheet object. To cover the face of a 2D or 3D object, use the **Modeler** > **Surface** > **Cover Faces** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Covering the face of an open 2D sheet object that had previously been uncovered results in a 3D solid object. For example, for a box, when you select and uncover a face, the solid box becomes a sheet with five faces. When you then select that sheet body box and use the **Cover Faces** command, the box becomes a solid again with six faces.

To cover the faces of objects:

1. Select the faces of the objects you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Faces**.

   The object faces are now covered.

Uncovering Faces

Uncover a surface of a 3D object using the **Modeler** > **Surface** > **Uncover Faces** command. Uncovering the surface of a 3D solid object results in an open 2D sheet object.

To uncover the face of a 3D object:

1. Switch to the *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select a face of the object you want to uncover.

   From the menu bar, click **Modeler** > **Surface** > **Uncover Faces**. The selected face is uncovered, leaving an open face on the object.

> **Note:**
>
> You can uncover one face of a 3D object at a time. If you select multiple faces, only the first face will be uncovered.

Detaching Faces

The **Modeler** > **Surface** > **Detach Faces** command enables you to remove the face of a 3D object, resulting in two separate objects.

To detach the face of an object:

1. Switch to *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select the face of the object you want to detach. You can select multiple faces to detach.

   Click **Modeler** > **Surface** > **Detach Faces**. The selected face is now detached, resulting in two 2D sheet objects.

Detaching Edges

The **Modeler** > **Edge** > **Detach Edges** command enables you to remove an edge of a wire object, resulting in two separate wire objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To detach an edge of an object:

1. Switch to *Edge* selection mode: Click **Edit** > **Selection Mode** > **Edges**.
2. Select the edge of the object you want to detach. You can select multiple edges to detach.
3. Click **Modeler** > **Edge** > **Detach Edges**.

   The selected edge is now detached, resulting in multiple wire objects.

> **Note:**
>
> Only edges from wire bodies can be used in a detach edge operation.

Creating a Cross-Section

You can take a cross section of a 3D object to create a new 2D object. This is done using the **Modeler** > **Surface** > **Section** command.

Use this command to create cross sections of 3D objects on the XY, YZ, or XZ plane. The cross sections are created as 2D closed polyline objects.

To create a cross section of an object:

1. Make sure the working coordinate system you want to use for the cross section plane is set.
2. Select the object from which you want to create a cross section.
3. From the menu bar, click **Modeler** > **Surface** > **Section**.

   The *Section* dialog box appears.

4. Select the **Section Plane** you will use to divide the object: **XY, YZ,** or **XZ**.
5. Click **OK**.

   A closed polyline object is created at the intersection of the 3D object and the specified section plane. The original 3D object is unmodified.

Connecting Objects

Use the **Modeler** > **Surface** > **Connect** command to perform the following operations:

- Connect two or more 1D polyline objects. Ansys Electronics Desktop will modify the first polyline you select to be a 2D sheet object that connects to the second and any subsequently selected polylines. The second and subsequent polylines selected are deleted.
- Connect two or more 2D sheet objects. Ansys Electronics Desktop will modify the first 2D object you select to be a 3D solid object that connects to the second and any subsequently selected objects. The second and subsequent objects selected are deleted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

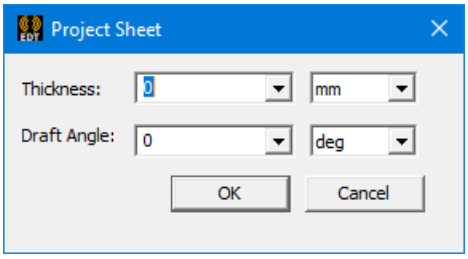
To connect objects:

1. Select the objects you want to connect. Select 3D objects as objects, not as faces.
2. Click **Modeler** > **Surface** > **Connect**.

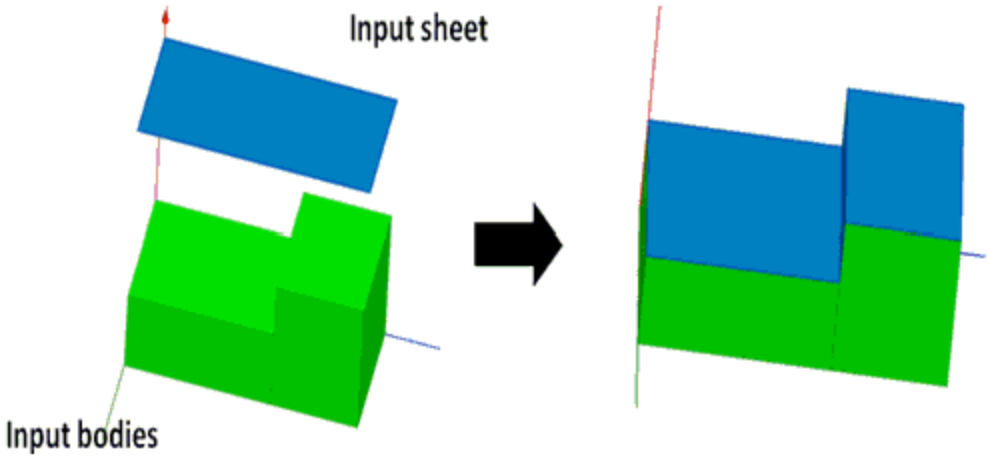
   A new object is created that connects the objects you selected. The first object you selected was modified to create the new object and all subsequently selected objects were deleted.

Project Sheet Object

Use the **Modeler** > **Surface** > **Project Sheet** command to apply a selected planar sheet object to a suitable 3D object. This permits easy modeling of thin conformal deposits. (The **Wrap Sheet** and **Imprint Projection** commands are not suitable for this use.) For modeling conformal deposits when using **Project Sheet**, you can specify a thickness and a draft angle.

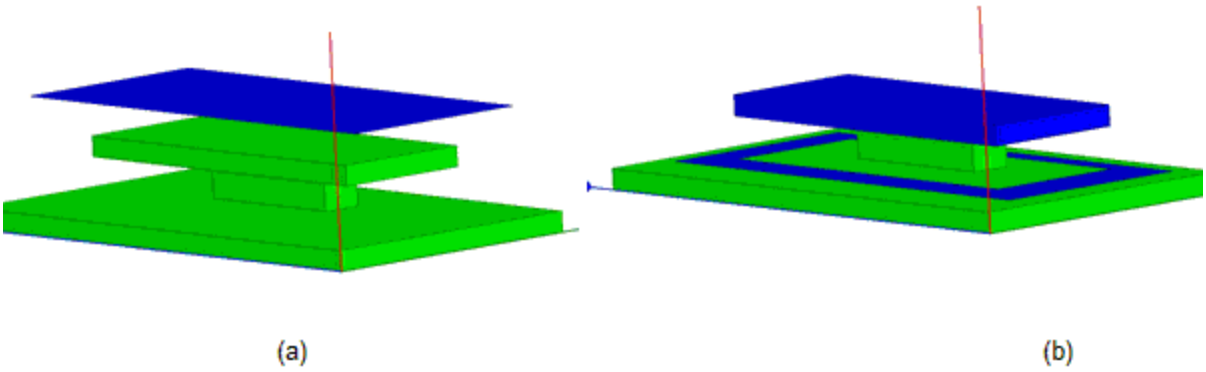


To use the **Project Sheet** command you must select one planar sheet and at least one solid body. If you select multiple solids, the sheet is project on all bodies as if they have been united. After the commend executes, a new sheet that lies on the surface of the selected bodies is created. The solid bodies are not changed.

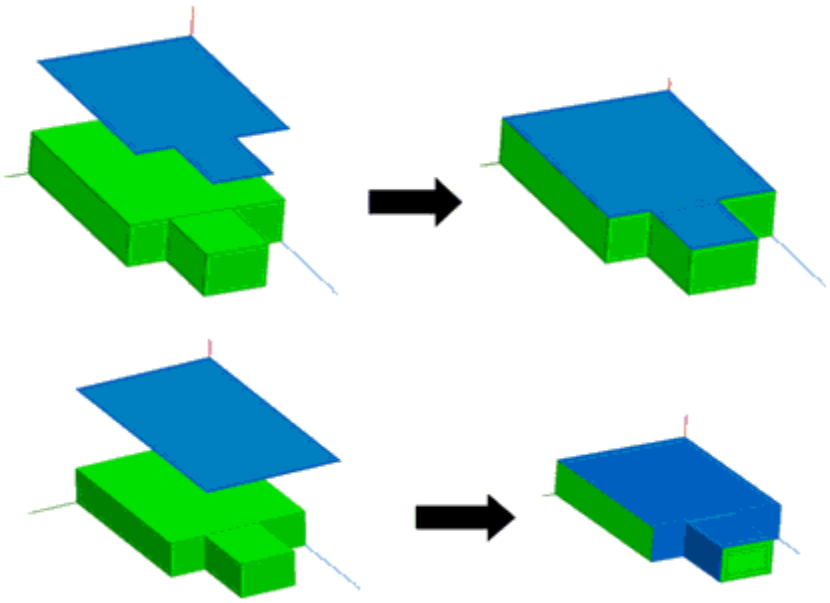


Faces that are hidden from the sheet being projected are not covered, due to the projected sheet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



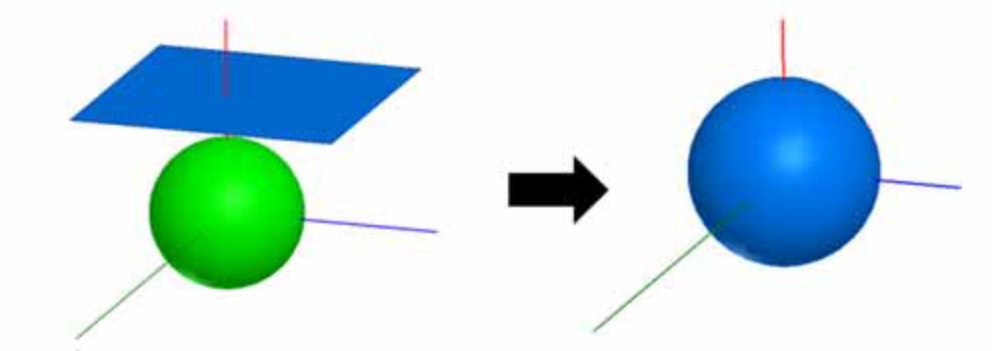
Only vertical faces (or faces parallel to the projection direction) that lie completely within the projection of the sheet will be covered by the projected sheet. Vertical faces that lie on the boundary of the projection are not covered.



If the sheet cross-section exactly matches the body cross-section, no vertical faces are covered. If the projected sheet exceeds the cross-section of solids, faces that lie with in the sheet projection are covered.

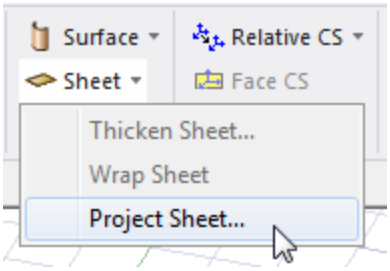
If a surface like a cylinder or sphere lies completely within the sheet projection, the entire surface will be covered, though there may be a warning that “Output may be incorrect because a self-obstructing face was found.”

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Project a Sheet:

1. Select a planar sheet object and one or more appropriate solid objects.
2. From the menu bar, click **Modeler** > **Surface** > **Project Sheet**, or on the **Draw** ribbon tab, click **Sheet** > **Project Sheet**:

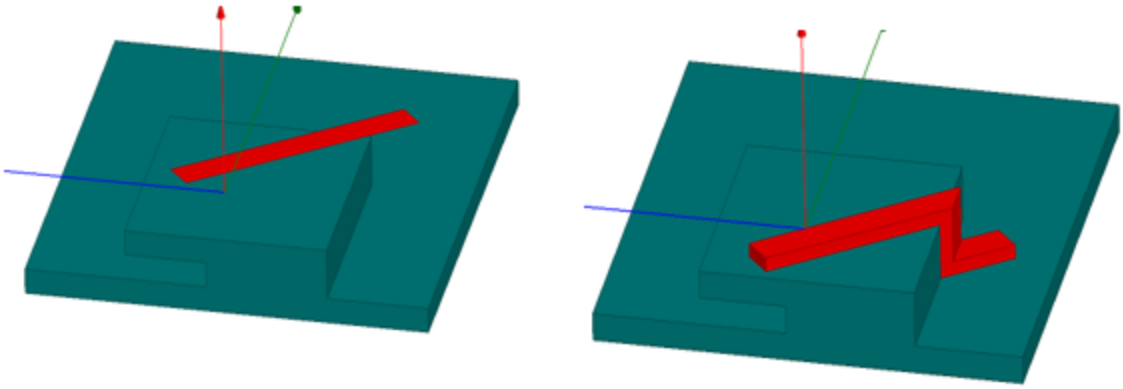


   For modeling conformal deposits, you can also specify a thickness and optional draft angle.

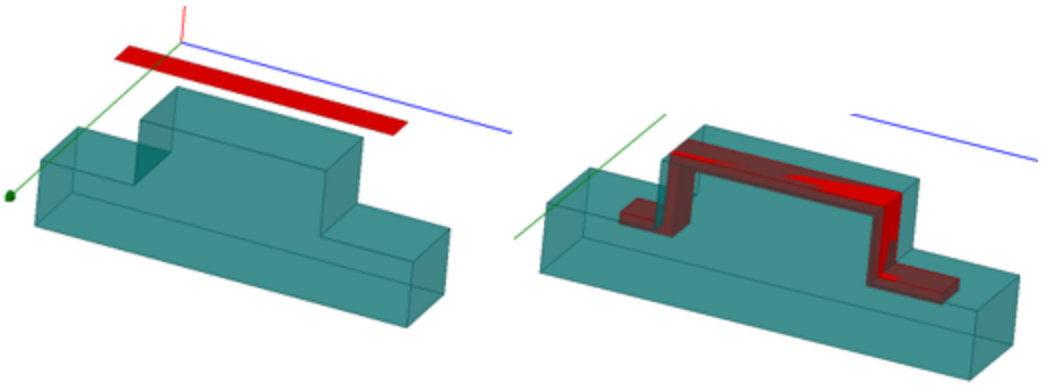
After you perform **Project Sheet**, the History Tree shows the **Project Sheet** command and the create command for the imprinted object. If you select the **Project Sheet** command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the Create <object> icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting.

Specifying a Thickness value has different effects depending on whether the value is positive or negative. The following figure shows the effect of a positive thickness, with the Projected sheet taking its thickness outward from the target object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

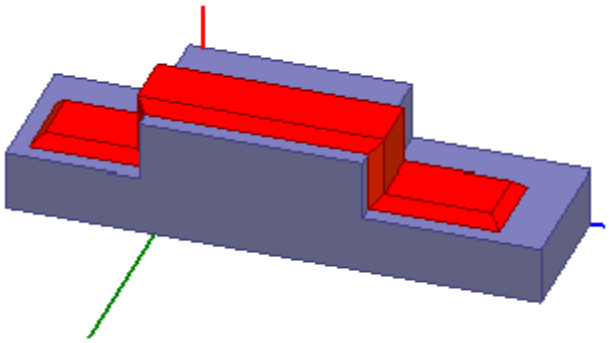


The following figure shows the effect of a negative value for thickness.



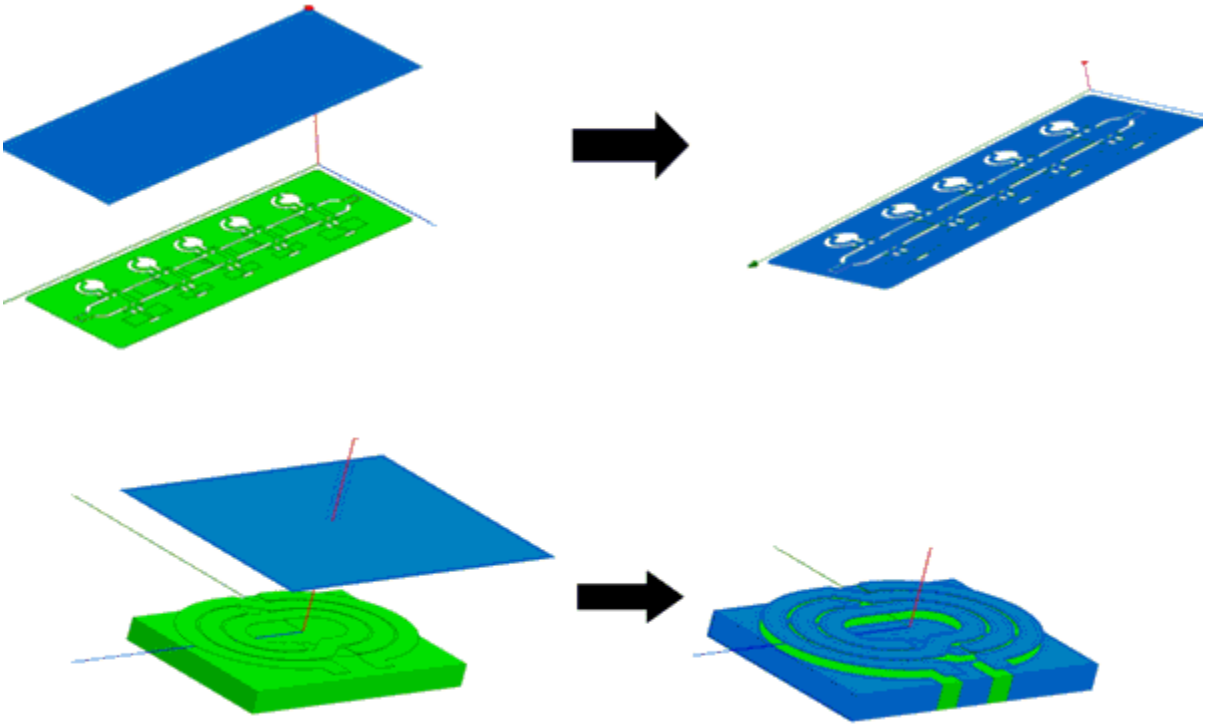
When a negative value is given the thickness will be limited by the target solid body. That is, the thickness can not be beyond the original solid body which was chosen. If you use a negative thickness it is also possible that not all overhangs will be eliminated.

The following figure shows the effect of a 45 degree draft angle.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other examples of Project Sheet on complex models are show in the following figure.



Moving Faces or Edges

You can move the faces of a 3D object in a normal direction using the **Modeler> Surface> Move Faces** commands. Moving object faces enables you to resize, reshape, or relocate an object.

Moving Faces Along the Normal

To move a 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler> Surface> Move Faces> Along Normal** command. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners, as shown below.

To move an object face in a normal direction:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

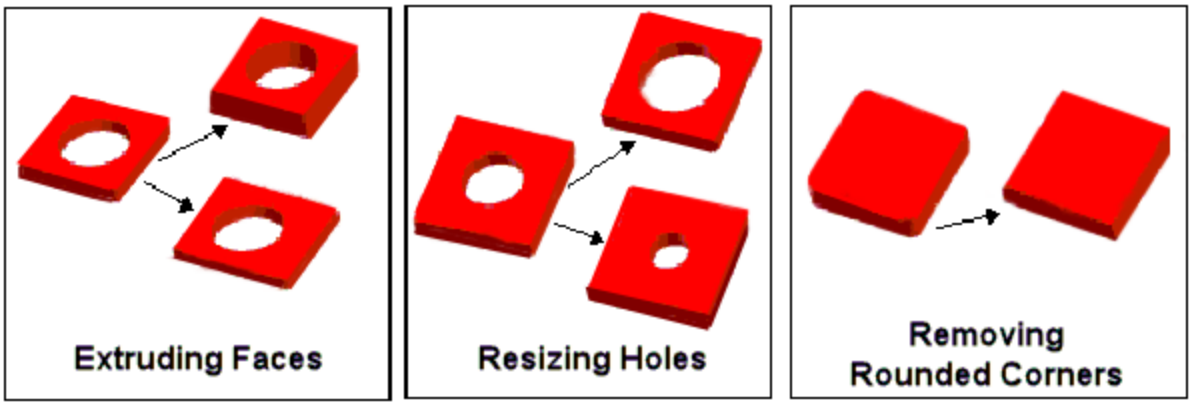
   If you have created a suitable face list, right-click the list and click **Select Assignment** from the shortcut menu. You can operate on faces in the list.

3. Click **Modeler> Surface> Move Faces**> **Along Normal**.

   The *Move Faces Along Normal* dialog box appears.

4. Type the distance you want to move the object face.
5. Click **OK**.

   The face will be moved the distance you specified.



To move every face of an object normal to its surface, use the **Edit> Arrange> Offset** command.

Moving Faces Along a Vector

To move the faces of a 3D object a specified distance along a vector use the **Modeler> Surface> Move Faces> Along Vector** command. Each selected face is moved along the vector, normal to its original plane. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for relocating holes in an object, as shown below.

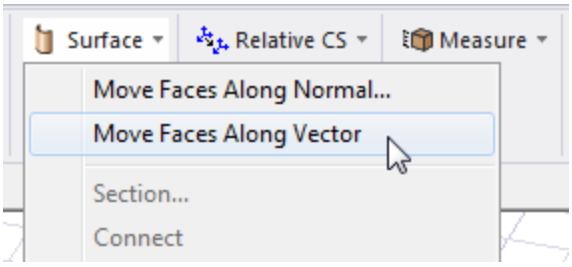
To move an object face along a vector:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

   If you have created a suitable face list, right-click the list and click **Select Assignment** from the shortcut menu. You can operate on faces in the list.

3. From the menu bar, click **Modeler> Surface> Move Faces> Along Vector**, or, on the **Draw** ribbon tab, click **Surface> Move Faces Along Vector**:



4. Specify the vector along which the face will be moved:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the **X, Y,** and **Z** boxes.

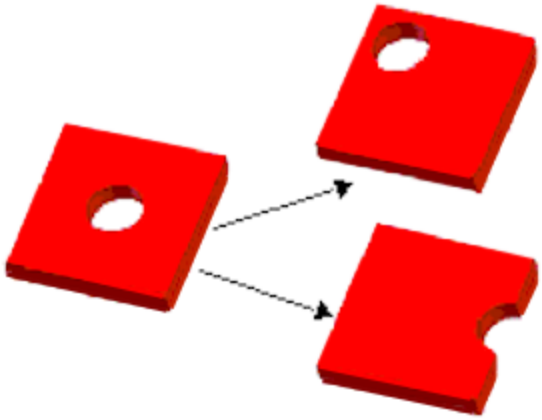
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the vector.

   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to move the face.

      The face is moved along the vector you specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Relocating Holes

To move every face of an object normal to its surface, use the **Edit>Arrange>Offset** command.

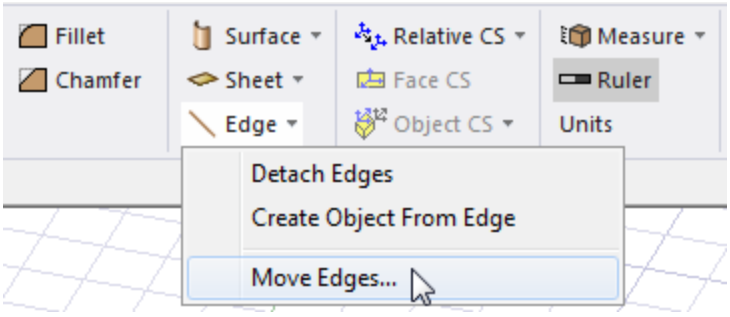
Moving Edges Along Normal

To move a 2D object's edge a specified distance in a direction normal to its original plane, use the **Modeler> Edge> Move Edge** command. The edge is extended or shortened along its own plane. Note that the adjoining faces will not be sheared or bent. The edge can be on a rectangle, an ellipse, a circle, a regular polygon, or an equation based surface.

This command is useful for extending or shrinking faces and resizing holes.

To move an object edge in a normal direction:

1. Right-click in the Modeler window and choose **Select Edge** on the shortcut menu.
2. Select the edge of the object you want to move.
3. From the menu bar, click **Modeler> Edge> Move Edge** or, on the **Draw** ribbon tab, click **Edge> Move Edges**:



The *Move Faces Along Normal* dialog box appears.

4. Type the distance you want to move the selected edge.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

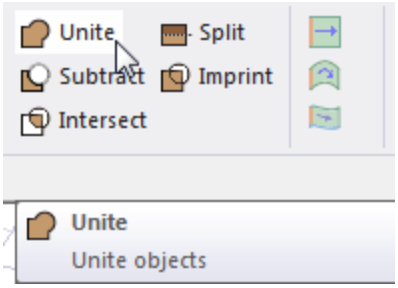
5. Click **OK**.

   The edge of the object is moved based on the value you specified.

Uniting Objects

To join two or more objects into one object, use the **Modeler> Boolean> Unite** command. The new object has the name, color, boundary, and material assignment of the first object selected. The objects are united at the point of intersection.

1. Select the objects you want to join.
2. From the menu bar, click **Modeler> Boolean> Unite** or, on the **Draw** ribbon tab, click the **Unite** icon:



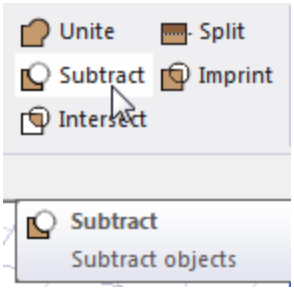
   The objects are united.

> **Note:**
>
> By default, the objects being joined to the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being joined to the first object selected, do one of the following:
>
> - Copy the objects, and then paste them back into the design after uniting them.
> - Select **Clone tool objects before uniting** in the *3D Modeler> Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original objects being joined.

Subtracting Objects

1. Select the object from which you want to subtract other objects.
2. Hold down the **Ctrl** key and select the objects you want to subtract.
3. From the menu bar, click **Modeler**> **Boolean**> **Subtract** or, on the **Draw** ribbon tab, click the **Subtract** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Subtract* dialog box appears.

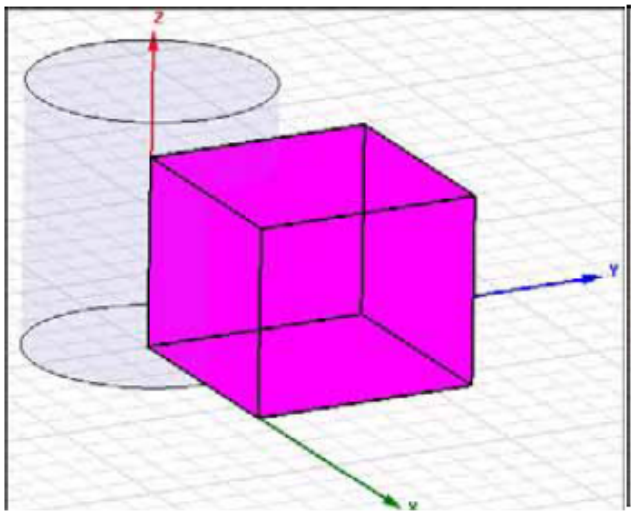
Objects listed in the *Tool Parts* list will be subtracted from the object or objects listed in the *Blank Parts* list.

4. Optionally, select an object name in either list and use the left and right arrow buttons to move the object name to the opposite list.

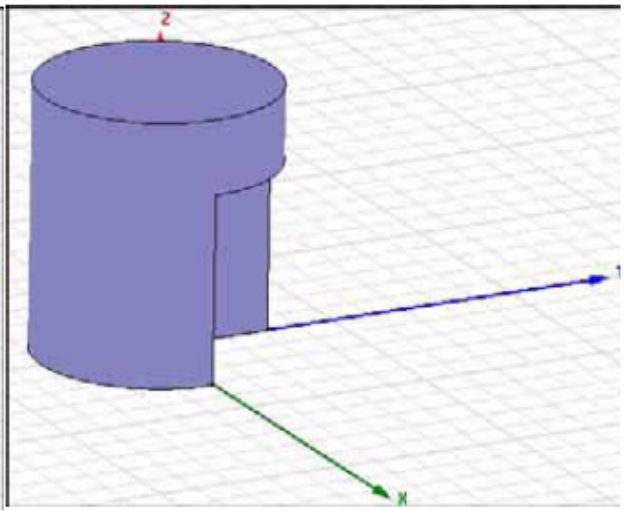
   Though you can type the name of objects into either list, you cannot access the *Subtract* command unless you first select two or more model objects. Therefore, the lists will initially be populated with the selected objects.

5. Optionally, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to keep a copy of the original objects being subtracted.
6. Click **OK**.

   The revised *Blank Parts* retain their original name, color, and material.



An intersecting box and cylinder



A box subtracted from a cylinder (the cylinder was selected first)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

By default, the *Tool Parts* (that is, the objects being subtracted from the *Blank Parts*) are *not* preserved for later use. If you want to keep a copy of the objects being subtracted from the blank parts, do one of the following:

- In the *Subtract* dialog box, select **Clone tool objects before subtracting**, as stated in step 5 above. This action is a one-time override of the defdefault behavior.
- In the *3D Modeler*> *Operation* section of the *General Options*, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to always (that is, by default) keep a copy of the original objects being subtracted.

Creating Objects from Intersections

To create a new object from the intersection of two or more objects, use the **Modeler** > **Boolean** > **Intersect** command.

To create an object from an intersection:

1. Select the objects from which you want to take the intersection.

   Warning:

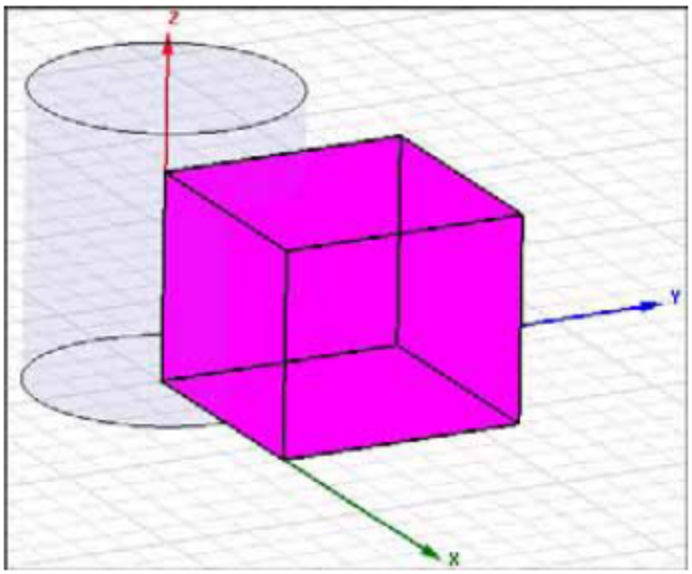
   If the objects you selected do not overlap, the result is a null object, and both objects vanish. When three or more objects have been selected, an intersection is created only if a common volume exists in which all the selected objects overlap. Otherwise, a null object is created and all original objects vanish.

   For example, assume that Part 1 overlaps Part 2, and Part 2 overlaps Part 3, but all three parts do not overlap in a common volume. This situation results in no intersection being created, and all three parts vanishing.

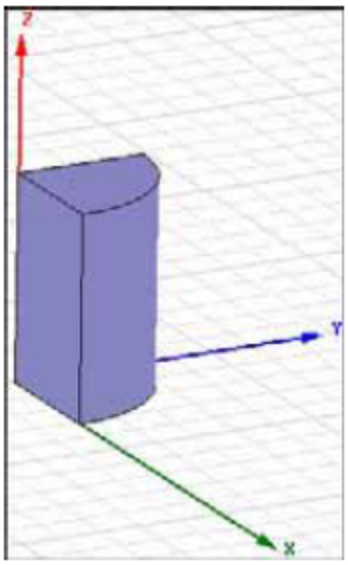
2. From the menu bar, click **Modeler** > **Boolean** > **Intersect**. (You can also access this command from the **Draw** ribbon tab.)

   The original objects vanish, leaving only the new object that was formed from their intersection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An intersecting box and cylinder.



Object formed from the intersection of the box and cylinder.

Note:

By default, the original intersecting objects are *not* preserved for later use. If you want to keep a copy of the objects that intersect the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after creating the new object from the intersection.
- Select **Clone tool objects before intersecting** in the *3D Modeler> Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original tool parts (the second and subsequent objects selected) that intersect the blank part (the first object selected). The blank part is not preserved. To preserve the blank part, do as instructed in the preceding bullet.

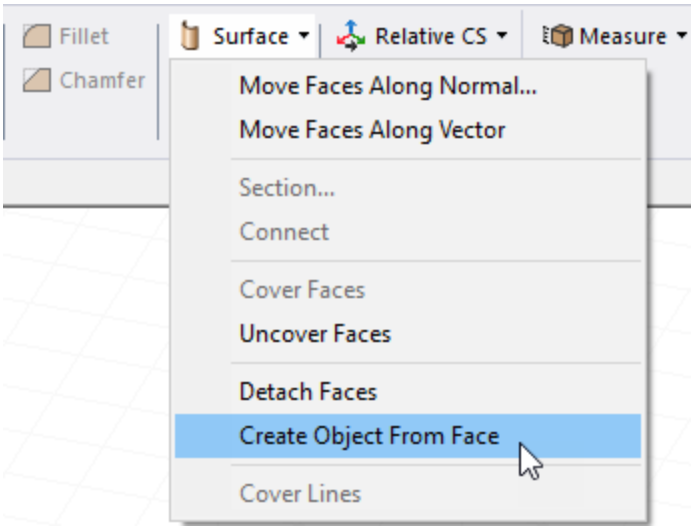
Creating an Object from a Face

The **Modeler> Surface> Create Object from Face** command copies a selected face, resulting in a new 2D sheet object.

To create a new object from a face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.
2. Select the object face you want to copy. If you select multiple faces, each becomes a new object.
3. From the menu bar, click **Modeler**>**Surface>Create Object From Face** or, on the **Draw** ribbon tab, click **Surface**> **Create Object From Face**:



The face is copied, resulting in a new 2D sheet object in the same location.

> **Tip:**
>
> This command is useful for assigning a boundary to the partial intersection of two faces. To do this, first select the faces and create objects from them using the procedure shown above. Next, make sure the **Clone tool objects before intersecting** option is cleared in the *3D Modeler> Operation* section of the *General Options*. Then, select the two sheet objects and use the **Modeler> Boolean> Intersect** command to create a sheet object that includes only the intersection of the two faces. Finally, assign the boundary to the new sheet object.

Creating an Object from an Edge

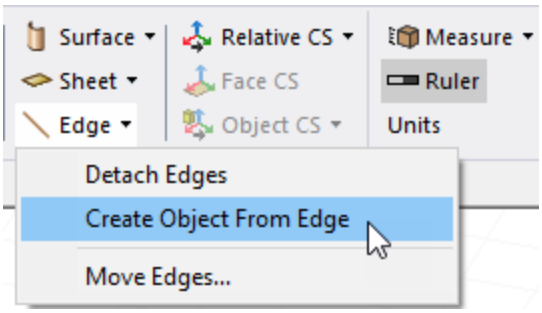
The **Modeler> Edge> Create Object From Edge** command copies a selected edge, resulting in a new line object.

To create a new object from an edge:

1. Right-click in the modeler window and choose **Select Edges** from the shortcut menu.
2. Select the object edge you want to copy. If you select multiple edges, each edge becomes a new line object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler**> **Edge**> **Create Object From Edge** or, on the **Draw** ribbon tab, click **Edge**> **Create Object From Face**:

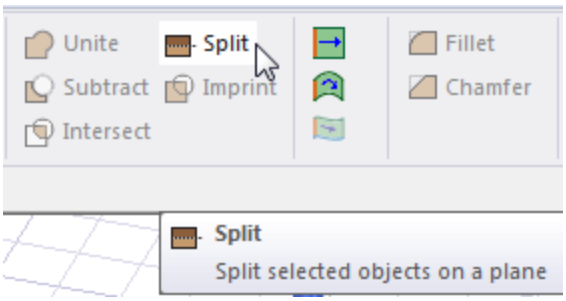


The edge is copied. The resulting object appears in the History Tree as a line object.

Splitting Objects

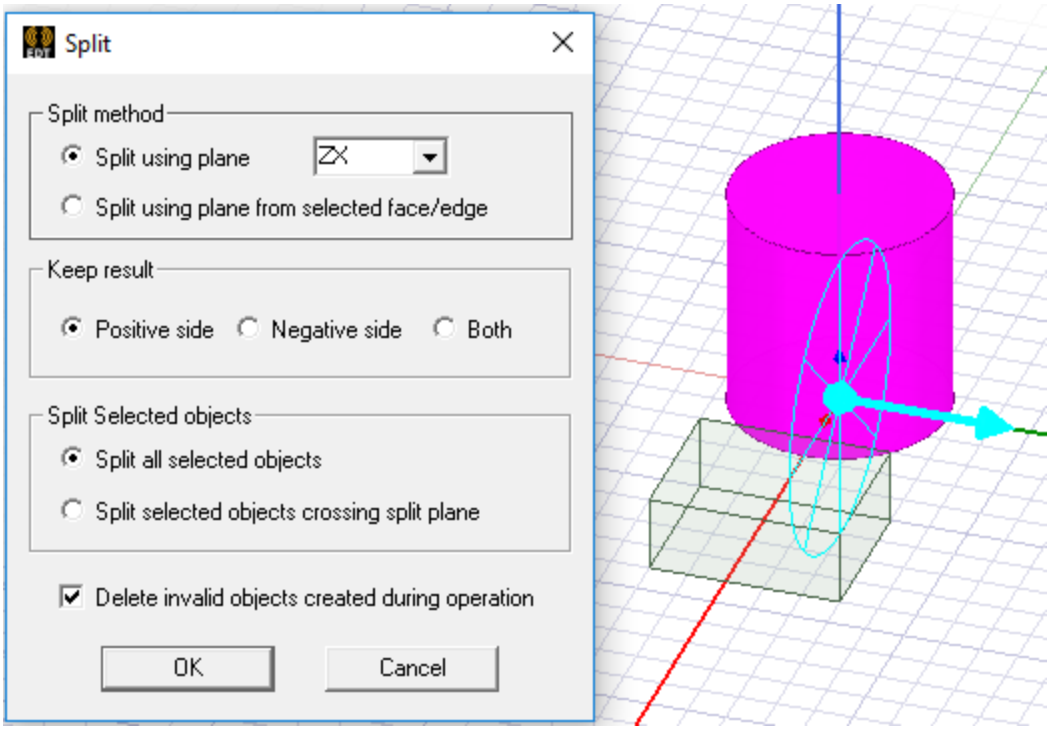
To split an object or several objects using one of the global planes (XY, YZ, or XZ) or a plane based on a selected face or arc, use the **Modeler> Boolean> Split** command.

1. Select the object you want to split. You can select more than one object.
2. From the menu bar, click **Modeler> Boolean>** **Split** or, on the **Draw** ribbon tab, click the **Split** icon:



The *Split* dialog box appears. Also, a cyan colored wheel shows the current split plane and points toward the positive side of the plane.

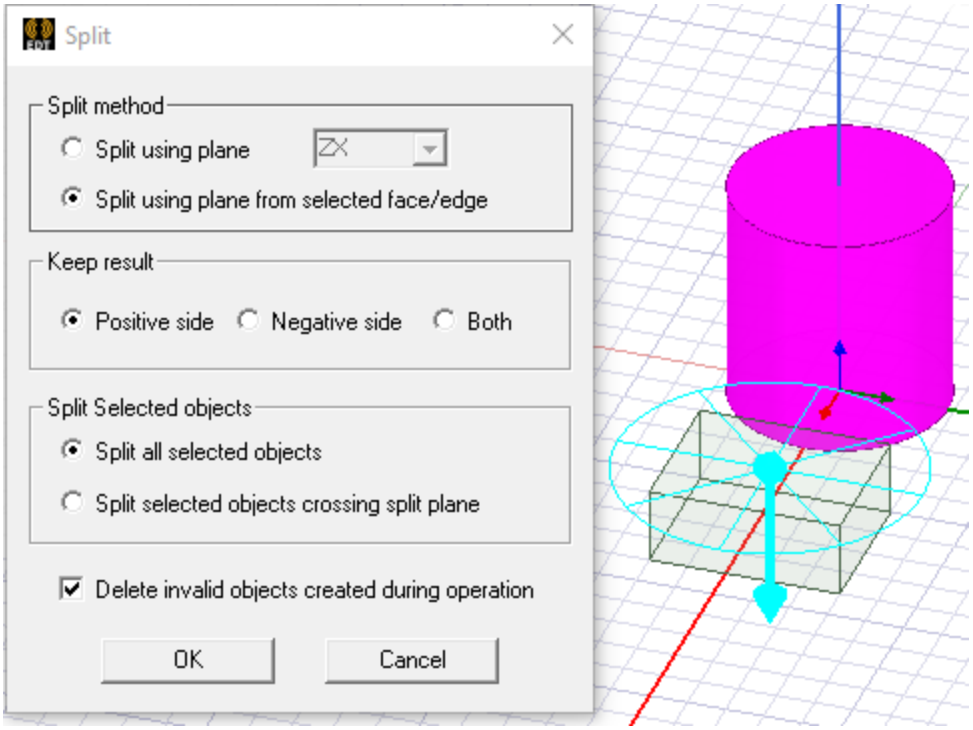
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Choose the desired *Split method* and specify the splitting plane:
   - **Split using plane:** Select the desired splitting plane (**XY, YZ,** or **XZ**) from the drop-down menu, or
   - **Split using plane from selected face/edge:** Select a single face or edge (specifically, a planar face or arc) to define the splitting plane.

     The following example shows the top face of the box being used to define the splitting plane for the cylinder:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

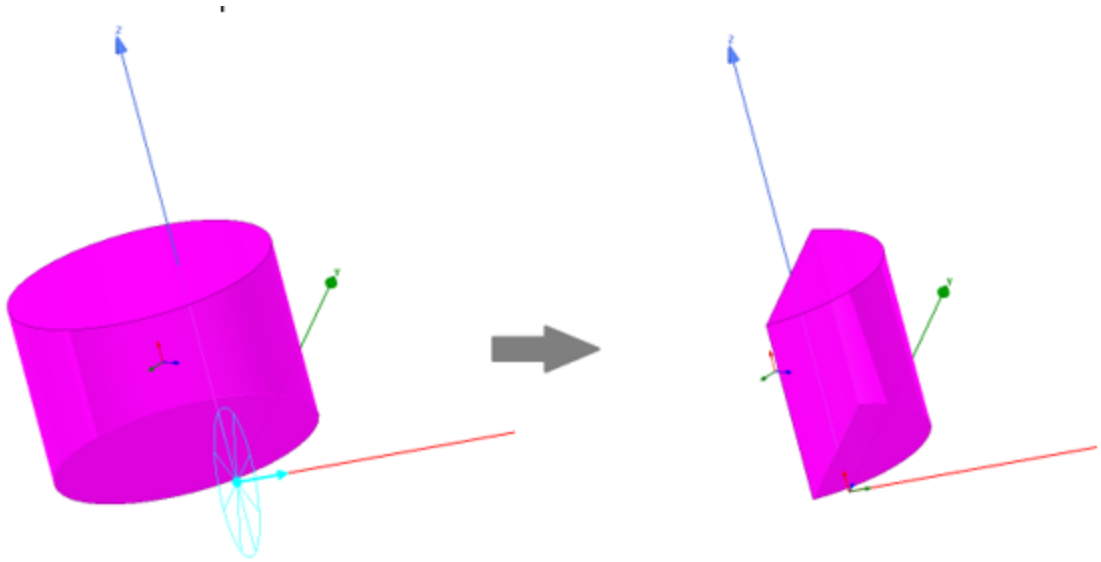


4. Select one of the following *Keep result* options to specify which object fragments you want to keep:
   - **Positive side:** Keep objects on the positive side of the splitting plane.
   - **Negative side:** Keep objects on the negative side of the splitting plane.
   - **Both:** Keep objects on both sides of the splitting plane.

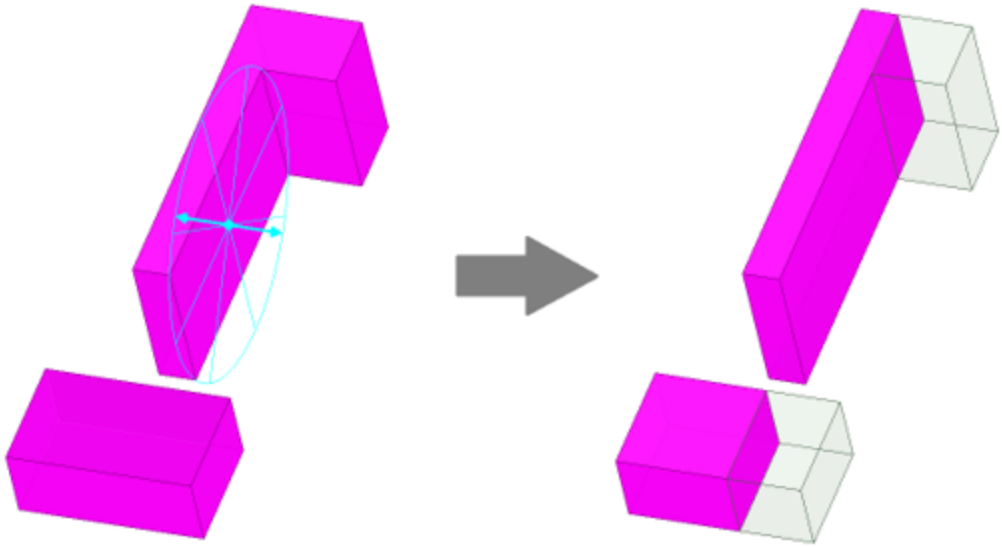
   The split plane is shown using a cyan colored wheel with spokes, as seen in the images above and below left. A cyan arrow indicates which side of the split plane is the positive side.

   In the following example, the *YZ* plane (green and blue axes) is the splitting plane, and the result on the *positive side* was kept. The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next example shows the second *Split method* (*Split using plane from selected face/edge*). You can only select one entity, a planar face or an arc (selected as an edge). The face or edge used to define the splitting plane need not belong to one of objects selected to be split. You can select any face or arc edge in the model.. You can pan, zoom, or rotate the model viewpoint as needed to make the selection easier. Also, the *Keep result* for *Both* sides option was selected. (Notice the arrow is in both directions for the split plane graphics.) The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Select one of the following *Split Selected objects* options:
   - **Split all selected objects:** Select this option if you do not want to preserve objects that are not crossing the split plane but are part of the selection.
   - **Split all selected objects crossing split plane:** This option allows you to identify selected objects that do not cross the split plane and ignore them for the operation. For a multiple selection, only those objects that cross the split plane are split; others are kept intact. By design, splits in existing designs from previous software versions are not changed.
6. Optionally, select the **Delete invalid objects created during operation**. If you choose to *Split all selected objects*, the operation could create invalid objects (if split plane does not cut any of selected objects and any are on the side not being kept). This option deletes any such invalid objects that might be created.
7. Once the proper selections have been made, click **OK**.

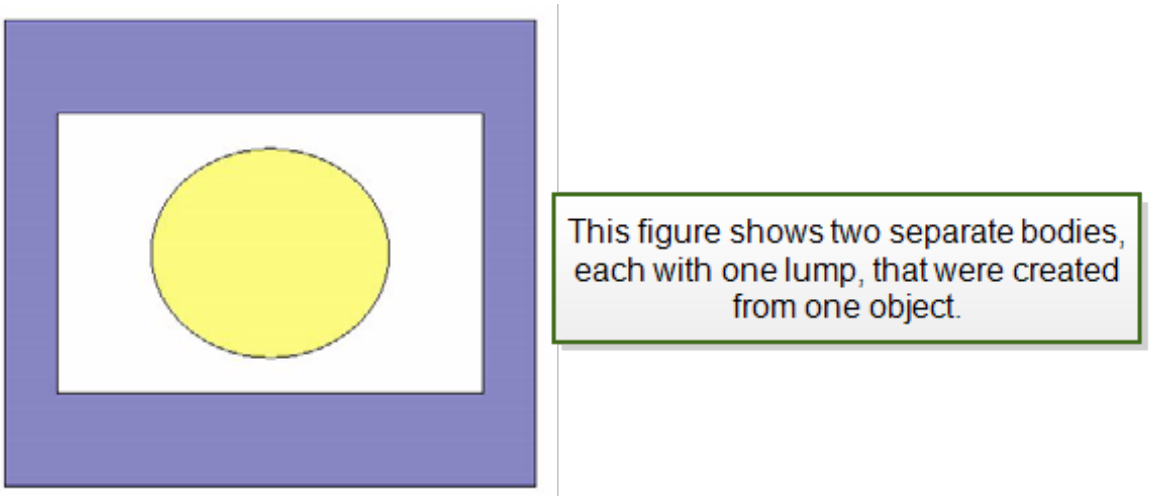
   The objects are divided and retained as specified.

Separating Bodies

To separate an object with multiple lumps (geometric bodies) into individual single-body objects:

1. Select the object you want to separate.
2. From the menu bar, click **Modeler**> **Boolean**> **Separate Bodies**.

   The bodies are separated into multiple objects.



Converting Polyline Segments

A polyline is a single object that includes any combination of straight line, arc, or spline segments. You can convert a polyline segment from one type to another. The following

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conversions are supported:

- Straight line segments to arc or spline segments.
- Arc segments to straight line or spline segments.
- Spline segments to straight line segments.

To convert polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to convert. Expand this part of the tree.
2. Click the polyline segment operation you want to change to select it.

   The segment properties appear in the *Segment* tab of the docked *Properties* window.
3. Click in the **Value** text box in the **Segment Type** row and select the desired polyline segment type from the drop-down menu.

   The polyline segment you selected is changed to the new type.

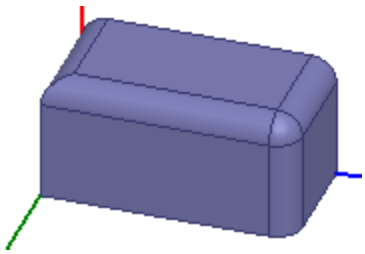
   > **Note:**
   >
   > Converting an arc or spline segment to a straight line segment results in two straight line segments; one segment is created between the start point and midpoint and one segment is created between the midpoint and endpoint.
4. By default, curved segments are generated as smooth (true) curves. If you want to approximate an arc or spline using multiple discrete straight segments, enter an integer of 2 or greater in the **Number of Segments** value. A value of zero produces a smooth curve.

   Segmented curves can be useful when sweeping polylines into surfaces. Segmented surfaces provide some regularity to the mesh and can enable you to visualize surface twisting when you sweep along a path with a twist angle specified.
5. If the changes are not what you expected, undo the change using the **Edit> Undo** command or by pressing **Ctrl+Z**.

Rounding the Edge of Objects (Fillet Command)

The fillet command rounds the object at the original edges and vertices. The edges and vertices are replaced by new rounded surfaces that are tangential to the original adjacent surfaces, providing a smooth transition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Vertices are only going to be replaced by new rounded surfaces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary). The edges are replaced by quarter-cylindrical surfaces, of which the radius can be customized (see the **Fillet Radius** property). Vertices are replaced by more complicated new faces. You can control the setback distance.

The fillet command is disabled if an edge is not selected.

To switch to edge selection mode:

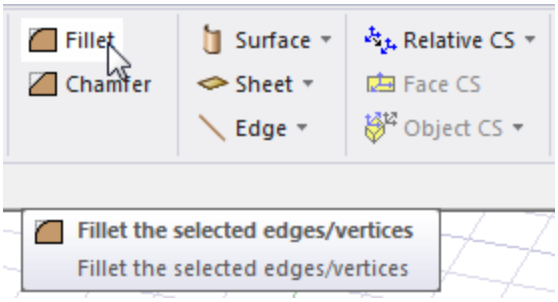
- Right-click the desktop, and choose **Select Edges** from the shortcut menu.

To round the edge of an object:

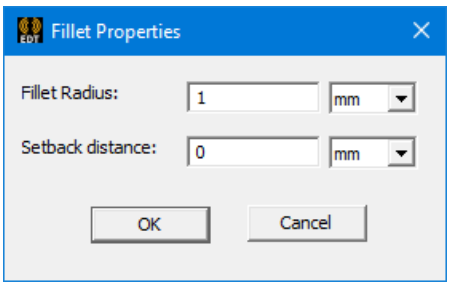
1. Select the edge you want to change.

   The edge is highlighted and the *Fillet* command is enabled.

2. From the menu bar, click **Modeler** > **Fillet** or, on the **Draw** ribbon tab, click the **Fillet** icon:



   The *Fillet Properties* dialog box is displayed:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a value for the **Fillet Radius** and select units from the drop-down menu.
4. Enter a value for the **Setback distance**.

   The setback distance controls the shape of the vertex. It is the distance of the cross curve from the vertex at the end of the edge. If it is less than the fillet radius the *Setback distance* has no effect. Also, you will get an error if it is greater then the length of the edge. Note that the setback feature only works on corners where three or more edges meet and only if all edges meeting at the vertex are selected.

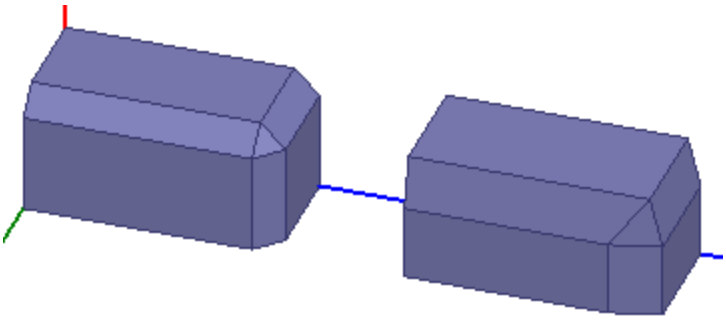
5. Click **OK** to apply the change to the edge.

   The dialog box closes and the object edges and vertices are rounded as specified.

Flattening the Edge of Objects (Chamfer Command)

The chamfer command flattens the selected edges and vertices of the object. Selected edges and vertices are replaced by new flat surfaces known as chamfers.

Chamfers can be symmetric, or asymmetric:



The angle between the original faces and the chamfer's face varies depending on the angle between the original faces, and it also depends on the left and right chamfer distances. For example, for two faces at 90° with an equal distance (symmetrical) chamfer, the edge chamfer's face will be 45° from both of the original faces.

Vertices are only going to be replaced by new flat faces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary).

The chamfer command is disabled if an edge is not selected.

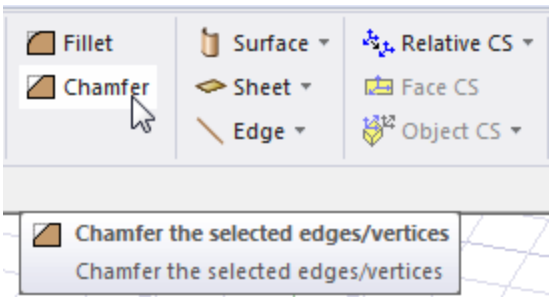
To flatten an object's edge for a symmetric chamfer:

1. Right-click the desktop, and select **Select Edges** from the shortcut menu.
2. Select the edge (or edges) you want to change.

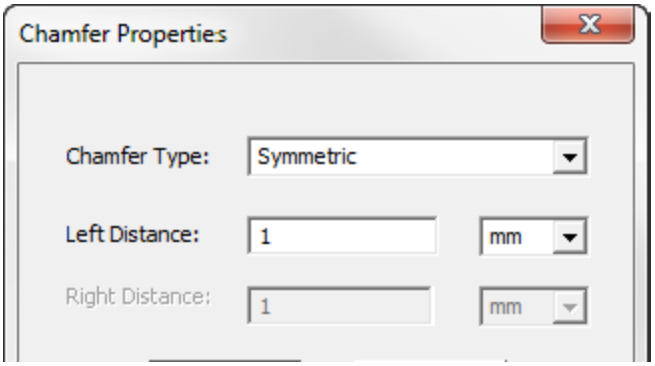
   The edge(s) is highlighted, and the **Chamfer** command is enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler> Chamfer** or, on the **Draw** ribbon tab, click the **Chamfer** icon:



The *Chamfer Properties* dialog box appears.



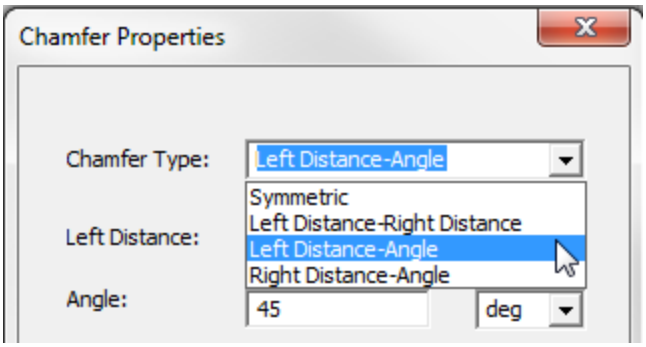
4. Type a **Left Distance** value and select the units from the drop-down menu.
5. Click **OK** to apply the change to the selected edges.

To flatten object's edge for an asymmetric chamfer:

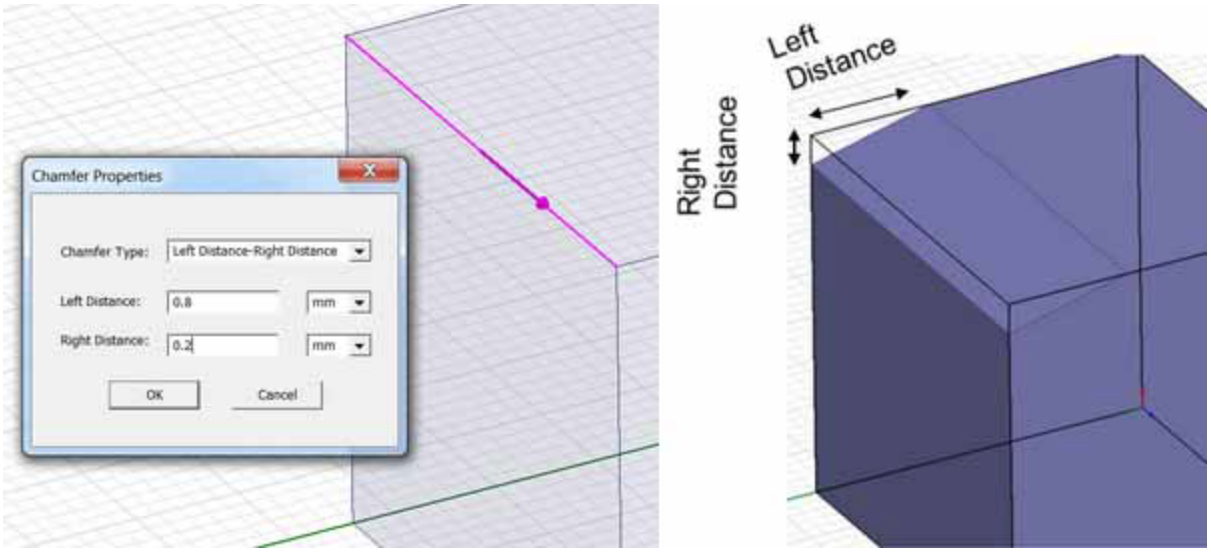
1. Select one or more edges and click **Modeler> Chamfer** to open the *Chamfer Properties* dialog box.
2. Use the **Chamfer type** drop-down menu to select the type:

   You control an asymmetric chamfer by selecting a type that defines the chamfer asymmetry as **Left Distance-Right Distance**, as **Left Distance-Angle**, or as **Right Distance-Angle**. Notice that red directional arrows on the selected edges provide the direction against which left distance and right distance are determined.

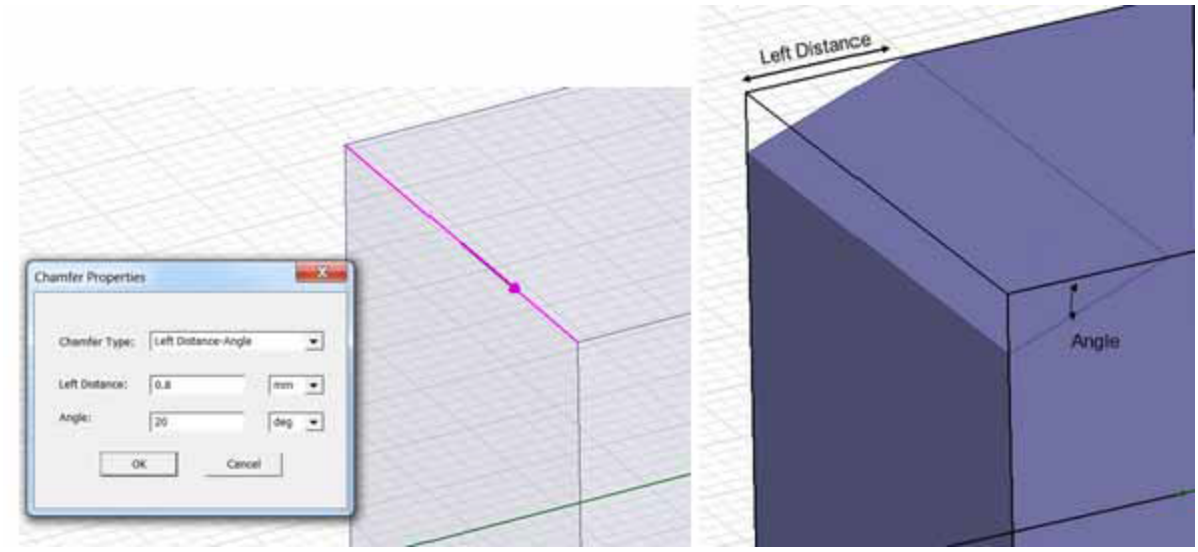
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example 1: Left Distance-Right Distance

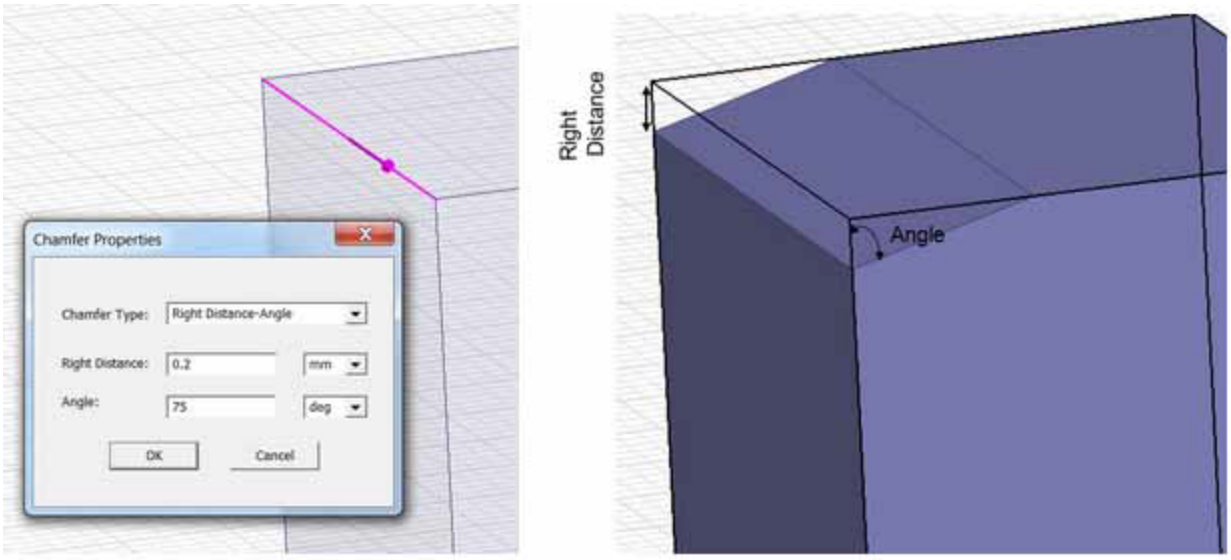


Example 2: Left Distance-Angle



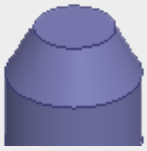
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 3: Right Distance-Angle



Note:

Only symmetric and left distance-right distance chamfers are supported for edges formed by a curved surface.



Angle-distance chamfers are not supported in such cases and do not appear on the Chamfer Type drop-down menu.

3. Click **OK** to apply the chamfer to the selected edges.

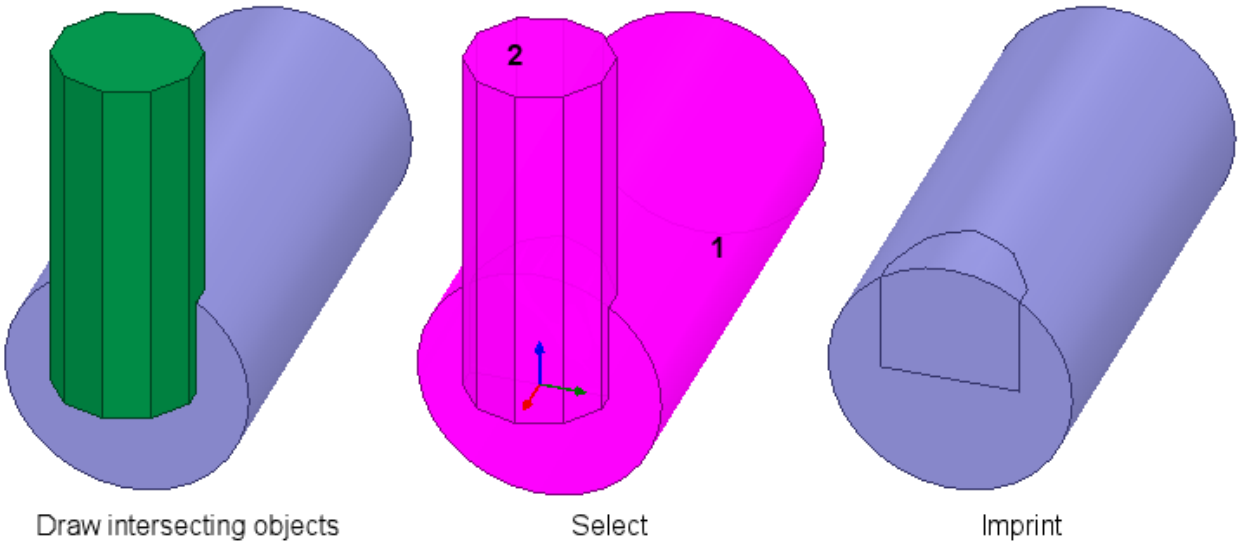
Imprinting an Object

The **Boolean** > **Imprint** command lets you imprint the geometry of one object upon another. For example, you could draw a polyhedron intersecting a cylinder, and then imprint the intersecting lines on the cylinder. This process splits the larger face into two or more faces, with the intersecting lines representing the edges of the imprinted face. The original face is cut so that it no longer extends into the imprinted area.

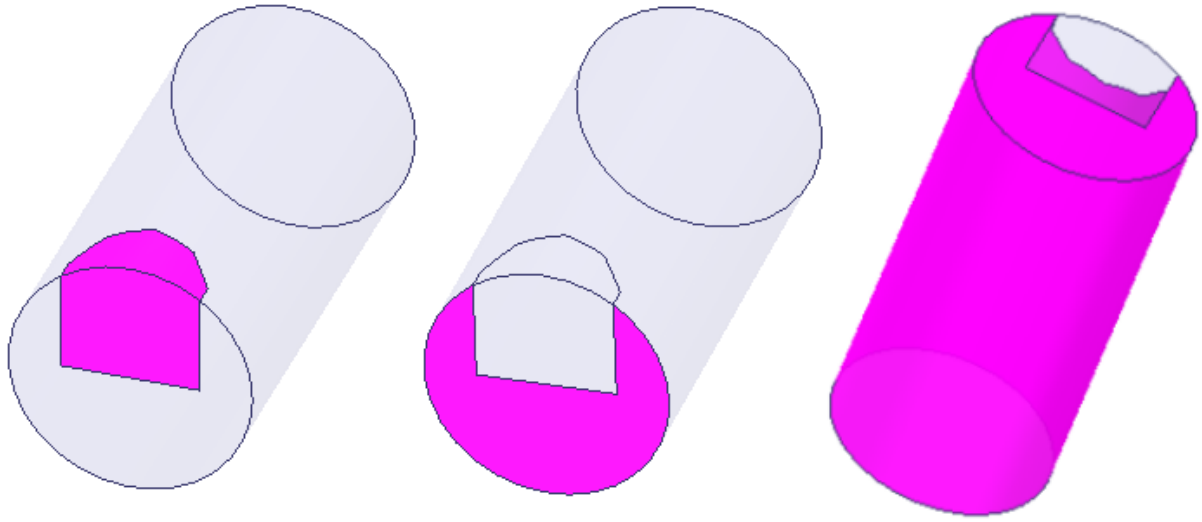
Imprinting is often an essential capability for finite element analysis. It enables a boundary or excitation to be applied to the exposed area of a model face while excluding it from areas of intersection with other parts. One example is a thermal analysis where convection is applied to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the exposed surfaces of a heat sink. The area in contact with a semiconductor (the source or heat) is not exposed to ambient air and should not receive a convection boundary. In this case, you would imprint the semiconductor on the heat sink, producing a separate heat sink face in the contact area. This face is excluded from the selection set when applying the convection boundary.



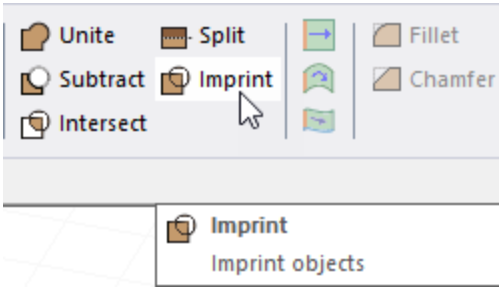
You can select the faces of the imprinted surface separately and assign properties as needed.



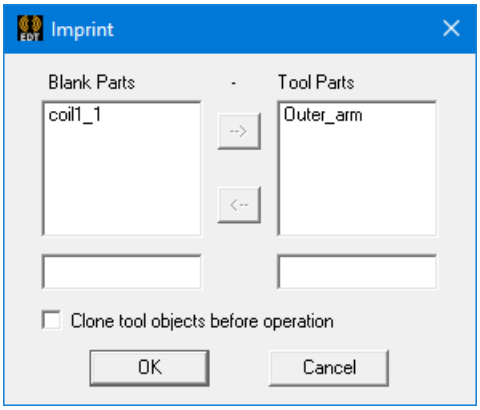
To imprint one object with another:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the intersecting objects.
2. From the menu bar, click **Modeler** > **Boolean** > **Imprint...** or, on the **Draw** ribbon tab, click the **Imprint** icon:

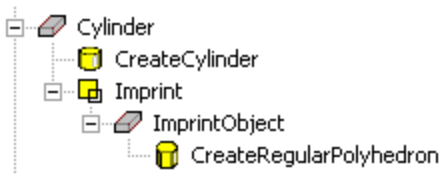


The **Imprint** dialog box displays in which you designate which objects are the *Blank Parts*, and which the *Tool Parts*. If necessary, you can select the objects in lists, and use the arrow keys to move them. If desired, you can clone the tool objects before the imprint operation.



3. Optionally, select **Clone the tool objects before operation** to retain the tool part. Otherwise, only its imprint on the blank part will remain.
4. Click **OK** to close the dialog box and perform the Boolean imprinting.

   The History Tree retains the ImprintObject command and the Create<object> command for the imprinted tool object:



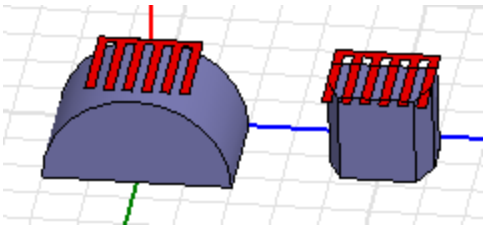
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select the Imprint command in the History tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon, you can edit the properties of that object. The changes you apply carry over to the imprinting operation, modifying the imprinted face accordingly.

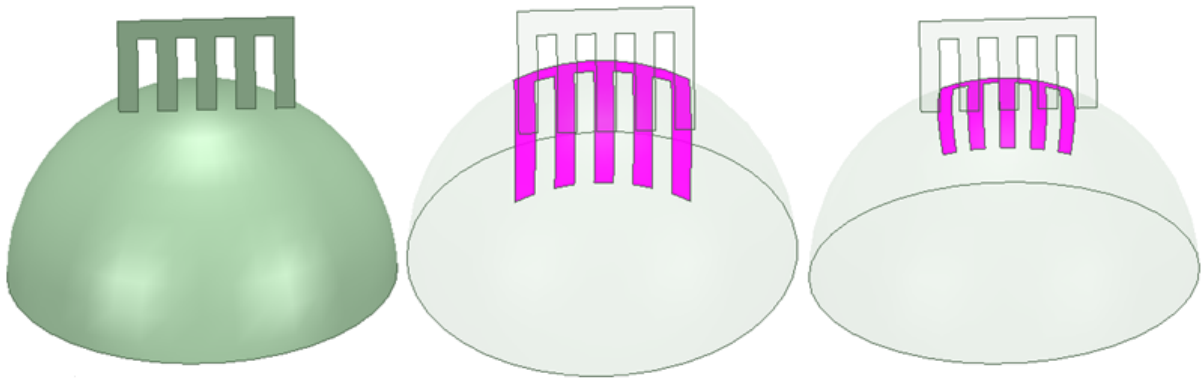
In the *3D Modeler* > *Operation* section of the *General Options*, you can also enable the **Automatically imprint wrapped sheets** option to automatically perform an *Imprint* operation command after executing the **Modeler** > **Surface** > **Wrap** command.

Imprint Projection Commands

The **Boolean** > **Imprint Projection** commands lets you project the form of a sheet object onto the face or faces of another object (either solid or sheet). The receiving object faces can be flat, curved, or faceted.



If the receiving surface is curved or inclined, and you choose to project *Along Normal* direction, the dimensions of the projection are affected. The reason is that the projection direction is normal to the curved or inclined face of the receiving object and not normal to the object being projected.



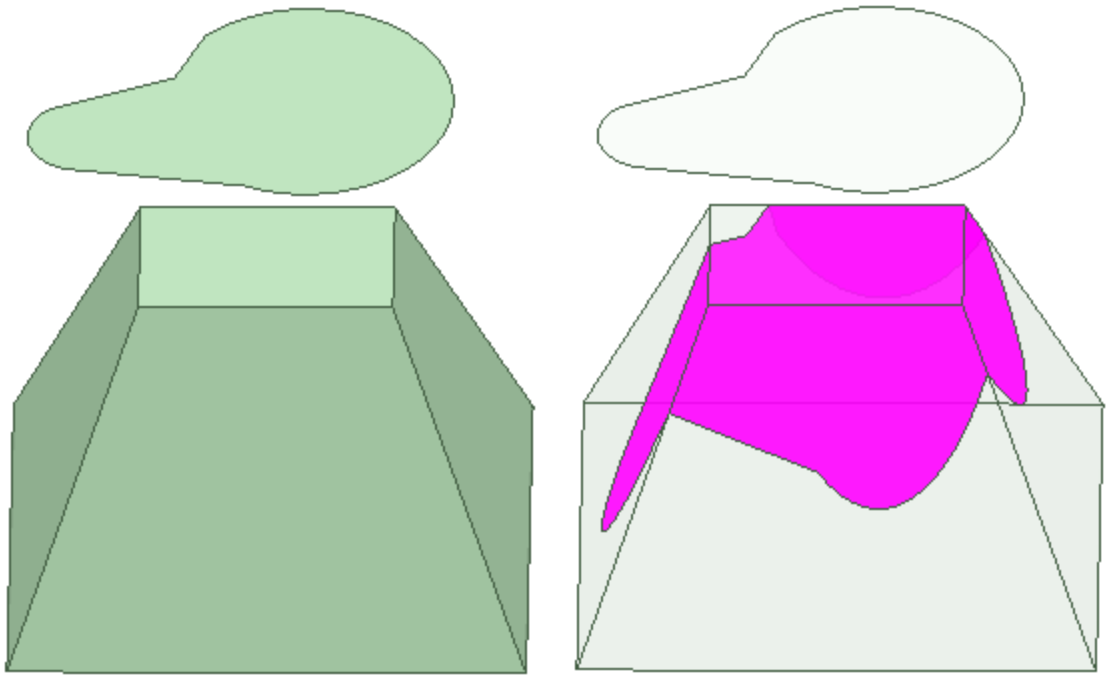
Comb & Hemisphere

Comb Projected Along Vector

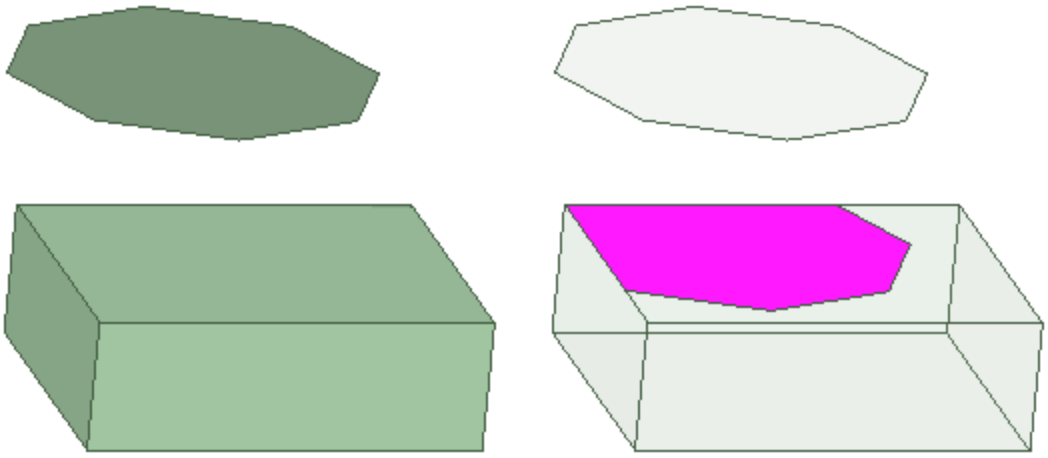
Comb Projected Along Normal (normal to hemispherical target object)

The projected shape will wrap around the target object, from the face nearest to the center of the imprinting area and extending to the adjacent target object faces:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the projection of the imprinting object extends beyond the extents of the receiving object, the imprinted shape is truncated:

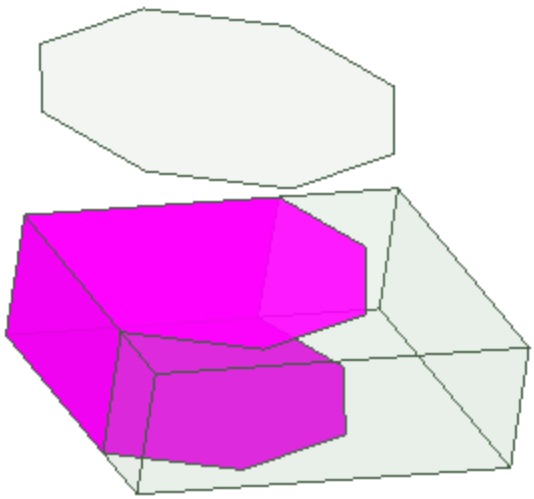


You can select the imprinted faces on the target object separately, and perform subsequent operations (such as creating sheet objects from the faces, assigning boundaries or excitations, or selecting for field overlays).

To perform an Imprint Projection:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

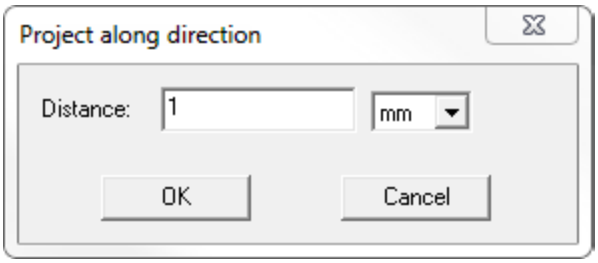
1. If you want to preserve the tool object that will be imprinted on the target object, use one of the following methods:
   - Copy the tool object to the clipboard so that you can paste it back into the model after the imprinting operation.
   - In the *3D Modeler > Operation* section of the *General Options*, select the **Clone tool objects before projecting** option.
2. Select the projecting and target *objects*.
3. From the menu bar, click **Modeler** > **Boolean** > **Imprint Projection** > **Along Normal** or **Modeler** > **Boolean** > **Imprint Projection** > **Along Direction...**
   - If you select **Along Normal**, the projection occurs along the direction normal to the target faces and the operation is completed immediately. Imprinting occurs only on target faces visible from the imprinting object's point of view.
   - If you select **Along Direction**, you must specify two points that define the imprinting vector direction (step 3 below). Once you have done this, you see a dialog box for specifying the projection **Distance**. This option allows you to limit the projection distance so that only near faces are affected. Unlike the *Along Normal* option, the *Along Direction* option can imprint onto the target object faces that are not visible from the imprinting object's point of view (that is, those on the opposite side of the target object):



4. If performing an imprint projection operation using the **Along Direction** option:
   a. Specify the imprinting vector direction. You can click in the Modeler window or use the coordinate entry text boxes in the Status Bar.

      The *Project along direction* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Specify the maximum projection **Distance** and select the length unit from the drop-down menu.
c. Click **OK**.

   The dialog box closes and the Boolean imprinting operation is performed.

After you perform the imprinting, the History Tree shows the ImprintProjection command and the create command for the imprinted object.

If you select the *Imprint Projection* command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting operation.

Purging Object History

Each object is a sequence of modeler-based operations. The history for each object is shown under its name in the History Tree. You can use the **Purge History** command to remove the history of operations while not affecting the geometry itself. This is useful when you wish to perform healing operations on the object. If there is an object for which you want to keep the history, you should make a copy of the object for that purpose before purging.

To purge the history:

1. Select the object.
2. From the menu bar, select **Modeler**> **Purge History**. The history for the model is purged, and the context for the **Undo** and **Redo** commands is updated.

Related Topics

Working with the History Tree

Generate History

Generate History to Reproduce Portions of Model

If a primitive object (cylinder, cone, box, circle, 3D polygon, etc.), polyline object (line, spline, or arc), circle, or ellipse is imported, or if the history was previously purged, you can select the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

object and use the **Generate History** command to reproduce, in the History Tree, the primitive object definition or the individual line segments used to create a polyline. *Generated History* uses the original coordinate system in which the objects were created. *Generate History* also retains boundary conditions.

To reproduce the line segments in the model history tree:

1. Select the primitive or polyline object.
2. From the menu bar, click **Modeler> Generate History**.

Selecting Items in the 3D Modeler Window

To modify or learn more about an item's properties, you must first select it. All commands you choose while an item is selected are performed on or in reference to the selected item.

What selection mode do you want to use?

- Select Submodels
- Select Objects
- Select Faces
- Select Edges
- Select Vertices
- Select Multi
- Coordinates in the drawing space
- Select By Area including optional filters
- Select by History Tree Group
- Select by Submodel
- Select by intersection error message
- Selecting the Face or Object Behind

Selection menu for:

The **Edit>Select Objects** menu and **Modeler** window shortcut menu includes selection for:

- By Name
- By Material
- By Variable
- All Model Objects
- All Non-model Objects
- All Primitive Shapes
- All Non Primitive Shapes
- All Solids

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All Sheets
- All Lines

Selections enabled for features of selected Object:

Some **Edit>Extend Selection** menu commands are enabled when you select an appropriate object:

- All Object Faces
- All Object Edges
- All Face Edges
- Faces on Plane
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain
- Select Face Chain
- Select Sheet Edges

History Tree Icon Menu Selection

You can also right-click **History** tree icons to **Select All** for ungrouped:

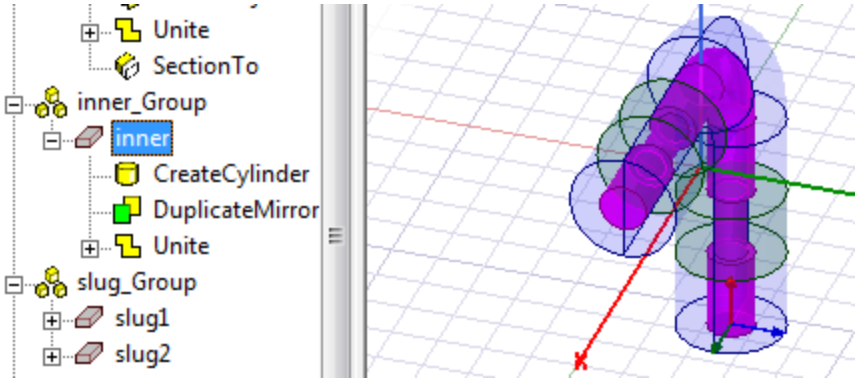
- Model
- Group
- Solids
- Selected Material, for example copper or air.
- Sheets
- Lines

> **Note:**
>
> If selected objects do not display correctly, for some graphics card, you can improve performance by changing settings in the *NVIDIA Control Panel* application. In the *Select a Task* tree, navigate to **3D Settings> Manage 3D Settings**. Then, under the **Global Settings** tab, choose **Workstation App - Dynamic Streaming** from the **Global presets** drop-down menu.
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

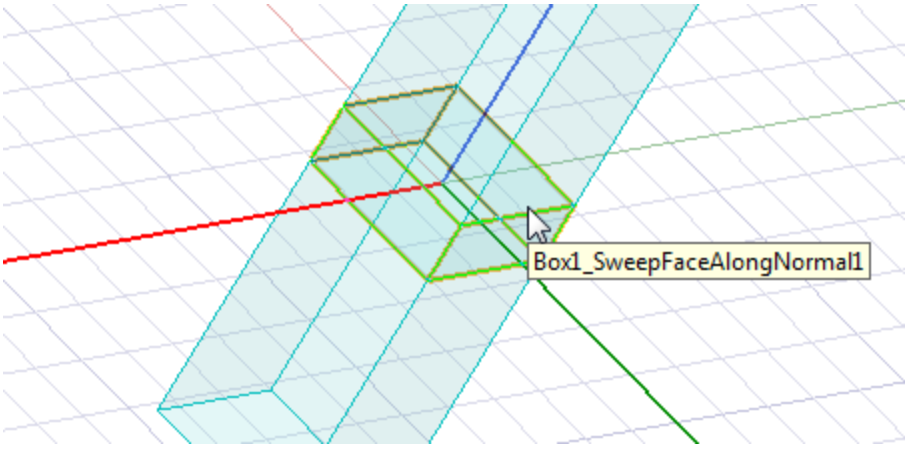
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects

By default, the modeler is in *object selection mode*. Simply click an object in the view window or an object name in the history tree and it will be selected. All other objects become relatively transparent. Selected objects become the color specified under the **Display** tab of the **Modeler Options** dialog box.



When the mouse hovers over an object in the view window, that object is highlighted, which indicates that it will be selected when you click. As you hover the cursor over an entity, tooltips indicate its type/ID (object name in the case of objects, Face_id in the case of faces, and so on). This feature helps you distinguish between face-of-sheet-object pick versus sheet-object pick.

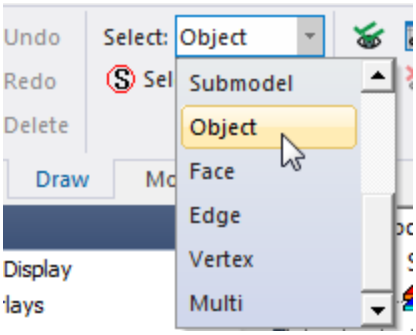


If the modeler is not currently in object selection mode, you can switch to it using one of the following methods:

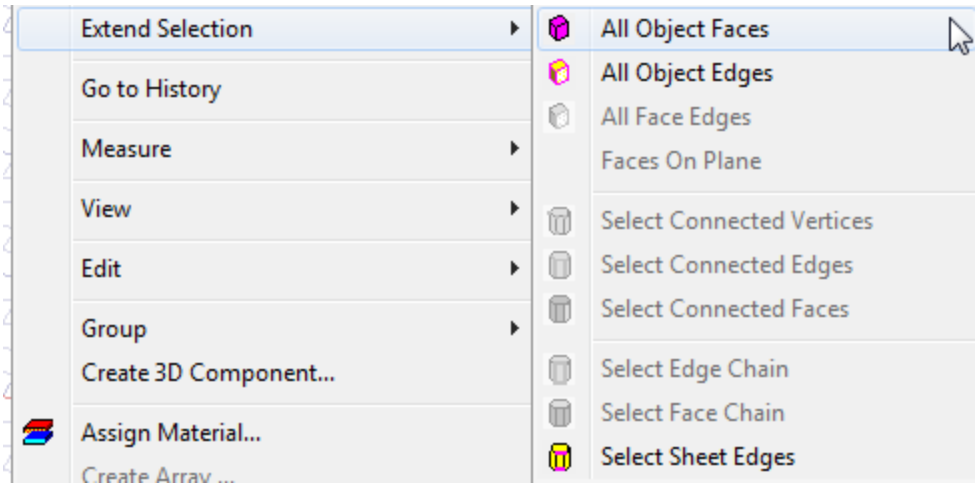
- Press the **O** shortcut key while the Modeler window is active.
- Right-click in the Modeler window and click **Selection Mode**> **Select Objects**.
- From the menu bar, click **Edit**> **Selection Mode**> **Objects**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** or **Model** ribbon tabs, choose **Object** from the **Select** drop-down menu:



When you've selected an object, the menu bar's **Edit**> **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:

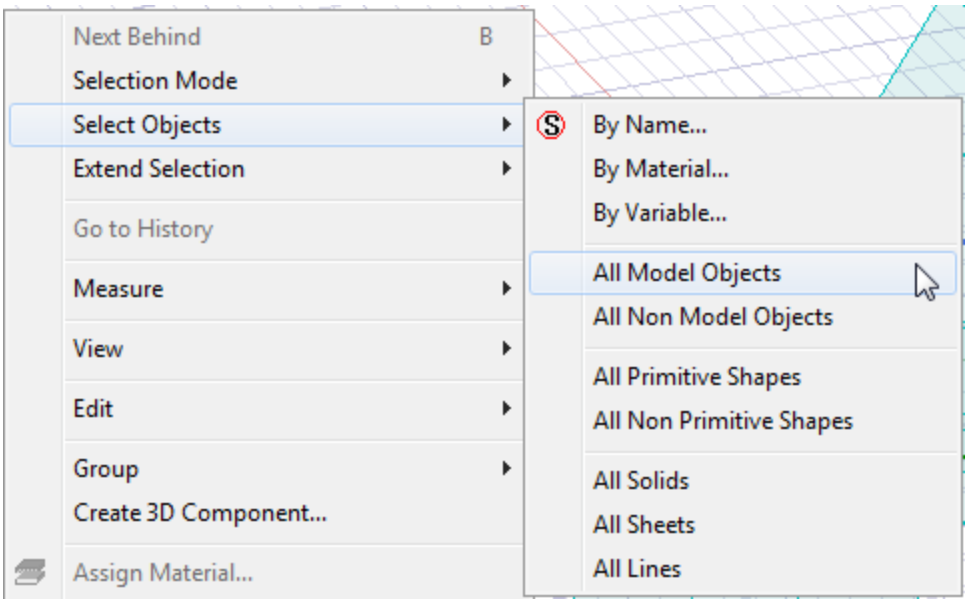


Selecting Several Objects

1. If you are selecting objects in the Modeler window make sure that the modeler is in object selection mode by pressing the **O** shortcut key. You can always select objects in the History Tree.
2. Select several objects in one of the following ways:
   - Hold down **Ctrl** and click the objects in the Modeler window that you want to select.
   - Hold down **Ctrl** and click the object names in the History Tree that you want to select.
   - In the History Tree, select a range of objects by first clicking one object to select it, and then Shift+click to extend the selection of visible items.
   - In the History Tree, under *Lists*, select **AllObjects**, which is an automatically created list of all model objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the menu bar, click **Edit> Select All** to select all objects that were drawn in the active Modeler window, including objects that are not currently visible.
- Press **Ctrl+A** or, from the menu bar, click **Edit> Select All Visible** to select all visible objects in the active Modeler window.
- From the menu bar, click **Edit> Select Objects>** or right-click in the Model window and go to the **Select Objects>** submenu. Then, choose a command from the submenu suitable for the objects you want to select:



- Right-click a History Tree icon for Model, Group, object, category (such as Solid or Sheet), or material to perform context-sensitive selections. See Select Objects in the History Tree.

To deselect all objects, do one of the following:

- From the menu bar, click **Edit> Deselect All**.
- Press **Ctrl+Shift+A**.

Selecting Objects by Name or Material

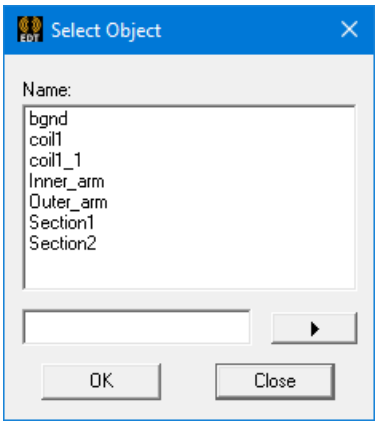
1. Make sure that the modeler is in the *Object* selection mode by pressing the **O** shortcut key or by choosing **Object** from the **Select** drop-down menu on the **Draw** or **Model** ribbon tab. Then, proceed to one of the following two procedures.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selection by Object Name

2. Access the *Select Object* dialog box using one of the following three methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Name**.
   - On the **Draw** or **Model** ribbon tab, click **Select by Name**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Name** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the objects in the model, by name:



3. In the *Name* list, click the name of the object you want to select. Use the **Ctrl** key to select more than one.

   Alternatively, type the name of an object in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

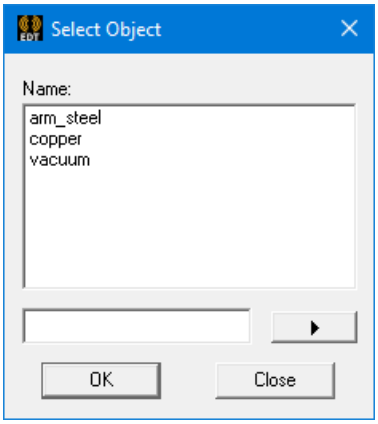
   The object or objects specified are selected.

Selection by Object Material

2. Access the *Select Object* dialog box using one of the following two methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Material**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Material** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the materials comprising the model objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the *Name* list, click the material of the object or objects you want to select. Use the **Ctrl** key to select more than one material.

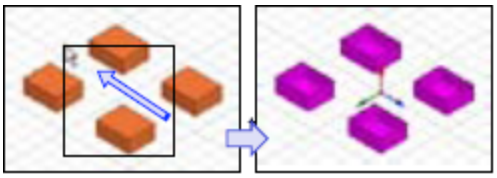
   Alternatively, type the name of a material in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

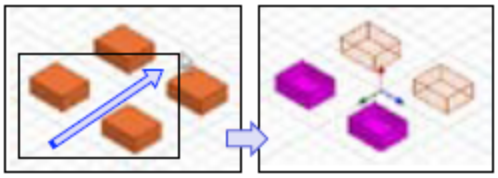
Select By Area

You can select items by area in the Modeler window by clicking and dragging a rubber band (selection box) around objects. Rubber band selection works differently depending on the direction you drag the mouse:

- **From right-to-left:** Selects all items that are wholly or partly enclosed within the rubber band.



- **From left-to-right:** Selects only the items that are wholly enclosed within the rubber band.

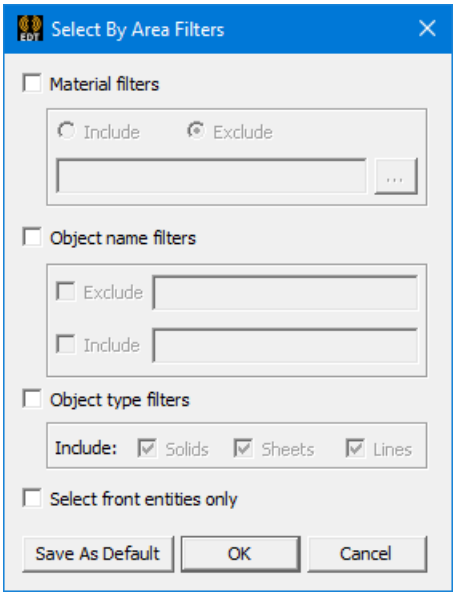


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Area selection works with Object, Face, Edge, and Vertex selection modes, but not for the Multi selection mode.

You can control which objects to include or exclude from area selections based on material, object name, or object type. To do this:

1. From the menu bar, click **Edit** > **Select By Area Filter** to display the following dialog box:



2. Check **Material filters** to enable the **Include** and **Exclude** radio buttons. Use the text box to specify filters by name, or use the ellipsis [...] button to display the *Select Definition* dialog box, from which you can browse and select materials.
3. Check **Object name filters** to enable the **Exclude** and **Include** check boxes and associated text boxes in which you can specify object names.
4. Check **Object type filters** to enable the *Include* check boxes for including **Solids, Sheets,** and/or **Lines**.
5. Check **Select front entities only** to prevent entities in the background (that is, those hidden behind other entities) from being selected.
6. If you click **Save As Default**, the settings persist for the project (until you change the settings and *Save as Default* again.
7. Click **OK** to close the dialog box.

   Now when you click and drag around an area, those objects which meet the filter criteria are highlighted in the Modeler window, and those objects are shown as selected in the History Tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Default Color and Transparency of Selected Objects

To set the color of objects when they are selected:

1. From the menu bar, click **Tools> Options> General Options** or, on the **Draw** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. In the tree at the left side of the dialog box, select the **3D Modeler> Display> Rendering** group of settings.
3. In the *Rendering Defaults* section, choose **Select** from the **Default color** pull-down menu.
4. Click the color button to the right of *Select*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Any objects you select after this point will temporarily become the default color you selected.

By default, the modeler shows selected objects as nearly opaque and shows non-selected objects as nearly transparent. This features helps you to easily distinguish between selected and non-selected objects and to see selected objects on the back side of the model.

To set the transparency of selected and non-selected objects:

1. From the menu bar, click **View> Options**.

   The *3D UI Options* dialog box appears. The *When there is a selection* region contains check boxes for setting the transparency for selected and non-selected objects.

2. Ensure that the check box is selected for any value you want to change.

   Each check box enables the associated value box. The default transparency for selected objects is 0.1, which makes them almost opaque. The default transparency for non-selected objects is 0.9, which makes them highly transparent.

3. Enter a new value, and click **OK**.

Setting the Default Color of Highlighted Objects

1. From the menu bar, click **Tools> Options> General Options** or, on the **Draw** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select the **3D Modeler> Display> Rendering** group of settings.

   The *Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the *Rendering Defaults* section, choose **Highlight** from the **Default color** pull-down menu.
4. Click the color button to the right of *Highlight*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Outlines of all object you hover over temporarily become the default color you specified above.

Creating an Object List

Create an object list when you want to group multiple objects for convenient selection and processing. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To create an object list:

1. If you are selecting in the Modeler window, make sure that the modeler is in object selection mode by pressing the **O** shortcut key. This is not necessary for selecting in the History Tree.
2. Select the objects you want to include in the list.
3. Click **Modeler> List> Create> Object List**.

   The object list is created with the default name Objectlist*n*. It appears in the History Tree under **Lists**. Selecting an object list displays the properties of that list in the docked *Properties* window. One of the properties is a list of objects contained in the list.

   To rename the Object list, edit its Name in the docked *Properties* window. Object lists are sorted in alphanumeric order.

The object list is treated as one volume when you plot and perform fields calculations. It will be listed in the *Geometry* window of the Fields Calculator when you select **Volume**.

There is an automatically created list called AllObjects. Select this list selects all objects in the model. If a list contains mixed types of geometry, for example, volume and sheet objects, the volume calculation only uses the geometry of the highest dimension in plots or integral, and so forth.

- Line objects in the list do not affect where the fields are saved. If all objects in the list are lines, fields will be saved on all objects.
- To post processing on line object when saving fields on object/face list, include both the line and its containing object/face in the list.

Example: To plot the E-field on a surface formed by the intersection of the XY-plane and several objects, first define a list of these objects. Then, when plotting fields, select the ***object list name***

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the **Geometry** window of the *Fields Calculator*. Fields will be plotted only at the intersection of the plane and the objects in the list.

Reassigning Objects to Another Object List

You can assign objects after you have created object lists. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To reassign objects to an existing object list:

1. If you are selecting in the Modeler window, make sure that the modeler is in object selection mode by pressing the **O** shortcut key. This is not necessary for selecting in the History tree.
2. Select the objects you want to reassign.
3. Click **Modeler** > **List** > **Reassign**

   A dialog box with the existing object lists is displayed. (They appear in the History Tree under **Lists**.)

4. Select the list to which you want to assign the selected object(s) and click **OK**.

   The object is reassigned to the selected list, replacing previous list members. The *Objects* property of the List shows the objects contained in the list, with each name separated by a comma.

The object list will be treated as one volume when you are plotting and performing fields calculations. It will be listed in the *Geometry* window of the *Fields Calculator*, when you select **Volume**.

Using or Viewing List Objects

To view the objects included in an Object list:

1. In the History Tree, expand the **Lists** branch.
2. Right-click the ***list*** you want to select, and click **Select Assignment**.

   The objects that are included in that list are selected in the Modeler window, and the attributes appear in the docked *Properties* window.

To view the properties of the object list (including a list of the objects included):

1. In the History Tree, expand the **Lists** branch.
2. Under *Lists*, right-click the ***list*** you want to view and click **Properties**.

   The *Properties* dialog box appears. The objects included in the list appear as comma-separated names in the *Objects* row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** or **Cancel** to close the *Properties* dialog box.

To use an object from a list in another operation:

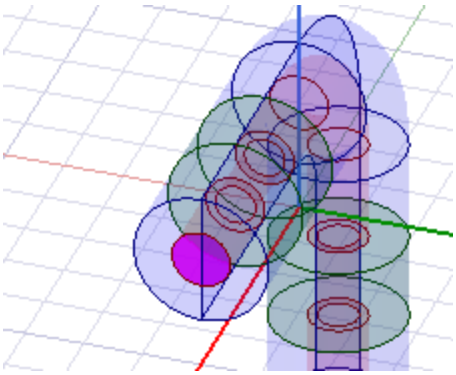
1. In the History Tree, expand the **Lists** branch.
2. Right-click the ***list*** you want to select and click **Select Assignment**.

   The objects that are included in that list are selected in the Modeler window, and the attributes appear in the docked *Properties* window.
3. Holding **Ctrl**, select any additional objects you might want to use in the operation.
4. Complete the operation.

   For example, you could select an object list and an additional object, and then execute one of the Boolean commands (such as *Unite* or *Subtract*).

Selecting Faces

If the modeler is in face selection mode, click an object face in the view window to select it. The selected face is rendered in the section color (magenta by default) and is relatively opaque by default. All other objects become relatively transparent.



When the mouse hovers over a face in the view window, that face is highlighted, which indicates that it will be selected when you click. Selected faces become the color specified under the *3D Modeler> Display> Rendering* section the *General Options*.

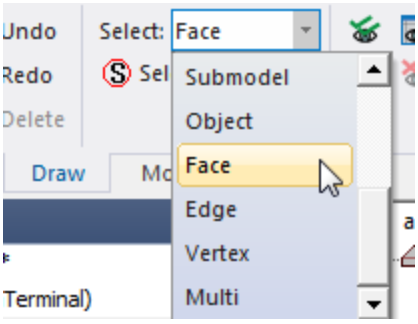
To select multiple faces, hold the **Ctrl** key as you click the faces. You also have the option to create face lists, which define a list of object faces, or you can make face selections from a Face ID list in the Select Face dialog box.

Switch to face selection mode using one of the following methods:

- From the menu bar, click **Edit> Selection Mode> Select Faces**
- Right-click in the Modeler window and choose **Selection Mode> Select Faces** from the shortcut menu.

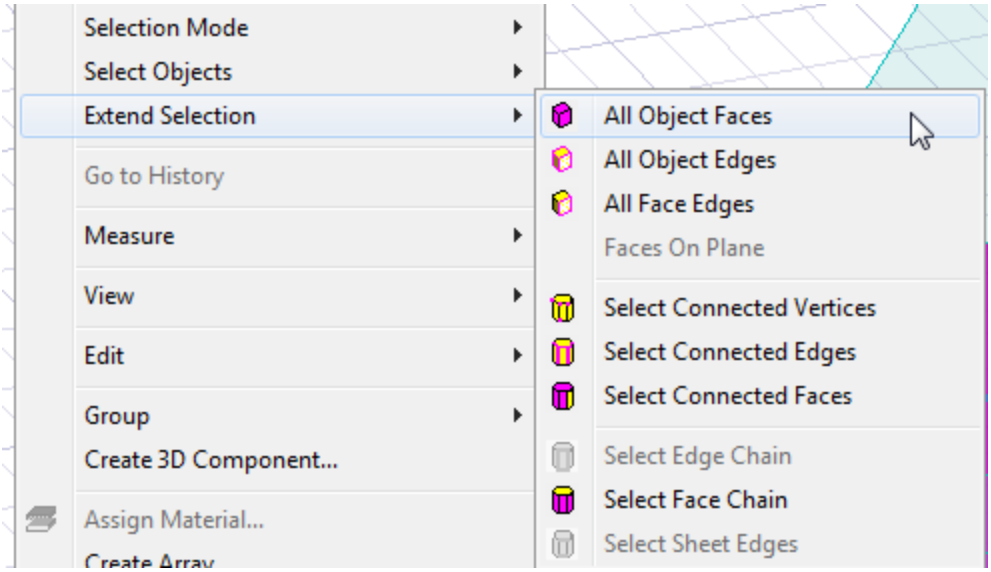
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- With the Modeler window active, press the **F** shortcut key.
- On the **Draw** ribbon tab, choose **Faces** from the **Select** drop-down menu.



You can also select faces in the Select Multi mode.

When you've selected a face, the menu bar's **Edit**> **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:



Various tools are available for face selection, as described in the *Related Topics* listed below.

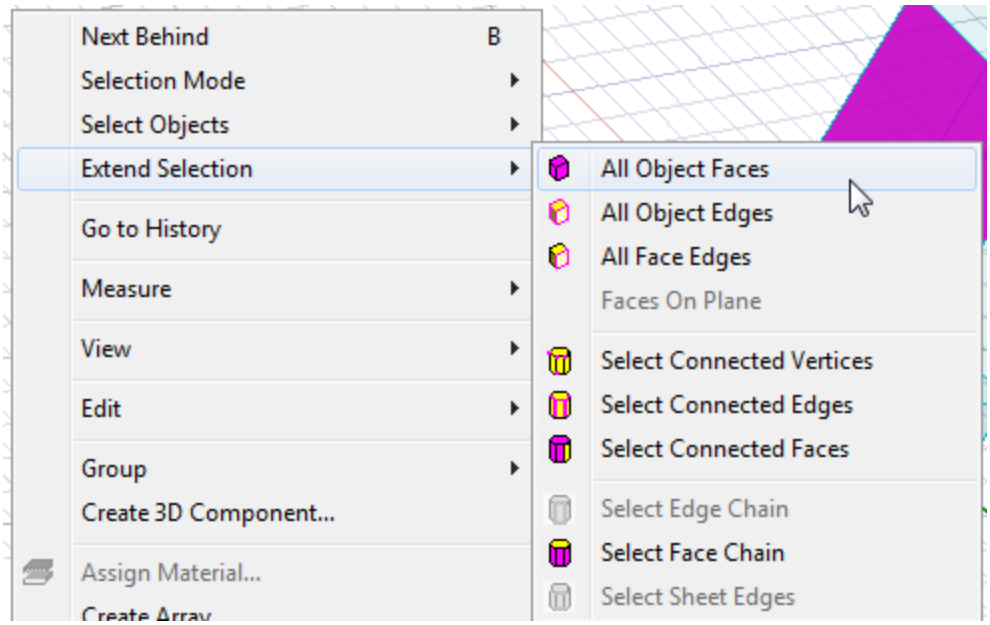
Selecting All Faces of an Object

1. With the Modeler window active, switch to the face selection mode by pressing the **F** shortcut key.
2. Click a face on the object of interest to select it.

   You can select a face on each of two or more objects if desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

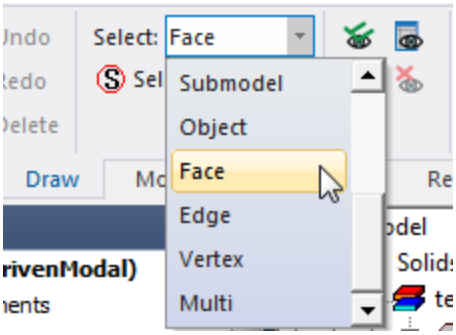
3. Use one of the following methods of selecting all faces of the objects of interest:
   - From the menu bar, click **Edit> Extend Selection> All Object Faces.**
   - Right-click in the Modeler window and choose **Extend Selection> All Object Faces** from the shortcut menu:



The selection is extended to include all faces of each object that had at least one face selected originally.

Selecting Faces by Name

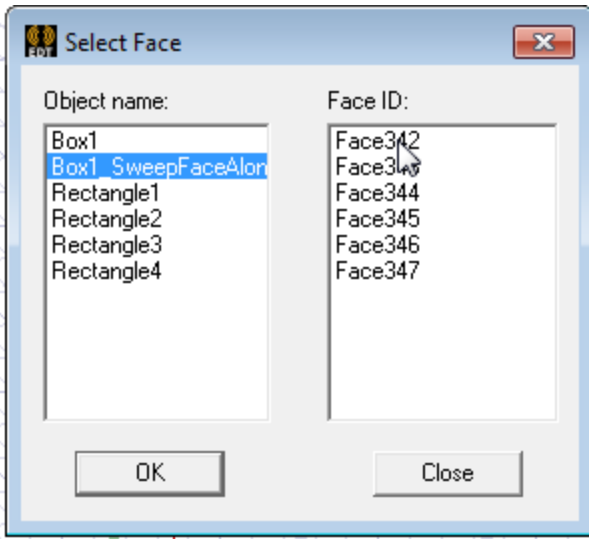
1. Make sure that the modeler is in face selection mode using one of the following methods:
   - With the Modeler window active, press the **F** shortcut key.
   - From the menu bar, click **Edit> Selection Mode> Faces**
   - On the **Draw** or **Model** ribbon tabs, select **Face** from the **Select** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the menu bar, click **Edit> Select Objects> By Name** or, on the **Draw** or **Model** ribbon tabs, click **Select by Name**.

   The *Select Face* dialog box appears:



3. In the *Object name* box, click the name of the object with the face or faces you want to select.

   The object's faces are listed in the *Face ID* box.

4. Click the ***face*** you want to select in the *Face ID* column. Use **Ctrl+click** to select more than one face or **Shift+click** to select a range of faces.

   The selected faces are highlighted in the list and in the Modeler window.

5. Click **OK**.

Selecting Faces by Plane

To select a face that is aligned with a global plane, use one of the following two methods.

Method 1:

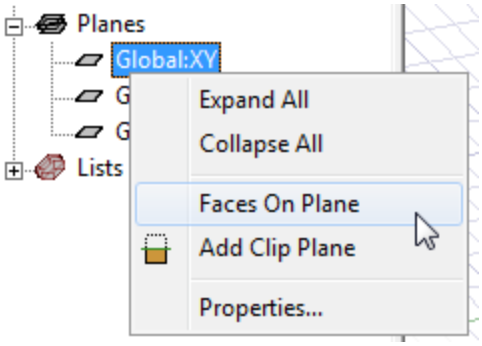
1. Make sure that the modeler is in face selection mode by pressing the **F** shortcut key.
2. In the History Tree, expand the **Planes** branch and select a plane to display (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. From the menu bar, click **Edit> Extend Selection> Faces on Plane** or right-click in the Modeler window and choose **Extend Selection> Faces on Plane**.

   All faces that lie on the chosen plane are selected.

Method 2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the History Tree, expand the **Planes** branch.
2. Right-click one of the listed planes (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. On the shortcut menu, click **Faces on Plane**.



All faces that lie on the chosen plane are selected.

Creating a Face List

Create a face list when you when you want to group multiple faces for convenient selection and processing. Creating a face list is a convenient way to identify and select a specific set of surfaces for field plots, calculations, boundaries or excitations. The same face can be included in several different lists.

To create a face list:

1. Make sure that the modeler is in face selection mode by pressing the **F** shortcut key.
2. Select the object faces you want to include in the face list.
3. From the menu bar, click **Modeler** > **List** > **Create** > **Face List**.

   The face list is created. It is listed in the History Tree under **Lists**. The default name is Facelist*n*. The lists appear in alphanumeric order.

4. Optionally, to change the name of a face list (for example, to a name describing the listed faces as ports or boundaries), select the list in the History Tree and change the **Name** value in the docked *Properties* window.

   If necessary, the lists in the History Tree are resorted to restore the proper alphanumeric order.

The face list will be treated as one selection of surfaces when you are plotting and performing fields calculations. The face list will be listed in the *Geometry* window of the *Fields Calculator*, when you select the *Surface* option.

- Line objects in the list do not affect where the fields are saved. If all objects in the list are lines, fields will be saved on all objects.

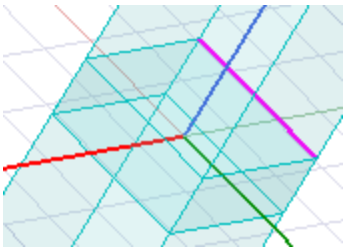
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To post processing on line object when saving fields on object/face list, include both the line and its containing object/face in the list.

If you right-click an existing face list and choose **Select Assignment** from the shortcut menu, you can make boundary assignments or execute the **Move Faces** geometry operations for all faces in the list.

Selecting Edges

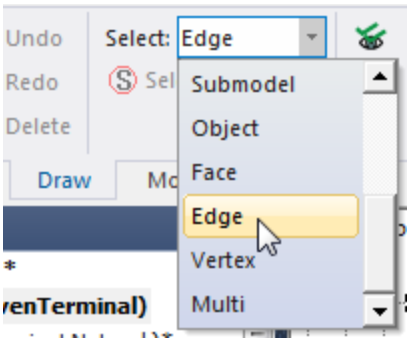
If the modeler is in edge selection mode, simply click an object's edge in the view window and it will be selected. The edge is rendered as nearly opaque using the default *Select* color. All other objects become relatively transparent.



When the mouse hovers over an edge in the view window, that edge is highlighted, which indicates that it will be selected when you click.

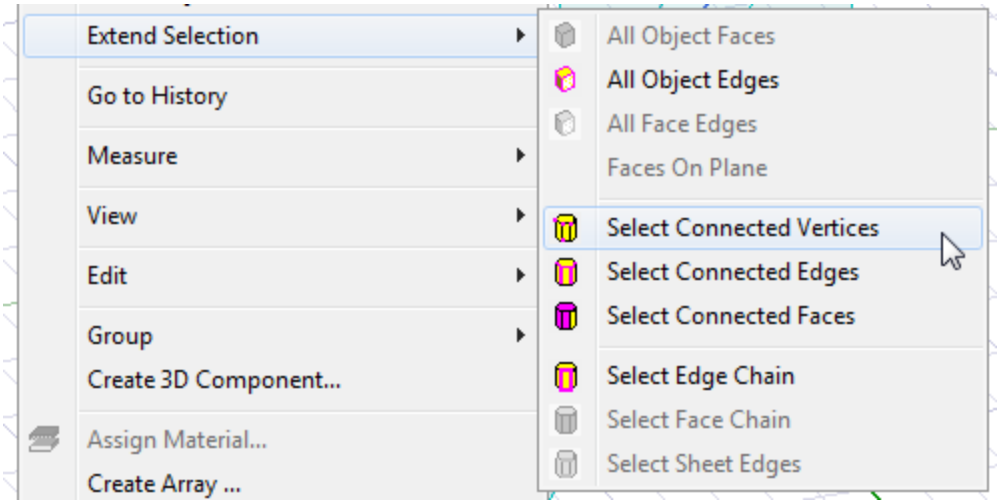
Switch to edge selection mode using one of the following methods:

- From the menu bar, click **Edit> Selection Mode> Edges**.
- Right-click in the Modeler window and click **Selection Mode> Edges**.
- With the Modeler window active, press the **E** shortcut key.
- On the **Draw** or **Model** ribbon tab, choose **Edge** from the **Select** drop-down menu.



Selecting an edge enables the following commands in the menu bar's **Edit> Extend Selection>...**submenu and the **Extend Selection>...** submenu that appears when you right-click in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use these commands to modify the current selection.

- All Object Edges
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain

You can also select multiple edges by holding the **Ctrl** key as you click additional edges.

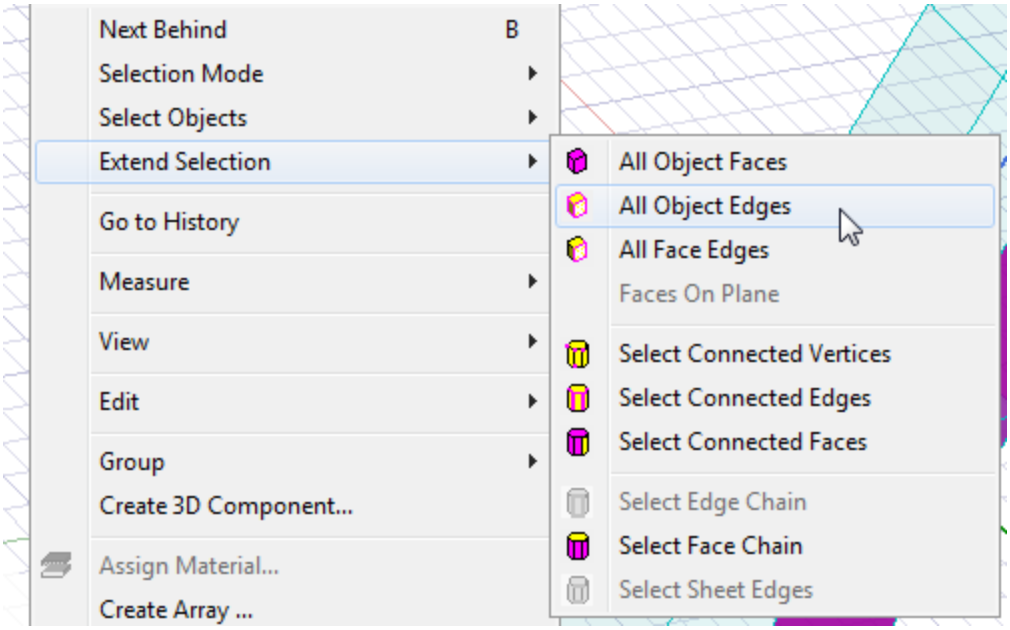
Selecting All Edges of an Object or Face

To select all edges of an object or face:

1. Select a single edge or face that belongs to the object for which you wish to select all edges.

   You may also select the object itself. The type of entity you initially choose affects which selection tools will be available in the next step:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you selected an **object** or an **edge** in step 1, the following *All Edge* command is available:
   - From the menu bar click **Edit> Extend Selection> All Object Edges** or right-click in the Modeler window and choose **Extend Selection> All Object Edges** from the shortcut menu.

     All edges of the object are selected.
3. If you selected a **face** in step 1, the following two *All Edge* commands are available:
   - From the menu bar click **Edit> Extend Selection> All Face Edges** or right-click in the Modeler window and choose **Extend Selection> All Face Edges** from the shortcut menu.

     All edges belonging to the face are selected.
   - From the menu bar click **Edit> Extend Selection> All Object Edges** or right-click in the Modeler window and choose **Extend Selection>All Face Edges** from the shortcut menu.

     All edges of the object to which the face belongs are selected.

If you selected multiple objects, all edges of those objects are selected.

Selecting Vertices

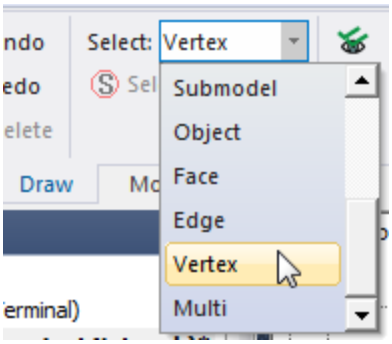
If the modeler is in vertex selection mode, simply click an object's vertex in the view window and it will be selected. To select multiple vertices, hold the **Ctrl** key as you click the vertices.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

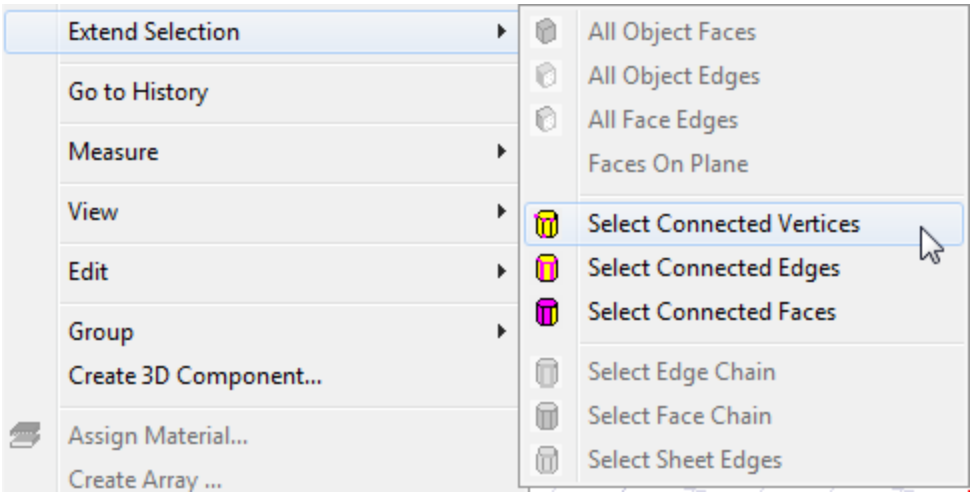
When the mouse hovers over a vertex in the view window, that vertex is highlighted, which indicates that it will be selected when you click. Selected vertices are rendered as using the default *Select* color. All other objects become relatively transparent.

Switch to vertex selection mode using one of the following methods:

- From the menu bar, click **Edit**> **Selection Mode**> **Vertices**.
- Right-click in the Modeler window and click **Selection Mode**> **Vertices**.
- With the Modeler window active, press the **V** shortcut key to begin the vertex selection mode.
- On the **Draw** or **Model** ribbon tabs, choose **Vertex** from the **Select**drop-down menu.



Selecting a vertex enables the following commands in the menu bar's **Edit> Extend Selection>...** submenu and the **Extend Selection>...** submenu that appears when you right-click in the Modeler window.



You can use these commands to modify the current selection:

- **Select Connected Vertices:** Selects the vertices of all edges that touch the initially selected vertices.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

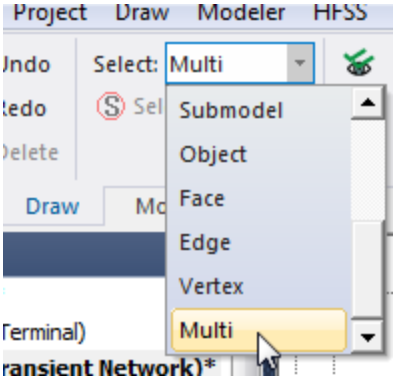
- **Select Connected Edges:** Selects all edges that touch the initially selected vertices.
- **Select Connected Faces:** selects all faces touching the initially selected vertices.

> **Note:**
>
> Multiple vertices can exist at the same coordinates, one vertex for each object that intersects at that point. There's no visual way to discern to which object a selected vertex belongs. If you click a vertex and use any of the commands listed above, only the entities that belong to the same object will be selected. To select the associated entities for all objects, start by box-selecting the vertices at the point of interest (clicking and dragging from left-to-right). This method selects all coincident vertices at the point. Then, use one of the above three commands.
>
> You can also pick a vertex and press **B** (*Next Behind*) to select a different vertex at the point of interest. Use the trial and error method to select the related vertices, edges, or faces of the desired object. For example, you may have to press *B* two or more times to select the vertex of the desired object.

Selecting Multi

The Select Multi mode permits you to select a mixture of objects, faces, vertices, or edges, depending on where you click. This mode can be very useful in conjunction with Measure Mode, for measuring the distances between different entities. Enter the *Multi* selection mode using one of the following methods:

- With the Modeler window active, press the **M** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Multi** from the shortcut menu.
- From the menu bar, click **Edit**> **Selection Mode**> **Multi**.
- On the **Draw** or **Model** ribbon tabs, choose **Multi** from the **Select** drop-down menu.

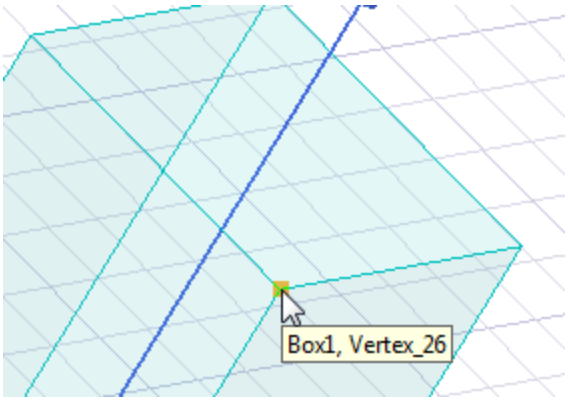


With Multi mode active:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To select a vertex, click near a vertex, within 10 pixel radius.
- To select an edge, click near an edge (and at least 10 pixels away from any vertex).
- To select an object, click little farther from any edge (between 10 and 20 pixels away).
- To select a face, click anywhere else on the interior of a face.

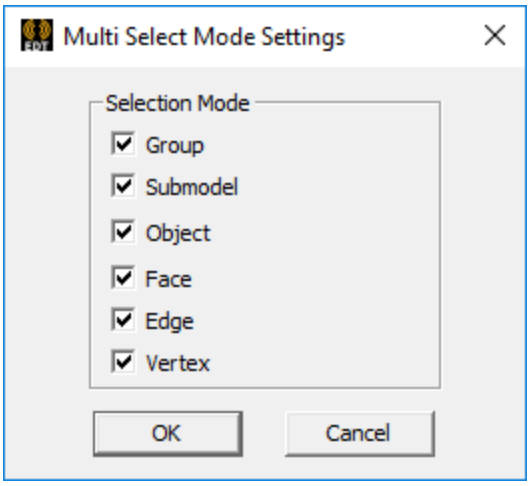
As you hover the cursor over an entity, a Tooltip indicates the type/ID of the entity (object name in the case of objects, Face_id in the case of faces, and so on). For example, this feature helps you to distinguish between the selection of the face of a sheet object versus a sheet object.



By holding down the **Ctrl** key, you can make multiple selections.

Controlling the Selection in Multi Mode

You can control the behavior of this selection mode. From the menu bar, click **Edit> Selection Mode> Multi Mode Settings**. This action displays a dialog box with check boxes for enabling the selection of specific types of entities (Group, Submodel, Object, Face, Edge, and Vertex):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clear a box filters out the associated item, preventing that type of entity from being selected while the *Multi* selection mode is active.

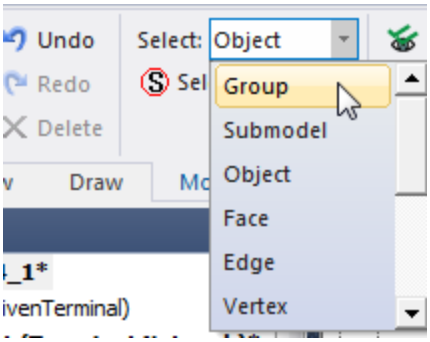
Selecting Groups and Submodels

With the *Group* selection mode, the parent group of selected object is selected. Selecting a group highlights all children under the group in the Modeler window. If you click an object that has not been assigned to a group, nothing is selected. To select multiple groups, hold the **Ctrl** key as you click the groups.

The *Group* selection mode is only available when the model contains at least one defined group.

Use one of the following methods of setting the selection mode to *Group*:

- From the menu bar, click **Edit**> **Selection Mode**> **Groups**.
- With the Modeler window active, press the **G** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Groups** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Group** from the **Select** drop-down menu. You may have to scroll up to see this option.



If your design contains submodels, you can set the selection mode to *Submodel*:

- From the menu bar, click **Edit**> **Selection Mode**> **Submodels**.
- With the Modeler window active, press the **U** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Submodels** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Submodel** from the **Select** drop-down menu. You may have to scroll up to see this option.

Clearing a Selection

To clear an object, face, edge, or vertex selection, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click in an empty area of the Modeler window.
- To clear an object selection, click a point away from the object name in the History Tree.
- From the menu bar, click **Edit> Deselect All**.
- Press **Shift+Ctrl+A**.

The items are no longer selected.

Selecting the Face, Edge, Vertex, or Object Behind

To select the face, edge, vertex, or object, behind another selected face, edge, vertex, or object, do one of the following:

- From the menu bar, click **Edit**> **Next Behind**.
- Right-click in the Modeler window and choose **Next Behind** from the shortcut menu.
- Press the **B** shortcut key.
- Press **Ctrl+B**.

> **Note:**
>
> The desired entity must be behind the point where you clicked, not just behind the initially selected face, edge, vertex, or object.

This option is useful when you are trying to select a face, edge, vertex, or object that is in the interior of a model, or when you do not want to change the model viewpoint to select an item.

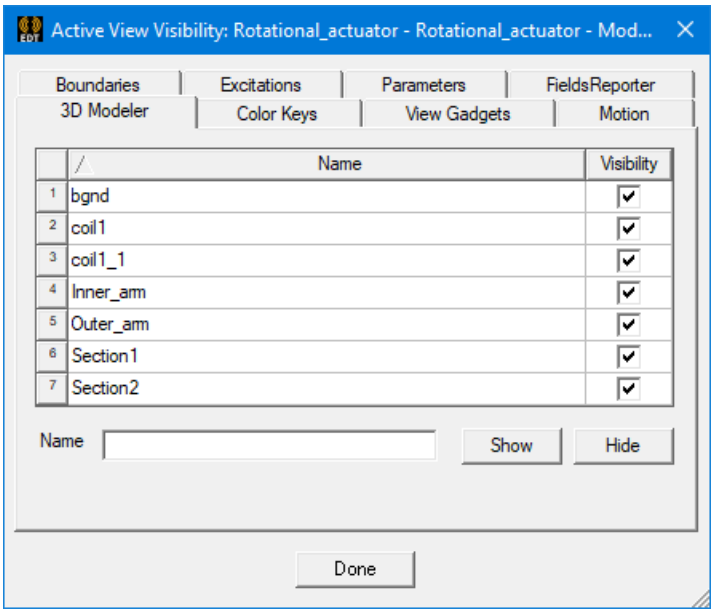
Selecting Objects and Surfaces That Lie Inside Other Objects

To select objects and faces that lie inside other objects (such as an object that lies within an air box, a conductive shield, or the background object), do one of the following:

- From the menu bar, **View** > **Visibility** > **Active View Visibility...** to access the *Active View Visibility* dialog box. Then, under the *3D Modeler* tab, use the check boxes in the **Visibility** column to control which objects are visible and which are hidden.

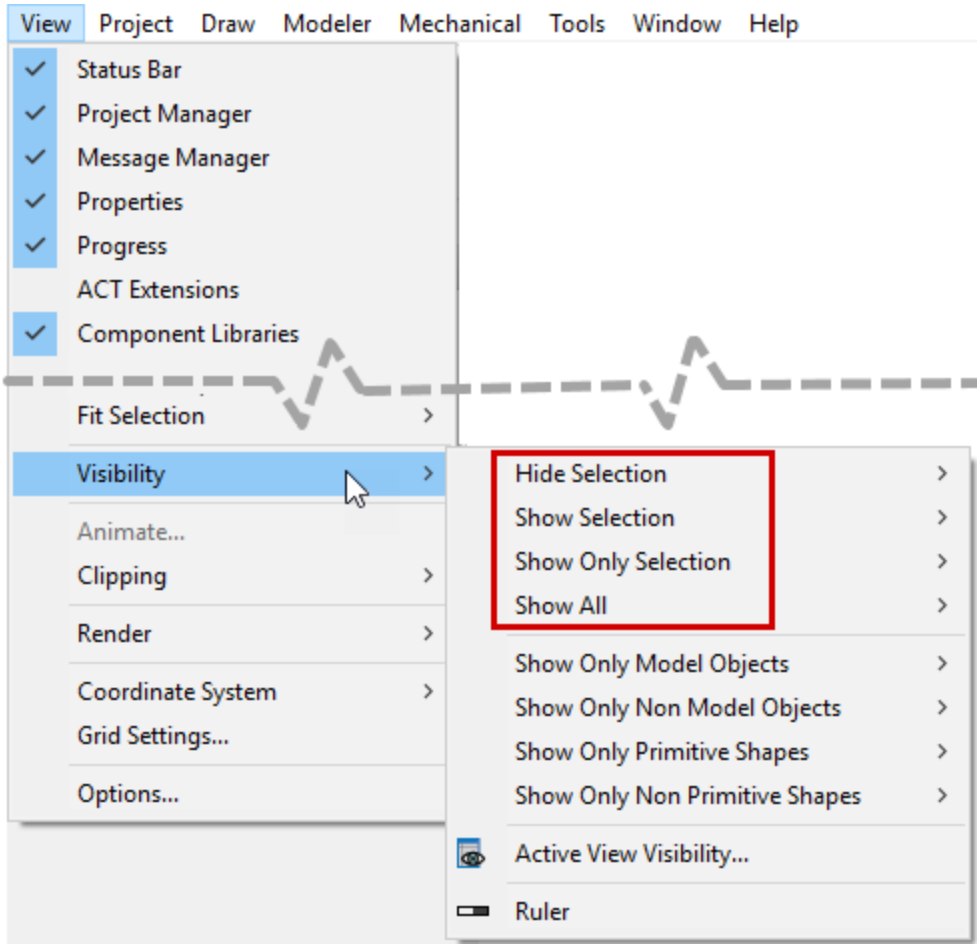
  This method, and the method that follows, is useful when you want to select objects using the mouse, since the mouse cannot select invisible objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

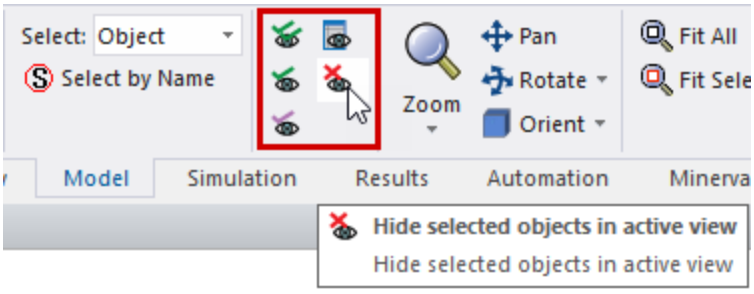


- Make the objects on the outside of the model invisible using the menu bar's other **View** > **Visibility** > commands:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



or the same commands on the **View, Draw,** or **Model** ribbon tabs:



- Use the **Select by Name** command to select Objects or Faces inside the model.
- Use the **Next Behind** command in the *Edit* menu or Modeler shortcut menu, or use the **B** shortcut key. This command selects the object that lies behind the one you initially selected. This command does nothing if no objects have previously been selected or if no object is behind the one you initially selected at the location you clicked.

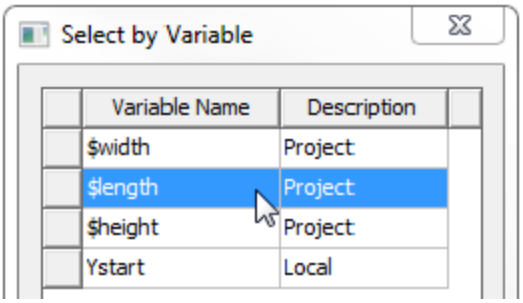
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects by Variable

You can select an object based on a variable that affects it. If your design includes variables, do so as follows:

1. From the menu bar, click **Edit> Select Objects> By Variable**.

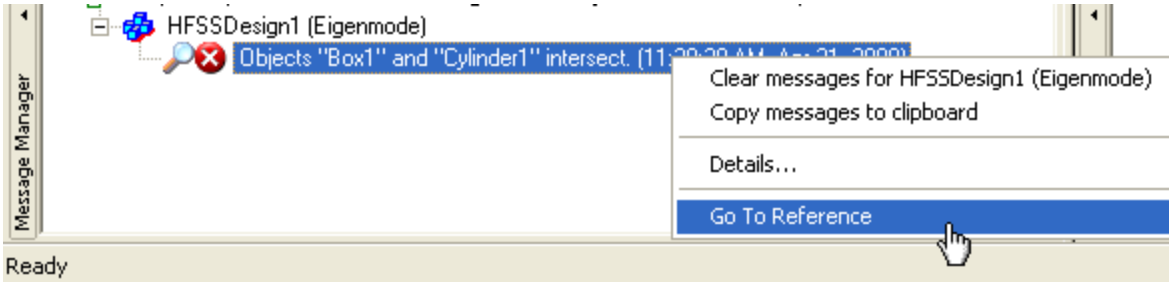
   The *Select by Variable* dialog box appears, which lists the variables in your design. Both columns are sortable by clicking the header. You can resize and move the dialog box. When you next open it, it remembers that size and location.



2. Select the variable of interest and click **OK**.

   The dialog box closes, and the object affected by the variable is highlighted in the Modeler window.

   The Message window contains a reference that you can select and use to go to the affected object.



   If you execute the command again, without clearing the current selection(s), the additional object can be highlighted. You can resize and move the dialog.

Assigning Coordinates to New Objects

When you insert and draw new 3D, 2D, or 1D objects in model, you may need to define coordinate systems, locations of points, distances between objects, and other geometry items.

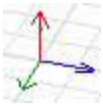
- Setting the Reference Point
- Defining Cartesian Coordinates

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining Spherical Coordinates
- Defining Relative Coordinates
- Defining Absolute Coordinates

Setting the Reference Point

When you draw objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis:



To change the reference point:

1. Select the drawing command to use.

   The Measure Data dialog box opens. As you move the cursor over the Modeler window, the top line in the *Measure Data* dialog box shows the coordinates of the current reference point.

2. Move the cursor to the desired reference point and press **Ctrl+click** or right-click and select **Set Reference Point** from the short cut menu.

   This action moves the reference point marker to the new location. The *Measure Data* dialog box updates. The coordinates boxes in the Status Bar change to accept Relative distance information. If you prefer, rather than setting the reference point with the cursor, you can press **Tab** to jump to a coordinate box in the status bar, where you can specify the coordinates numerically. Navigate among the three coordinate boxes by pressing the **Tab** key again.

Defining Cartesian Coordinates

When you draw an object, define a point using Cartesian coordinates by typing the point's x, y, and z distances from the origin in the **X, Y,** and **Z** text boxes, respectively. When defining a second point, specify its x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

1. Select the desired drawing command.
2. Select **Cartesian** from the drop-down menu in the Status Bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the point's x, y, and z coordinates in the **X, Y,** and **Z** text boxes.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
4. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

Defining Cylindrical Coordinates

To define a point using cylindrical coordinates, specify the point's radius, measured from the origin, in the **R** text box, the angle from the x-axis in the **Theta** text box, and the distance from the origin in the z direction in the **Z** text box. When defining a second point, specify its distance from the previously selected point in the in **dR, dTheta,** and **dZ** text boxes.

1. After clicking the desired drawing command, select **Cylindrical** from the pull-down menu in the Status Bar.
2. Type the point's r, theta, and z coordinates in the **R, Theta,** and **Z** boxes.

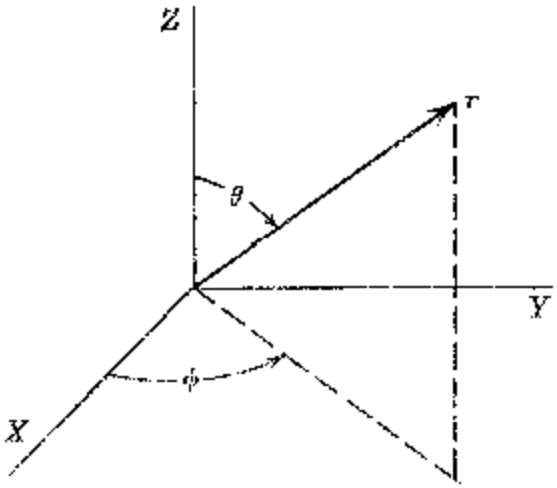
   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dR, dTheta,** and **dZ** text boxes.

Defining Spherical Coordinates

To define a point in spherical coordinates, specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the positive z-axis in the **Theta** text box, and the angle from the positive x axis in the **Phi** text box. When selecting a second point, specify its distance from the previously selected point in the in **dRho, dTheta,** and **dPhi** text boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Even though you are inputting spherical coordinates, all data is internally stored in Cartesian coordinates.

To define a point in spherical coordinates:

1. After clicking the desired drawing command, select **Spherical** from the drop-down menu in the status bar.
2. Type the point's r, theta, and phi coordinates in the **Rho, Theta,** and **Phi** text boxes in the Status Bar.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dRho, dTheta,** and **dPhi** text boxes.

Using Absolute Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0). This entry mode is the default setting for the first point you select after clicking a drawing command. Relative coordinates are relative to the reference point, or the previously selected point.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To enter a point's absolute coordinates:

1. Click the desired drawing command.
2. Select **Absolute** from the **Absolute/Relative** drop-down menu in the Status Bar.
3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

> **Note:**
>
> When drawing objects other than polylines and helices, by default, the second point you select is relative to the first point; **Relative** is automatically selected in the **Absolute/Relative** drop-down menu in the Status Bar. Be sure to select **Absolute** from this drop-down menu if you want the second point to be relative the origin of the working coordinate system.

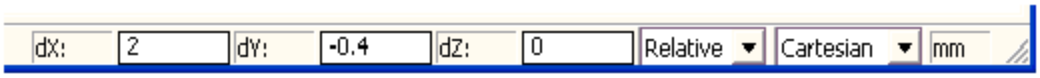
Using Relative Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Relative coordinates are relative to the reference point or the previously selected point. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0).

To enter a point's relative coordinates:

1. Click the desired drawing command.
2. Select **Relative** from the **Absolute/Relative** drop-down menu in the Status Bar.

   When you are in *Relative* mode, the text boxes for a coordinate show an "d" before the coordinate description, to indicate "distance from" the working reference. For example:

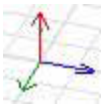


3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

Choosing the Movement Mode

When drawing objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis when you click a drawing command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To change the reference point, move the cursor to the desired point and **Ctrl+click** the point.

When creating 1D, 2D, or 3D geometry graphically (that is, by clicking points instead of by defining coordinates numerically), points that you can select are dependent on the current *Movement Mode* setting. You can move the cursor and place a point at one of the following locations:

- In the same plane as the reference point (In Plane movement mode)
- Perpendicular to the reference point (Out of Plane movement mode)
- If an object is present, you can snap to one of its points – any snapable point in 3D space (3D movement mode).
- Along the X-axis of the working coordinate system (CS)
- Along the Y-axis of the working CS
- Along the Z-axis of the working CS

Drawing commands that you choose may temporarily override the current movement mode. For example, when you draw a box or cylinder, the first two points you click (defining the base) may be in the drawing plane, but the third (defining the height) must be out of plane relative to the base. In this case, you do not have to manually change the movement mode; it is automatically overridden for the third click.

> **Note:**
>
> For all six movement modes covered in this section, the following rules apply regarding mode persistence:
>
> - If you change the movement mode *before* executing a drawing command, the new mode setting will be retained until you change it again.
> - If you change the movement mode *after* executing a drawing command, it will be treated as a temporary override. Upon completion of the drawing operation, the mode will revert to the previous setting.

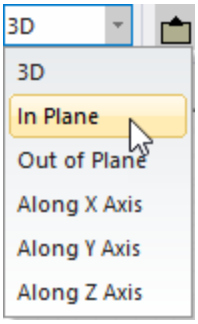
Moving the Cursor In Plane

Use the *In Plane* movement mode to specify points of objects being drawn that are *on the same plane* as the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

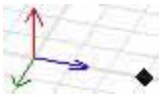
- From the menu bar, click **3D Model**> **Movement Mode**> **In Plane**.
- On the **Draw** ribbon tab, choose **In Plane** from the **Movement mode** drop-down menu:



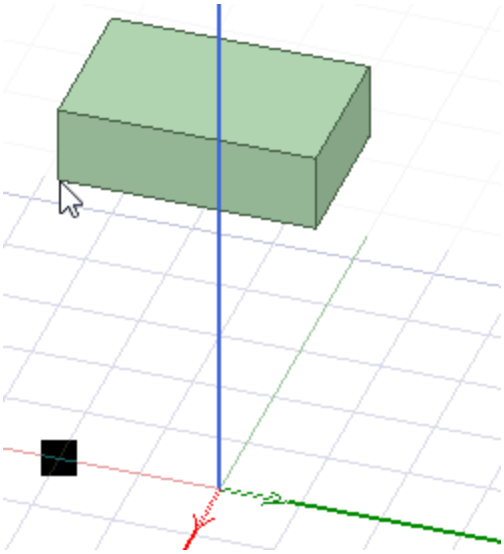
The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **In Plane** from the shortcut menu.

The next point you select will be on the same plane as the reference point. The cursor's location is displayed with a black diamond that indicates where it will snap to the drawing plane grid.



If you click a snapping point on an object in 3D space (that is, one not on the drawing plane), the point location is projected to the drawing plane along a perpendicular vector. In the following example, a filled black square indicates the projected snapping point:



The symbol used for projected point matches the type of snapping point on the object (vertex, edge center, face center, and so on).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

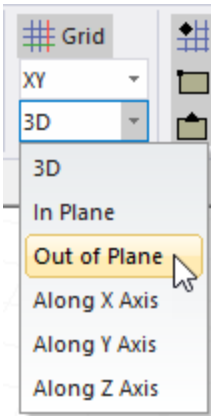
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Out of Plane

Use the *Out of Plane* movement mode to specify points of objects being drawn that are along a vector *perpendicular* to the drawing plane and intersecting it at the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

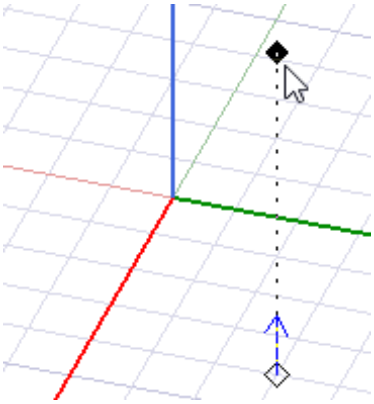
- From the menu bar, click **3D Model**> **Movement Mode**> **Out of Plane**.
- On the **Draw** ribbon tab, choose **Out of Plane** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **Out of Plane** from the shortcut menu.

A dashed line perpendicular to the drawing plane is displayed, and a filled black diamond indicates the cursor's location:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

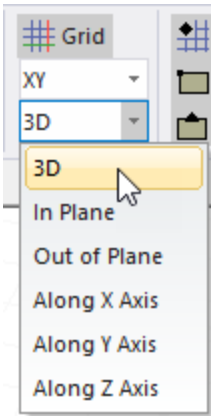
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor in 3D Space

To snap to drawing plane grid points or to object snapping points anywhere in 3D space, use the *3D* movement mode.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **3D**.
- On the **Draw** ribbon tab, choose **3D** from the **Movement mode** drop-down menu:

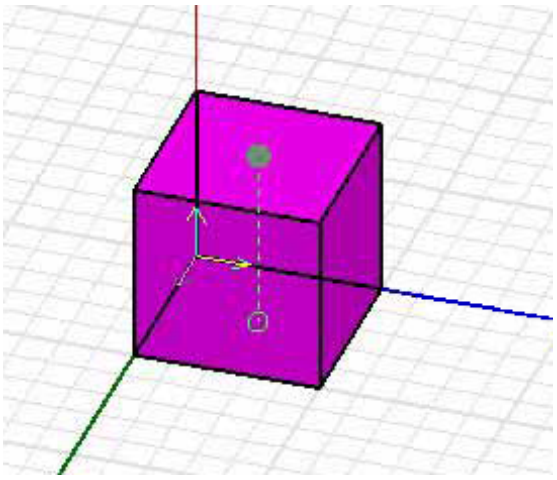


The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and click **Movement Mode**> **3D** from the shortcut menu.

If one of an object's snapping points (vertex, edge center, face center, and so on) is within snapping range of the cursor, the geometry you are drawing will snap to that point. The symbol will indicate the type of snapping point. In the following example, the cursor will snap to the face center point of the boxes top face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If an object is not within snapping range of the cursor, the *3D* movement mode is identical to the *In Plane* mode. That is, the geometry being drawn will snap to a grid point on the drawing plane.

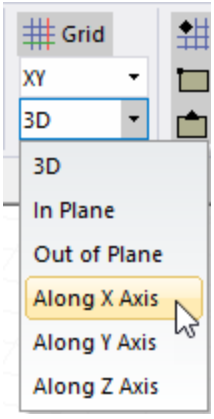
In either case, the *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the X-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- X direction relative to the reference point. Movement is constrained to the X direction as defined by the working coordinate system, not necessarily the global X direction. The line of motion passes through the reference point, so it may be parallel to the X-Axis and not necessarily along the actual X-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along X Axis**.
- On the **Draw** ribbon tab, choose **Along X Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press and hold the **X** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode**> **Along X Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative X direction relative to the reference point.

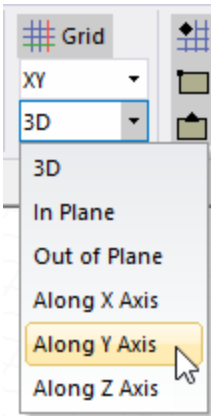
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the Y-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Y direction relative to the reference point. Movement is constrained to the Y direction as defined by the working coordinate system, not necessarily the global Y direction. The line of motion passes through the reference point, so it may be parallel to the Y-Axis and not necessarily along the actual Y-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along Y Axis**.
- On the **Draw** ribbon tab, choose **Along Y Axis** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **Along Y Axis** from the shortcut menu.
- Press and hold the **Y** key while clicking the next point.

The next point you select will be on the same plane as the reference point and in the positive or negative Y direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

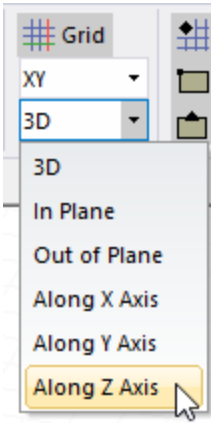
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving the Cursor Along the Z-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Z direction relative to the reference point. Movement is constrained to the Z direction as defined by the working coordinate system, not necessarily the global Z direction. The line of motion passes through the reference point, so it may be parallel to the Z-Axis and not necessarily along the actual Z-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along Z Axis**.
- On the **Draw** ribbon tab, choose **Along Z Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

- Press and hold the **Z** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode**> **Along Z Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative Z direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

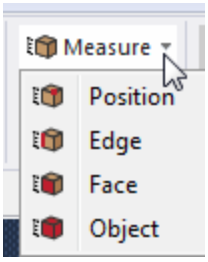
Measure Modes

The Measure modes lets you measure the position, length, area, and volume of objects. With two faces selected, with two edges selected, or with an edge and a face selected, the Measure Mode displays the angle and distance between them. The Measure Position mode dynamically measures the distance between a reference point and the cursor location. You can pause

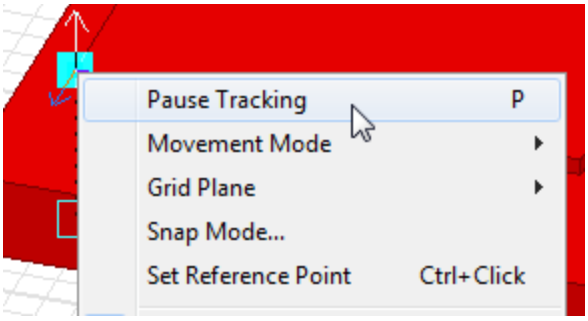
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic tracking in order to copy text information from the *Measure Data* window, and then Resume dynamic updating.

1. To access a Measure mode, do one of the following:
   - From the menu bar, click **Modeler** > **Measure** > [**Position, Edge, Face,** or **Object**].
   - Right-click in the Modeler window and choose **Measure** > [**Position, Edge, Face,** or **Object**] from the short-cut menu.
   - On the **Draw** ribbon tab, choose **Position, Edge, Face,** or **Object** from the **Measure** drop-down menu.



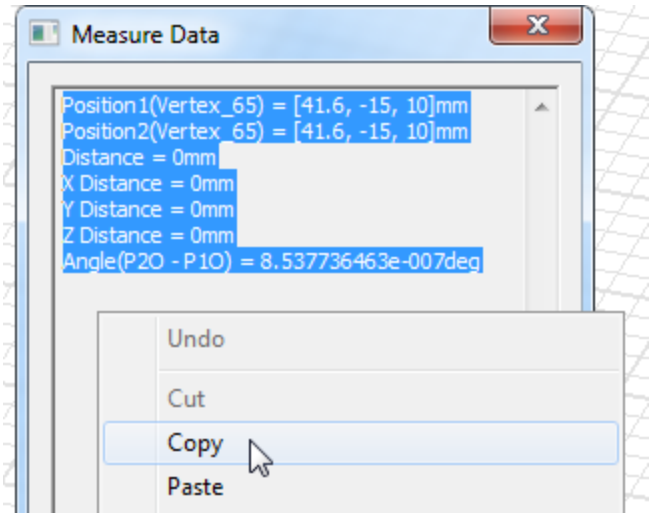
2. Select items to measure as described below under Measurement Modes and Data.
3. Optionally, to *Pause* the dynamic tracking, do one of the following:
   - Right-click in the Modeler window and select **Pause Tracking**:



   - Press **P**.

   While paused, you can move the cursor without changing the *Measure Data* window contents. You can also copy and paste the data to a text file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To resume measurement tracking, do one of the following:

- Right-click in the Modeler window and choose **Resume Tracking**.
- Press **P** again.

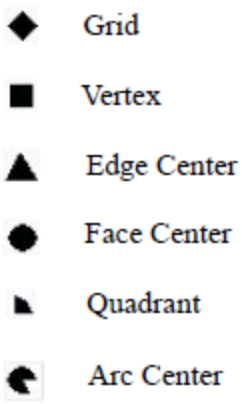
4. To exit the Measure mode, do one of the following:
   - Click **Close** on the *Measure Data* window.
   - Press **Esc**.

Measurement Modes and Data:

- **Position:** Use the following procedure to measure the position of points and the distance between them:
  1. Click a snap point anywhere on the model, which becomes *Position 1* (the measurement reference point).
  2. Hover over another snap point but do not click the mouse (or **Ctrl+click** the second point to redefine the reference point but retain the measurement data).

     The following symbols indicate the available types of snap points:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> As you move the cursor, the Measure Data window displays the reference point, current cursor location, and measurement information. Clicking on a new vertex or other snap point updates the reference to the new location and resets the measurement results to zero. To resent the reference point location without clearing the measurement data, **Ctrl+click** the point.

The *Measure Data* window gives you the following information:

- Position1 coordinates
- Position2 coordinates
- Distance between Position1 and Position2
- X Distance: X-component of the Distance.
- Y Distance: Y-component of the Distance.
- Z Distance: Z-component of the Distance.
- Angle: The angle of the line connecting Position1 and Position2

- **Edge:** The *Measure Data* window gives you the following information:
  - Edge name
  - Length
  - Reference coordinate system
  - Start Vertex Position (with Vertex name)
  - End Vertex Position (with Vertex name)
  - Additional data for circles and arcs only:
    - Arc Center Position
    - Arc Radius
    - Arc Diameter
    - Plane

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Face:** Each item listed below only appears in the *Measure Data* window when applicable to the selected face.
  - Face ID
  - Area
  - Perimeter
  - Cylinder Radius or Sphere Radius
  - Cylinder Diameter or Sphere Diameter
  - Plane (orientation of the plane – the face may be parallel to the listed plane and does not necessarily lie on it)
  - Plane Normal (normal vector for planes not aligned with a global or relative coordinate system)
  - Reference coordinate system
  - Center Position
  - No. of Loops
  - No. of Edges
  - No. of CoEdges
  - No. of Vertices
- **Object:** Each item listed below only appears in the *Measure Data* window when applicable to the selected object.
  - Volume(with *ObjectName* — zero for shell objects)
  - Area(with *ObjectName* — total surface area for solids and shells)
  - No. of Lumps
  - No. of Shells
  - No. of Faces
  - No. of Loops
  - No. of Edges (the total number of unique edges)
  - No. of CoEdges (the total number of coincident edges, where adjacent faces intersect)
  - No. of Vertices

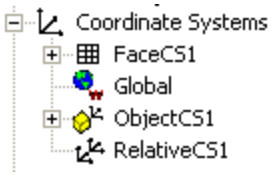
Setting Coordinate Systems

The modeler has four types of coordinate systems (CS) that enable you to easily orient new objects:

- Global CS
- Relative CS
- Face CS
- Object CS

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Every coordinate system has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes. The default Global coordinate system and any additional coordinate systems that you create for a project appear in the History Tree of the Modeler window.



- The *Global* coordinate system (CS) is the fixed, default CS for each new project. It cannot be edited or deleted.
- A *Relative CS* is user-defined. Its origin and orientation can be set relative to an existing CS. Relative CSs enable you to easily draw objects that are located relative to other objects. If you modify a relative CS, all objects drawn on that CS will be affected and change position and/or orientation accordingly. You can define a relative CS to be *Offset* and/or *Rotated* from an existing CS. This feature provides a way for objects made of the same anisotropic materials to have different orientations.

  When you set a new relative coordinate system, you specify whether to express the coordinates as Absolute or Relative Coordinates. Absolute uses the specified values in terms of the global coordinate system. Relative interprets the values as differences from the current working CS.

  You have choices for expressing the coordinates as Cartesian, Cylindrical, or Spherical. These are evaluated as Cartesian for the coordinate system properties.

- A *Face CS* is also user-defined. Its origin is specified on a planar object face. Face CSs enable you to easily draw objects that are located relative to an object's face.
- An *Object CS* is user-defined as attached to a specific object.

Switch between global, relative, object and face CSs by changing the *working CS*. Simply click the CS you want to use in the History Tree. The working CS is indicated by a red *W* that appears at the lower-left corner of the CS name in the History Tree (as shown on Global in the figure above). The docked **Properties** window lists the CS associated with an object as the *Orientation*. By default, the orientation is Global, but if you have created the object under a different coordinate system, that CS will be shown for the orientation. You can click on the current orientation to see a drop-down menu of other orientations that you can assign for an object.

User-defined CSs are saved with the active project. When you open a project, the CS designated as working CS when you last saved it is active.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Working Coordinate System

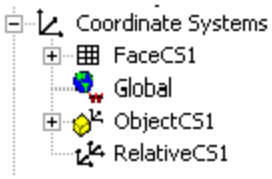
The working coordinate system (CS) is the current CS with which objects being drawn are associated. The working CS can be the Global CS or a user-defined Relative, Object, or Face CS. Select the working CS by clicking its name in the History Tree, or follow this procedure:

1. Click **Modeler> Coordinate System> Set Working CS**.

   The *Select Coordinate System* dialog box appears.

2. Click a CS in the list.
3. Click **Select**.

   A red *W* appears at the lower-left corner of the CS name in the History tree, indicating that it is the working CS. In this following figure the Global is the working CS.



   Objects that you draw hereafter will be associated with the CS you selected.

Creating a Relative Coordinate System

When creating a relative CS, you have the following options:

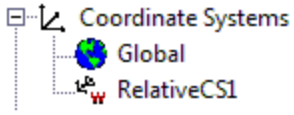
- You can create an offset relative CS, that is, a relative CS whose origin lies a specified distance from another CS's origin. By moving a CS's origin, you can enter coordinates relative to an existing object, without having to add or subtract the existing object's global coordinates.
- You can create a rotated relative CS, that is, a relative CS whose axes are rotated away from another CS's axes. By rotating the axes of a CS, you can easily add an object that is turned at an angle relative to another object.
- You can also create a relative CS that is both offset and rotated.
- You can click **Tools> Options> General Options** to display the *Options* dialog box.

  Under the *3D Modeler> Drawing* options, choose your preferred *Relative Coordinate System Creation Mode*. The **Axis/Position** option uses the cursor or coordinate entry boxes in the Status bar to define the CS. The **Euler angle** option causes the *Set Origin, Set Euler Angles,* or *Set Origin and Euler Angles* dialog box to display (depending on the type of Offset CS you are creating). You define the origin and angles numerically in the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also press **F3** or **F4** after clicking a *RelativeCS* command to switch between the two entry modes on the fly.

After you have created a relative CS it appears in the History Tree:

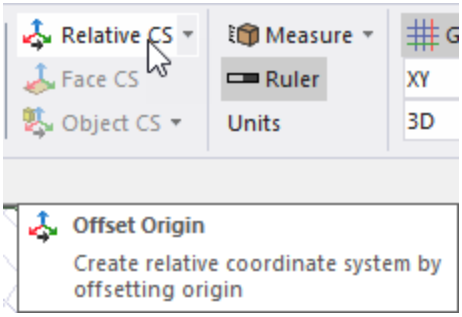


Selecting the CS in the History Tree causes the docked *Properties* window to show the CS properties.

Creating an Offset Relative CS

To create a relative CS with an origin that lies a specified distance from another CS's origin:

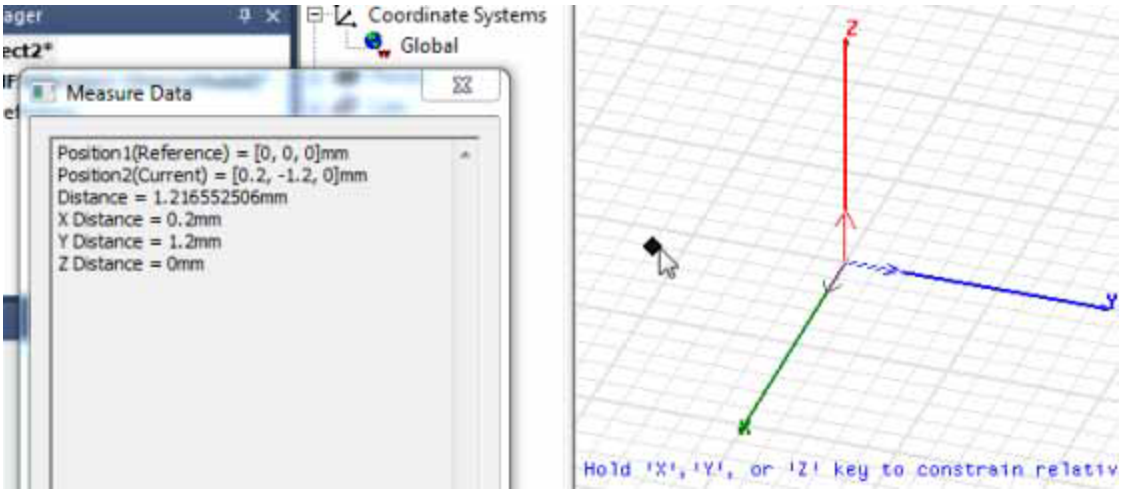
1. In the History Tree, click the CS upon which you want to base the new relative CS, making it the working CS.
2. From the menu bar, click **Modeler> Coordinate System> Create> Relative CS> Offset** or, on the **Draw** ribbon tab, click **Relative CS**. (You do not have to access the drop-down menu because *Offset* is the default option for a *Relative CS* in the ribbon:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options**, *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can select the origin in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – Use one of the following two coordinate entry methods:
     - Specify the coordinates graphically (click to select):

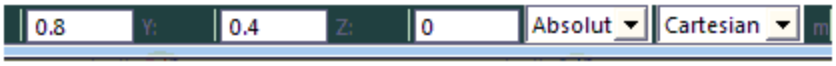
       Use the cursor to click the point. You can see the coordinate information in the *Measure Data* window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

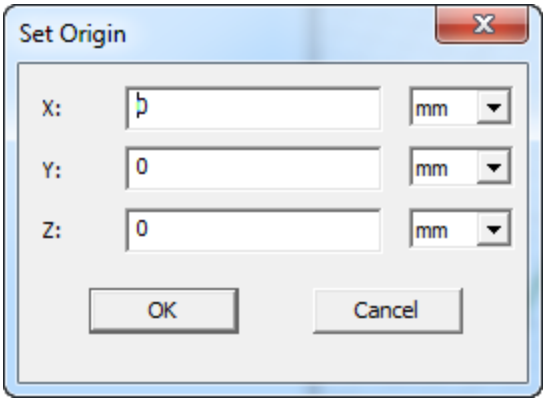
- Specify the coordinates numerically (using the coordinate entry text boxes in the Status Bar):



  a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
  b. Select either Relative or Absolute coordinates from the other drop-down menu.
  c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.

- **Euler angle** option selected, or **F4** pressed:

  In the *Set Origin* dialog box, specify the **X, Y,** and **Z** coordinates and units and click **OK**.



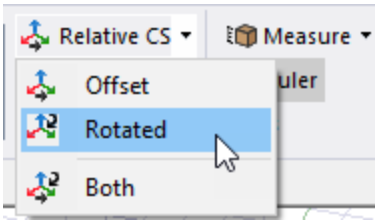
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new relative CS is created. Its origin has moved from the previous working CS, but its axes remain the same. It is listed in the History Tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Creating a Rotated Relative CS

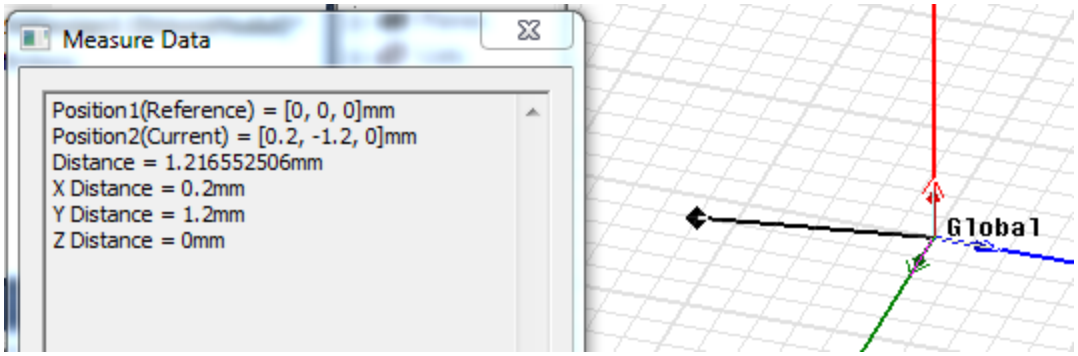
To create a new relative CS with its axes rotated away from another CS's axes:

1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler> Coordinate System> Create> Relative CS> Rotated** or, on the **Draw** ribbon tab, click **Relative CS> Rotated**:



.

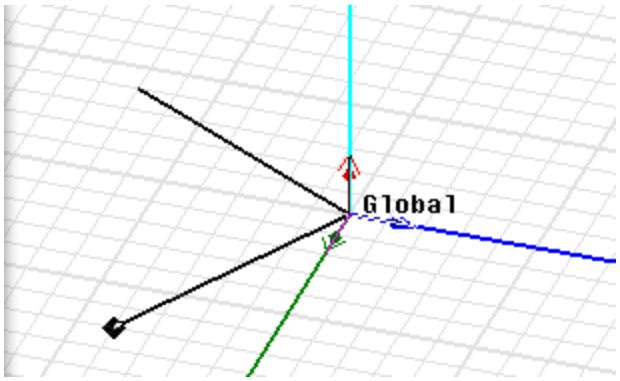
3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the axes in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
     - Specify the X-axis and XY plane graphically (click to select), as follows:
       a. Use the cursor to click a point along the desired X-axis. You can see the coordinate information in the *Measure Data* window.



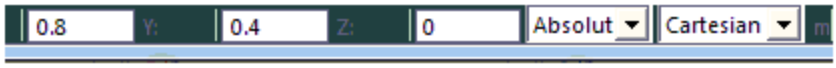
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

b. In the same manner, select any point lying on the XY plan that's not on the X-axis.



- Enter the coordinates numerically in the Status Bar, as follows:



a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).

b. Select either Relative or Absolute coordinates from the other drop-down menu.

c. Type the coordinates of a point on the desired X-axis in the X, Y and Z (or dX, dY, and dZ) text boxes and press **Enter**.

d. In the same manner, type the coordinates of a point lying on the XY plane, but not on the X-axis, and press **Enter** again.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

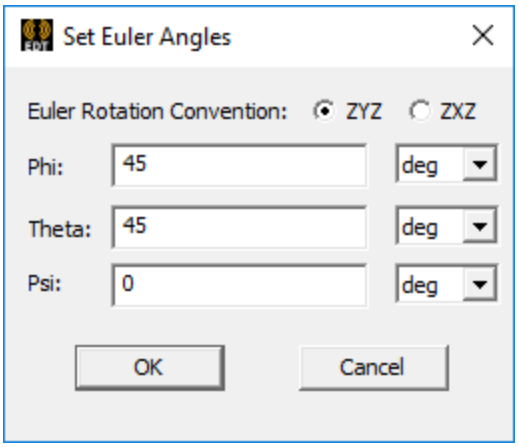
- **Euler angle** option selected, or **F4** pressed:

a. In the *Set Euler Angles* dialog box, select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).

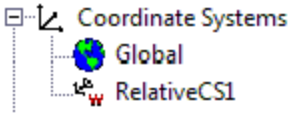
b. Specify the **Phi, Theta,** and **Psi** values and select units for each from the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down menus.



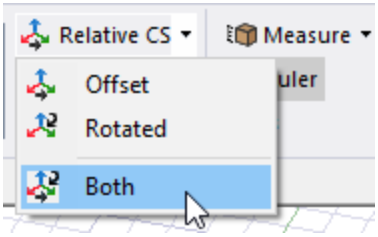
The new relative CS is created. It has the same origin as the previous working CS, but its axes are rotated. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.



Creating an Offset and Rotated Relative CS

To create a new relative CS that is both offset and rotated from an existing CS:

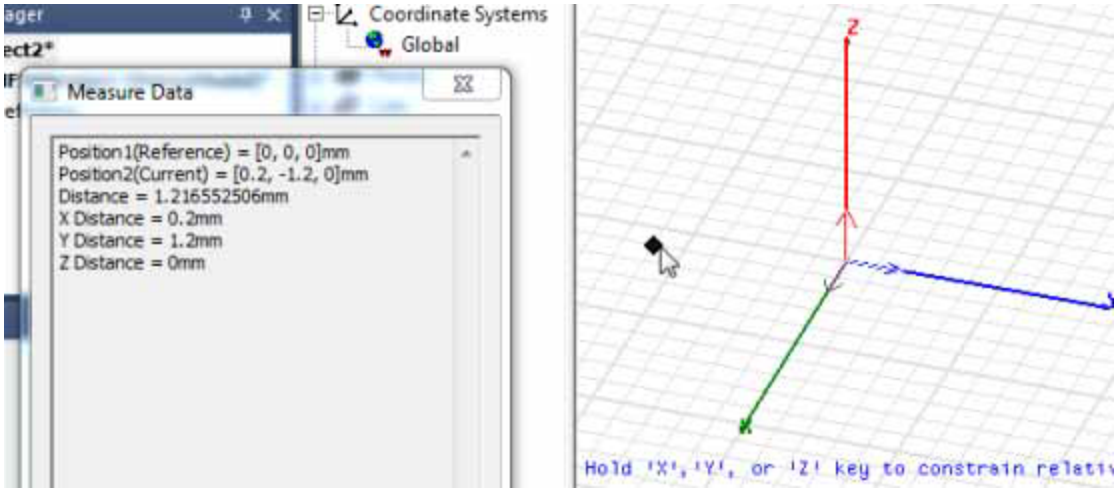
1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler> Coordinate System> Create> Relative CS> Both** or, on the **Draw** ribbon tab, click **Relative CS> Both**:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the origin and axes in one of the following two ways:

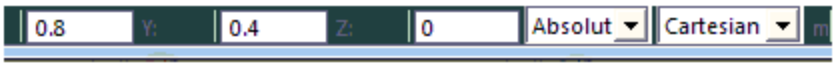
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
  - Specify the Origin, X-axis, and XY plane graphically (click to select), as follows:
    a. Use the cursor to click the Origin point. You can see the coordinate information in the *Measure Data* window.



    To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

    b. In the same manner, click a point to define the X-axis.
    c. Select a third point, one lying on the XY plan that's not on the X-axis, to complete the CS definition.
  - Specify the Origin, X-axis, and XY plane numerically in the Status Bar, as follows:

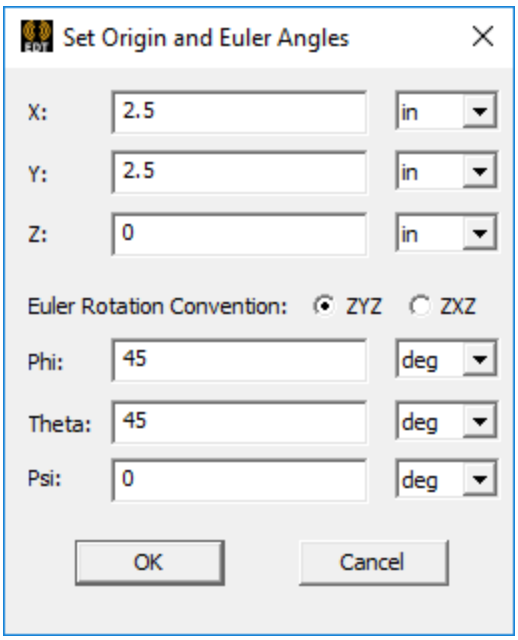


    a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
    b. Select either Relative or Absolute coordinates from the other drop-down menu.
    c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.
    d. In the same manner, type the coordinates of a point on the X-axis and press **Enter**.
    e. Type the coordinates of a third point, one lying on the XY plan that's not on the X-axis, and press **Enter** to complete the CS definition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

- **Euler angle** option selected, or **F4** pressed:
  a. In the *Set Origin and Euler Angles* dialog box, specify the **X, Y,** and **Z** coordinates of the Origin and select units for each from the drop-down menus.
  b. Select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).
  c. >Specify the **Phi, Theta,** and **Psi** values and select units for each from the drop-down menus.

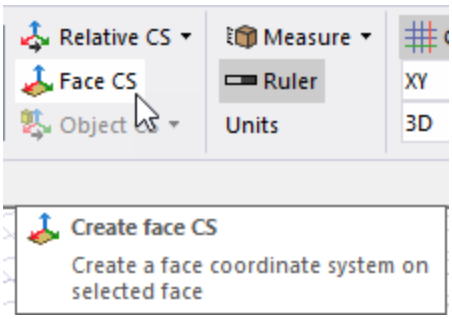


The new relative CS is created. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.

Creating a Face Coordinate System

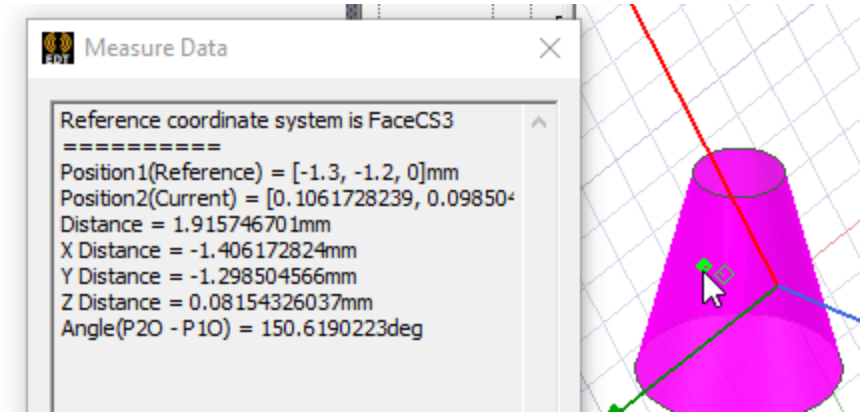
1. Select the object face upon which you want to create the face CS. The CS face can be planar or curved.
2. Click **Modeler> Coordinate System> Create> Face CS** or, on the **Draw** ribbon tab, click the **Create Face CS** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



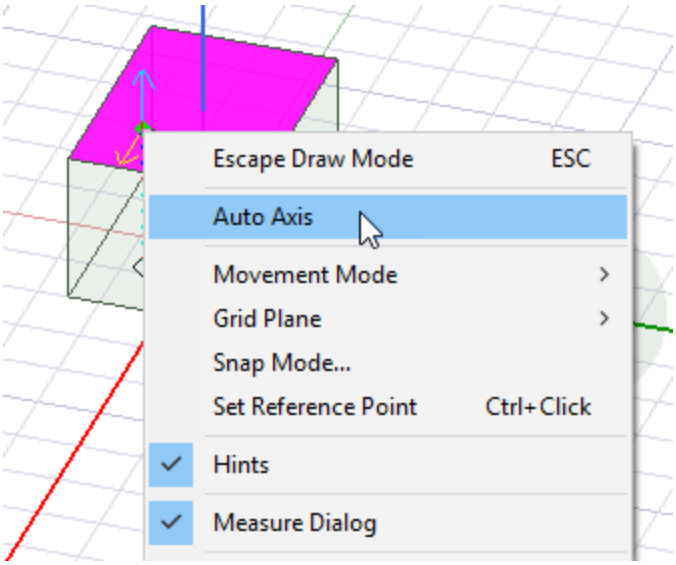
3. Select the Origin in one of the following ways:
   - Click a point on the selected face to define the Origin. If the point is not on the face, you will receive an error message.

     The *Measure Data* window shows the current cursor coordinates. The cursor shape will change to identify grid points, vertices, edge center, face center, quadrant, and arc center points, but you can click any point on the selected face.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes on the Status Bar.
4. Specify the positive X-axis direction in one of the following ways:
   - Click a point on the selected face. If the point is not on the face, you will receive an error message.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX, dY,** and **dZ** text boxes, where **d** is the distance from the previously selected point.

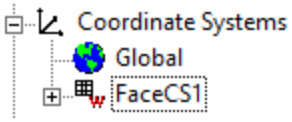
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Right-click in the Modeler window and select **Auto Axis** from the shortcut menu. The modeler selects the X-axis based on the principal curvature of the face or the dimensions of a flat face. The *axial* direction of a curved face or the long dimension of a flat face is selected as the X-axis direction.

You do not need to specify the Y or Z axes. The modeler assumes that the Z-axis is normal to the object face and the Y-axis is automatically calculated to be perpendicular to the XZ plane.

The new face CS is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be referenced to the coordinates of this face CS. Default planes are created on its XY, YZ, and XZ planes.



Only operations listed in the History Tree *before* the face CS's creation will affect the face CS and, in turn, affect objects dependent upon that face CS. A face CS, or any object created on it, is *not* affected by operations that occur after the face CS is created. Also see the Move CS to End command.

For example, suppose you create a box, then a face CS on a face of the box, and then a cylinder on the face CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit> Arrange> Rotate** command, the box will move, but the cylinder will not move (because the rotation operation occurs later in the History Tree).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatically Creating Face Coordinate Systems

You can instruct the modeler to automatically create a new face CS every time you draw on an object's face.

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

   The *Options* dialog box appears.

2. In the tree on the left side of the dialog box, select **3D Modeler> Operation**.
3. In the *Coordinate System* section, select **Automatically switch to face coordinate system**.
4. Click **OK**.

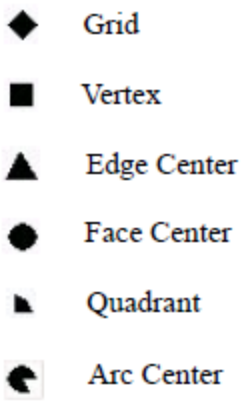
Now, when you select a face, and then click a drawing command, a new face CS will be created on the selected face. The modeler automatically sets the new face CS as the working CS, and the object you draw is oriented according to the new face CS.

> **Note:**
>
> The modeler will not automatically create a new face CS if a face CS has already been assigned to the selected face.

Creating an Object Coordinate System

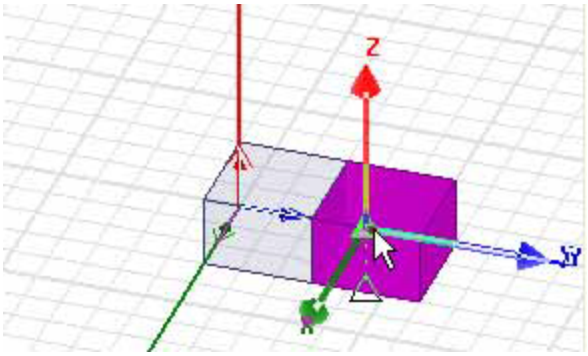
You can create coordinate systems based on any object of solid, sheet, or wire type. The **Modeler> Coordinate System> Create> Object CS** command, and the equivalent ribbon command, is enabled when you select an object. An Object CS can be Offset, Rotated or Both. Executing one of the Object CS commands changes the cursor to the selection marker mode.

As you drag the selection marker over an object, it follows the 3D surfaces of the object, dropping a dashed reference line to a point on the current plane. The cursor changes shape to provide information about the object at the corresponding coordinate:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in this case, the cursor shows Edge Center triangles as valid selection points for a **Modeler> Coordinate Systems> Create Object CS> Offset** command.



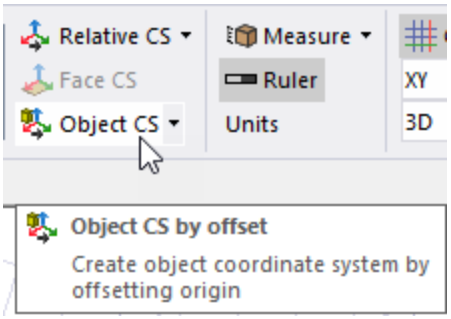
Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS, and in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. Also see the Move CS to End command.

For example, suppose you create a box, then an Object CS on a face of the box, and then a cylinder on the Object CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit> Arrange> Rotate** command, the box will move, but the cylinder will not move because the operation occurs later in the History Tree.

Creating an Offset Object CS

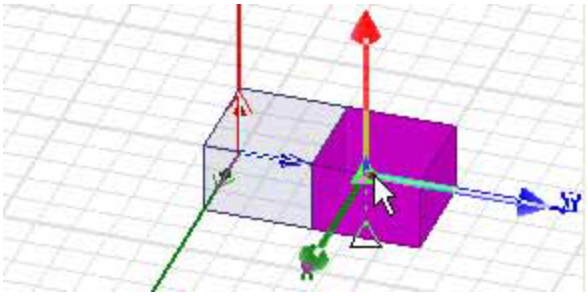
To create an Offset Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.
3. From the menu bar, click **Modeler> Coordinate System> Create> Object> Offset** or, from the **Draw** ribbon tab, click **Object CS**. (You do not have to access the *Object CS* drop-down menu because *Offset CS by offset* is the default type of Object CS):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the Origin for the new CS.
    - You can select any snap point on the object (depending on the current snapping mode options) to select origin of the Object CS. When you hover the mouse over a valid point, a coordinate system preview is shown.

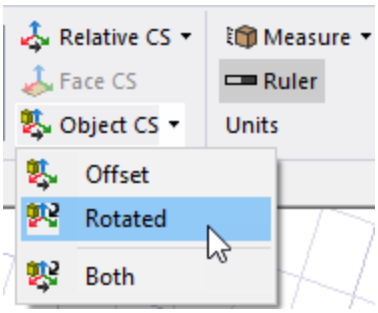


    - The point must be on the selected object.
    - When you select the Origin point, the location is validated, and an appropriate message indicates when a point is invalid. Points where a CS preview appears are always valid.
    - The X-axis {1,0,0}, Y-axis {0,1,0} and Z-axis {0,0,1} are parallel to the corresponding axes of the working CS that was active when you executed the Object CS command, but the Origin has been relocated.
    - The axis coordinates can be later edited through the docked *Properties* window or the *Properties* dialog box.

Creating a Rotated Object CS

To create a Rotated Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Rotated Object CS.
2. Select the object to use as the basis of the new Rotated Object CS.
3. From the menu bar, click **Modeler> Coordinate System> Create> Object> Rotated** or, on the **Draw** ribbon tab, click **Object CS> Rotated**:



You are prompted to *Select X axis*. A preview of X axis is shown for valid selections.

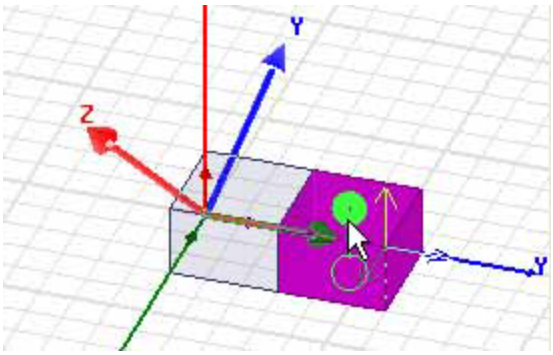
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the X-axis direction for the new CS.
   - The Origin {0,0,0} remains at the location defined for the working CS that was active when you executed the Rotated Object CS command. You can edit the coordinates later through the docked *Properties* window or the *Properties* dialog box.
   - The clicked point must be on the selected object.

   You are prompted to *Select to define XY plane*.

5. For 3D models, click a second point on the selected object to define the XY plane.

   A CS preview displays as the cursor hovers over valid selections.



   In the 2D modeler, you are prompted to select only the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Creating an Object CS that is Both Offset and Rotated

To create an Object CS that is both offset and rotated:

1. Set the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.

   The requirements to define this type of Object CS are a combination of *Offset* and *Rotated* Object CS definitions. You are first prompted to select the Origin (refer to the Creating an Offset Object CS page).

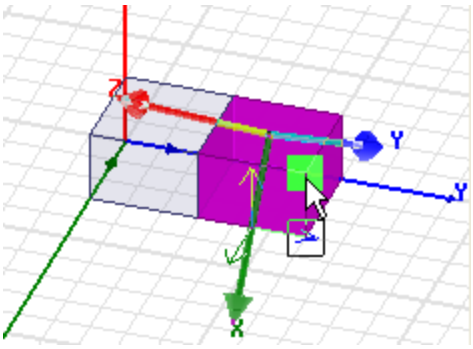
3. Select a point on the object to define the Origin.

   You are then prompted to define the XY plane (refer to the Creating a Rotated Object CS page.

4. For 3D models, select a second point on the object to define the XY plane.

   A CS preview appears when you hover over valid snapping points:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



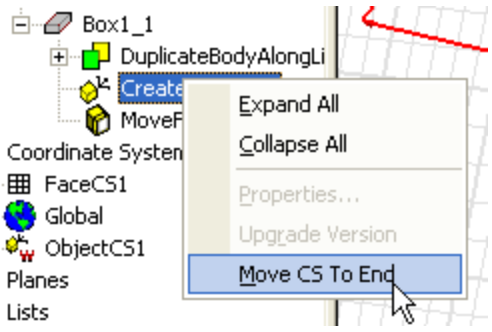
In the 2D modeler, you are asked only to select the origin and the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Move CS to End Command for History Tree

Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS and, in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. It is sometimes useful to have the coordinate system affected after any other operations that might have edited, moved, or rotated the object.

If you have at least one operation after a *CreateFaceCS* or *CreateObjectCS* in the History Tree:

Then, selecting a Face or Object CS enables the **Move CS to End** command in the **Modeler> Coordinate System** submenu and in the shortcut menu that appears when you right-click the operation in the History Tree:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Executing this command moves the selected CreateObject CS to the end position in the History Tree and updates associated items (other CS, object history, any dependent parts, and so on).

Modifying Coordinate Systems

Keep in mind that when you edit a CS, the following will also be affected:

- All objects drawn on the CS.
- All CSs that were defined relative to that CS.
- All objects drawn on a CS that were defined relative to that CS.

There are two ways to modify a coordinate system. You can select the coordinate system in the History Tree and edit its parameters in the docked *Properties* window. This approach does not allow you to change whether the coordinate system is Absolute or Relative. Nor does it allow you to change how you express the coordinates (as Cartesian, Cylindrical, or Spherical).

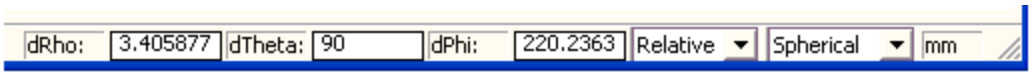
If you want to also modify the whether the coordinate system is Absolute or Relative, and to change how your express the coordinates, do the following:

1. From the menu bar, click **Modeler> Coordinate System> Edit**.

   The *Select Working CS* dialog box appears.

2. Select the CS you want to modify.
3. Click **Select**.

   This action places you in the graphical selection mode for redefining the CS (origin or X-axis) and enables the editable text boxes in the Status Bar at the bottom right area of the Ansys Electronics Desktop application.



4. You can click points to redefine the CS. Alternatively, you can specify the type of coordinate definition (Absolute or Relative) and tab between the coordinate entry text boxes in the Status Bar to numerically define new coordinates.
   - If you selected a Relative CS, follow the directions for creating a relative CS.
   - If you selected a Face CS, follow the directions for creating a face CS.
   - If you selected an Object CS, follow the directions for creating an object CS.
5. Select the type of coordinate system from rightmost drop-down menu in the Status Bar (Cartesian, Cylindrical, or Spherical).

   The values you specify are translated to Cartesian coordinates in the *Properties* of the revised coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

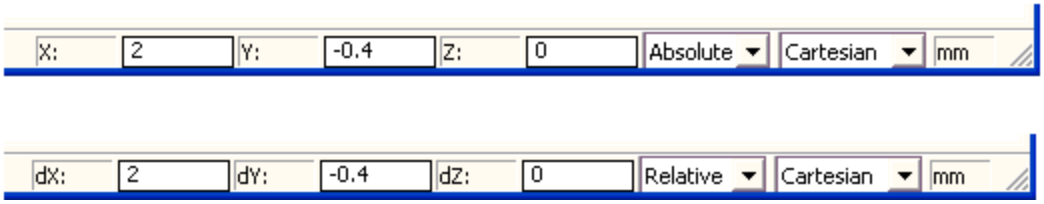
An Object CS can be edited in the same way as it was created. For example, if the CS was created in Offset mode, it will be edited in Offset mode only.

- You can edit the reference CS of an object CS through the docked *Properties* window or the *Properties* dialog box. Doing so will impact object CSs created in *Offset* or *Rotated* mode. It will not impact an object CS created in *Both* (offset and rotated) mode, as the CS fully depends on the object.
- The Origin will be editable when an Object CS was created in *Rotated* mode. Otherwise the Origin coordinates will be read-only text.
- The X Axis and Y Axis will be editable when an Object CS was created in *Offset* mode. Otherwise it will be axis coordinates will be read-only text.
- Either the X Axis or Y axis can be reversed through the docked *Properties* window or the *Properties* dialog box. When the X Axis is reversed, the Y Axis does not change and vice versa.
- When origin or axis properties are defined from snapping points, the point ID is listed in the properties instead of the coordinates. Some examples of snapping point IDs are: *Vertex_10, Face_7 center, Edge_9 midpoint, Edge_8 quadrant,* and *Edge_17 arc center*.

Expressing Cartesian, Cylindrical, or Spherical Coordinates

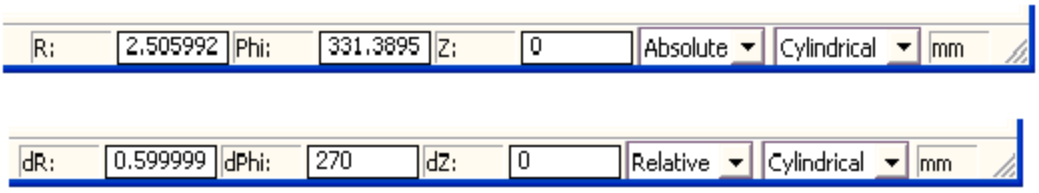
You have choices for expressing coordinates, specifically as Cartesian, Cylindrical, or Spherical by using the editable fields in the Status Bar. In each case, you can also specify whether to enter the coordinates as Absolute or Relative to a reference point. Click the first text box, or press **Tab** to begin entering values or variables. **Tab** to the next text box or **Ctrl+Tab** to the previous one. Press **Enter** when you are done. After you enter values or variables in the text boxes, they are evaluated as Cartesian for the coordinate system *Properties*, regardless of the way they were defined.

- **Cartesian:** Coordinates represent the point's distance from the origin in the x, y, and z directions. The values are specified in the **X, Y,** and **Z** text boxes.

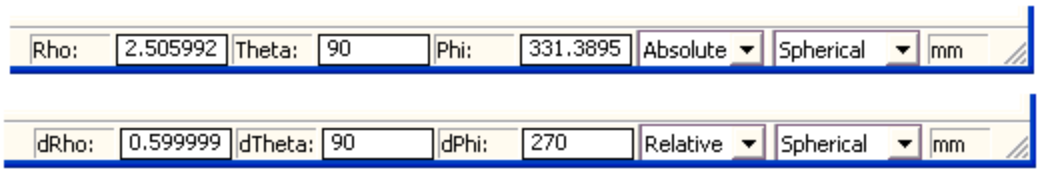


- **Cylindrical:** Specify the point's radius, measured from the origin, is the **R** text box, the angle from the x-axis in the **Theta** text box, and the z-distance from the origin in the **Z** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Spherical:** Specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the x-axis in the **Theta** text box, and the angle from the origin in the z direction in the **Phi** text box.



Deleting Coordinate Systems

1. In the *Coordinate System* branch of the History Tree, click the name of the CS you want to delete.
2. Delete the CS using one of the following methods:
   - Press **Delete**.
   - From the menu bar, click **Edit**> ✕ **Delete**.
   - On the **Desktop** ribbon tab, click ✕ **Delete**.

   The CS will be deleted and, all objects drawn on it will be deleted. Furthermore, any CS that was dependent upon the deleted CS will be deleted, and any objects that were drawn on the dependent CS will also be deleted.

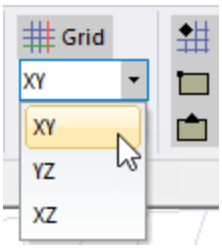
Setting the Drawing Plane

The *Drawing plane* options are available from a drop-down menu on the *Draw* ribbon tab and from the *Grid Plane* submenu of the *Modeler* menu. The XY, YZ, and XZ drawing planes correspond to the active coordinate system, not the global planes (unless *Global* is the currently active CS).

To set the drawing plane, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** ribbon tab, select **XY, YZ,** or **XZ** from the **Drawing plane** drop-down menu:



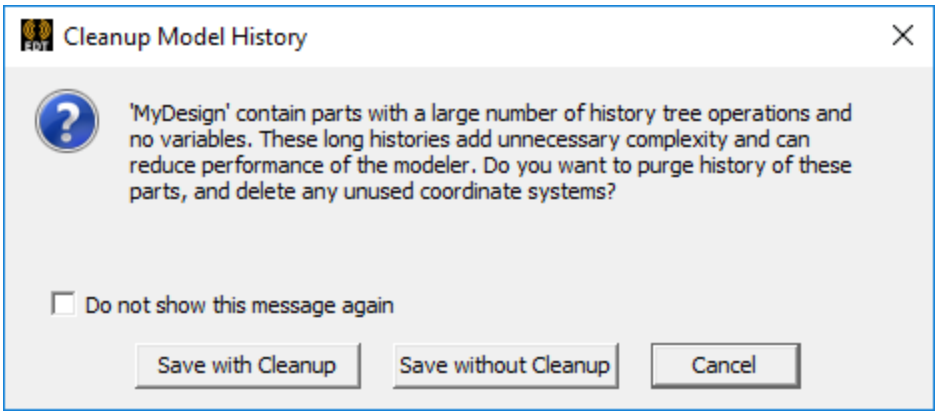
- From the menu bar, click **Modeler> Grid Plane>** and select one of the planes **XY, YZ,** or **XZ**.

Cleaning Up Model History

Ansys Electronics Desktop can clean up history tree operations in order to improve performance.

When saving your project, you will be prompted to clean up your model's history if some parts meet two conditions:

- No variables assigned to the part
- A history greater than a specified number of actions



The number of actions that triggers this prompt can be set in 3D Modeler Operations Options.

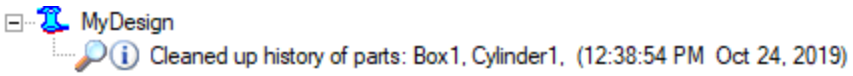
When promoted, you can choose **Do not show this message again** to hide the prompt for the active project.

To clean up the model history manually:

- Select **Modeler** > **Cleanup Model History**.

After the model has been cleaned up, you can view cleaned up parts in the **Messages** window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

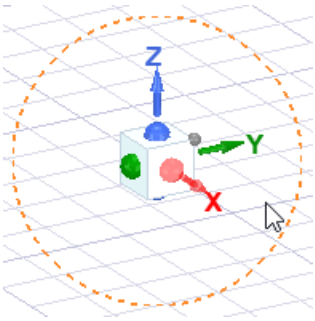
13 - Modifying the 3D Model View

You can modify the view of objects in the 3D **Modeler** window without changing their actual rotation or position within the coordinate system. You vary the viewing direction and the relative location of the "camera" (that is, the viewpoint) relative to the model geometry. The tools allow you to zoom, pan, or rotate the view orientation and to quickly switch to predefined or user-defined view orientations. Additionally, you can choose different object visualization options, show/hide objects, and customize the window background.

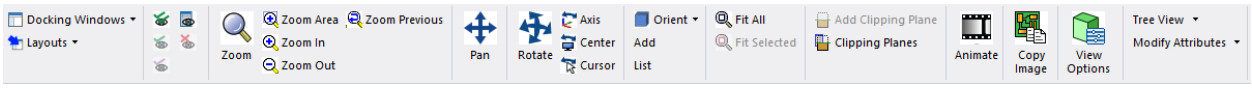
What do you want to do?

Change the View	Show or Hide Design Objects
Change Object Visualization	Change the Background

In addition to various menu bar, ribbon, and shortcut menu commands, you can manipulate the view orientation of 3D models and 3D plots using the Orientation Gadget:



Finally, you can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



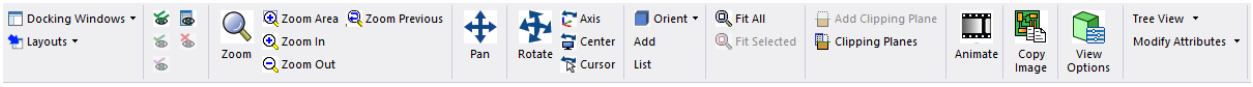
Change the View

- Using the Orientation Gadget
- Changing the Model View with Alt+Double-Click Areas in the Modeler Window
- Apply an Orientation to the Current View

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the View Options
- Rotate the View
- Pan the View
- Zoom In or Out
- Zoom to Selected Excitation
- Zoom In or Out Using a Mousewheel
- Fit Contents in the View Window
- Use Clip Planes

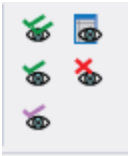
You can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



Show or Hide Design Objects

- Show or Hide Objects
- Showing Only Selected Objects in All or Active Views
- Show or Hide Boundaries or Excitations

You can also access the Show or Hide features by using the following icons on the **View, Draw,** or **Model** ribbon tabs:



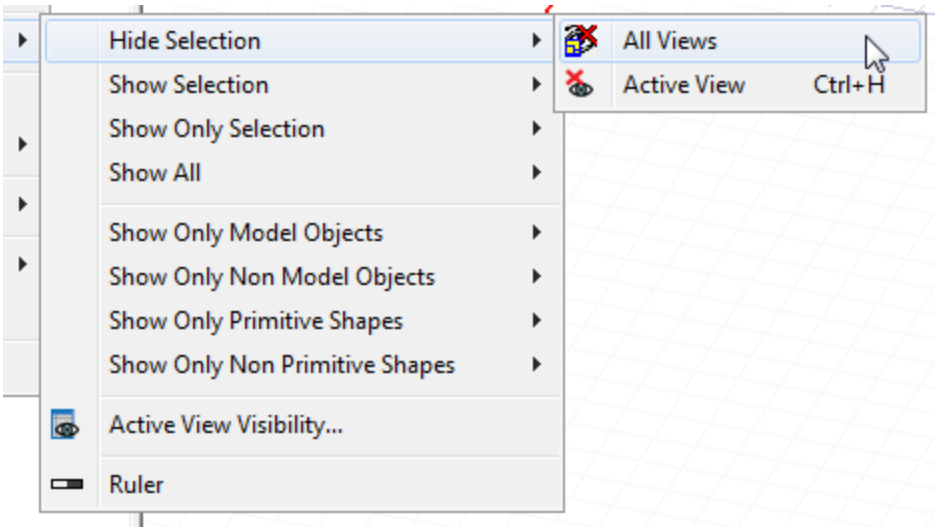
Hiding Objects from View

To hide selected objects.

1. Select the object you want to hide from view. This enables the menu commands and **View, Draw,** and **Model** ribbon tabs icons for hiding objects.
2. Click **View** > **Visibility** > **Hide Selection** and one of the following commands:
   - **All Views** to hide the selected object in every open view window.
   - **Active View** to hide the selected object in the active view window. You can also use **Ctrl+H** to hide a selected object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right-click in the **Modeler** window for a short-cut menu and use the **View** commands for **Hide Selection** or **Show Only Selection**.



You can also use the **Hide selected objects** icon in the **View, Draw,** or **Model** ribbon tabs to hide selected objects in the active view.

The objects you selected are hidden.

If there are many objects, you may find it easier to hide objects using the Active View Visibility dialog box. You can also choose to Show only selected objects in all or active views, effectively hiding all unselected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

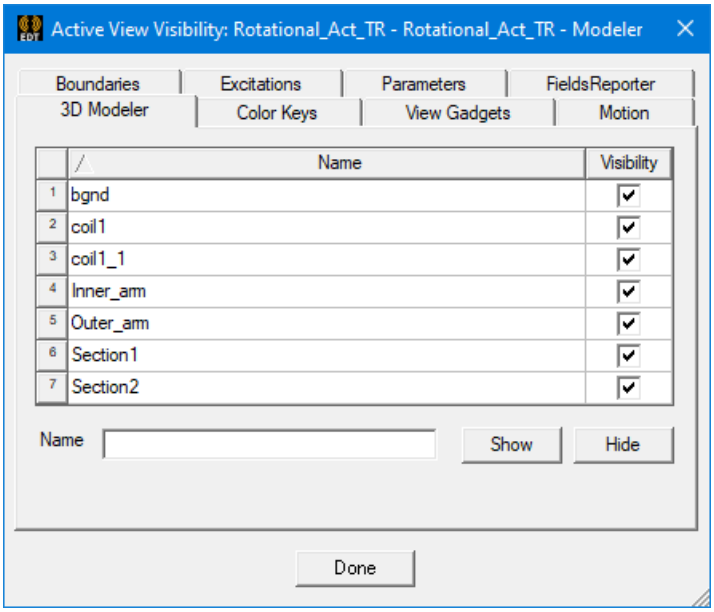
> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Showing Objects

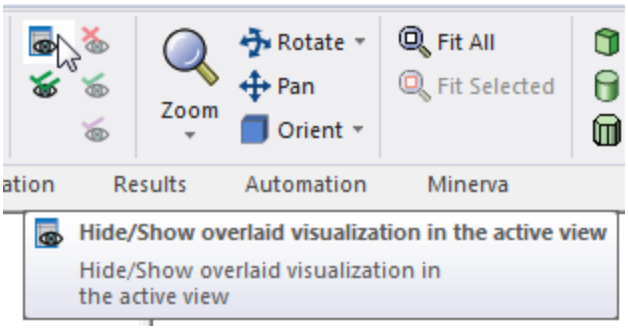
You can show or hide objects in several ways, using the **Active View Visibility** dialog box for individual objects, or **Visibility** menu selections for All Objects, Only Selection, Model Objects, Non-model Objects, Primitive Shapes, or Non-primitive shapes.

The **Active View Visibility** dialog box contains tabs for 3D Modeler, Color Keys, Array Setup, Boundaries, Excitations, and Fields Reporter. Select the tab for the objects you want to show or hide, and select or deselect **Visibility**.

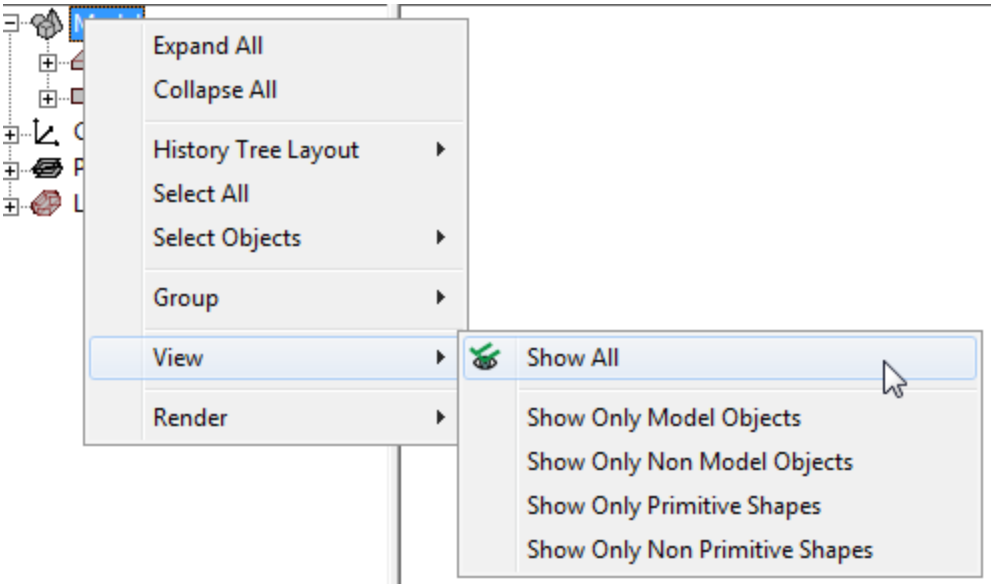


You can access the dialog box from the menu bar (**View** > **Visibility** > **Active View Visibility...**) or by clicking the Hide/Show visualization icons in the **View, Draw,** or **Model** Ribbon tabs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Visibility** menus are available on the **View** menu and as shortcut menus in the *Modeler* window and in the History Tree when you right-click the **Model** icon:



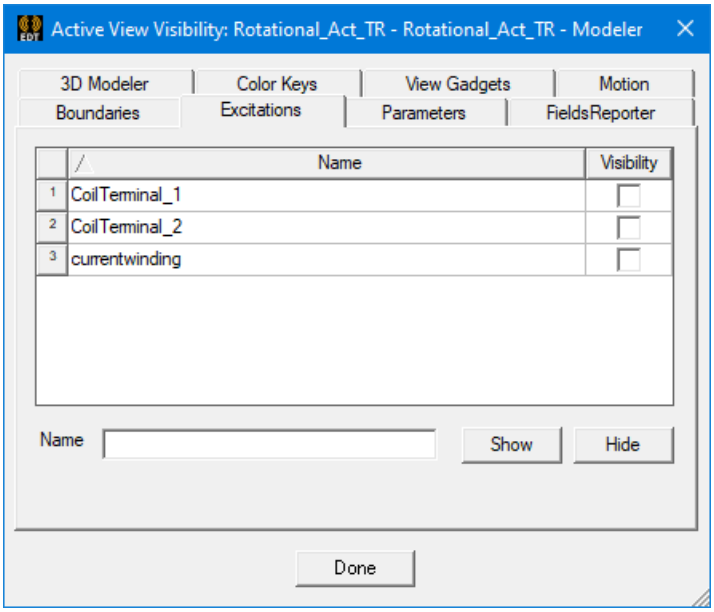
To show *one or more objects* that are currently hidden by using the *Active View Visibility* dialog box:

1. Click **View** > **Visibility** > **Active View Visibility...**, or click the visibility icons on the **View, Draw,** or **Model** ribbon tabs.

   The **Active View Visibility** dialog box appears.

2. Select the tab for the objects you want to show or hide. The dialog box contains tabs for 3D Modeler objects: Color Key objects, Boundaries, Excitations, and Fields Reporter objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
    - For designs with large numbers of objects, you can resize the dialog box for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

    The objects you select and designate as Visible (by selecting the property or using Show) reappear.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

To show *all objects* that are currently hidden by using the **Visibility** menu:

1. Click **View** > **Visibility** > **Show All** or use the shortcut menu in the Modeler window or History Tree and one of the following commands:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **All Views** to show all objects in every open view window
- **Active Views** to show all objects in the active view window.

The selected objects reappear.

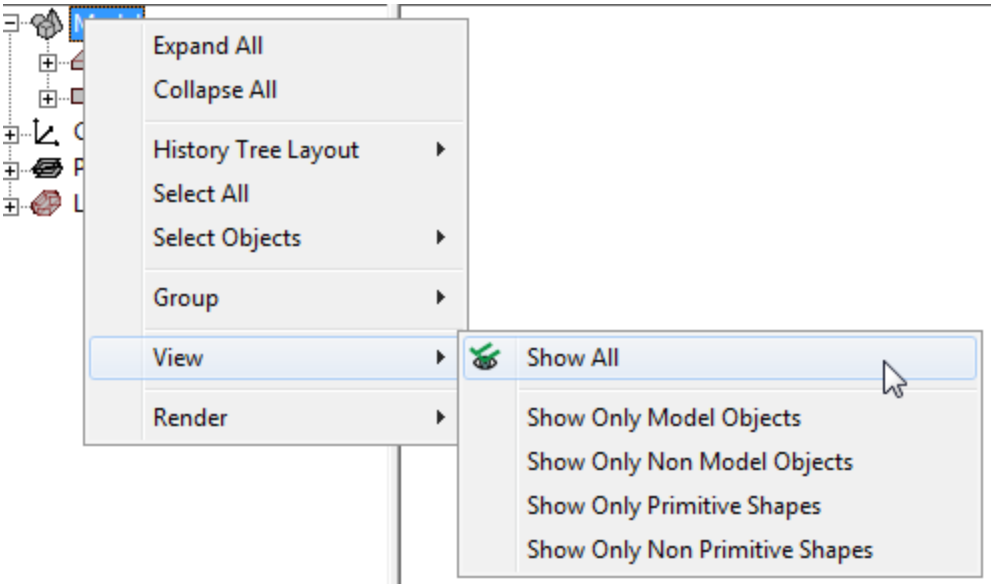
To show *selected objects* that are currently hidden:

1. Select the object. Hidden items are selected once the node corresponding to them is clicked in the history tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window.

   You can also use the **View, Draw,** or **Model** ribbon tab icons to **Show selected objects in active views**. Or, open the **Active View Visibility** dialog box.

To show one or more *3D components* that are currently hidden:

1. In the History Tree, select one or more 3D components. This activates the **Hide/Show** icons on the **View,Draw,** and **Model** ribbon tabs and the **View** menu selections.
2. You can right-click to display the shortcut menu and select **View** > **Show All**.



Object visibility is saved with the project.

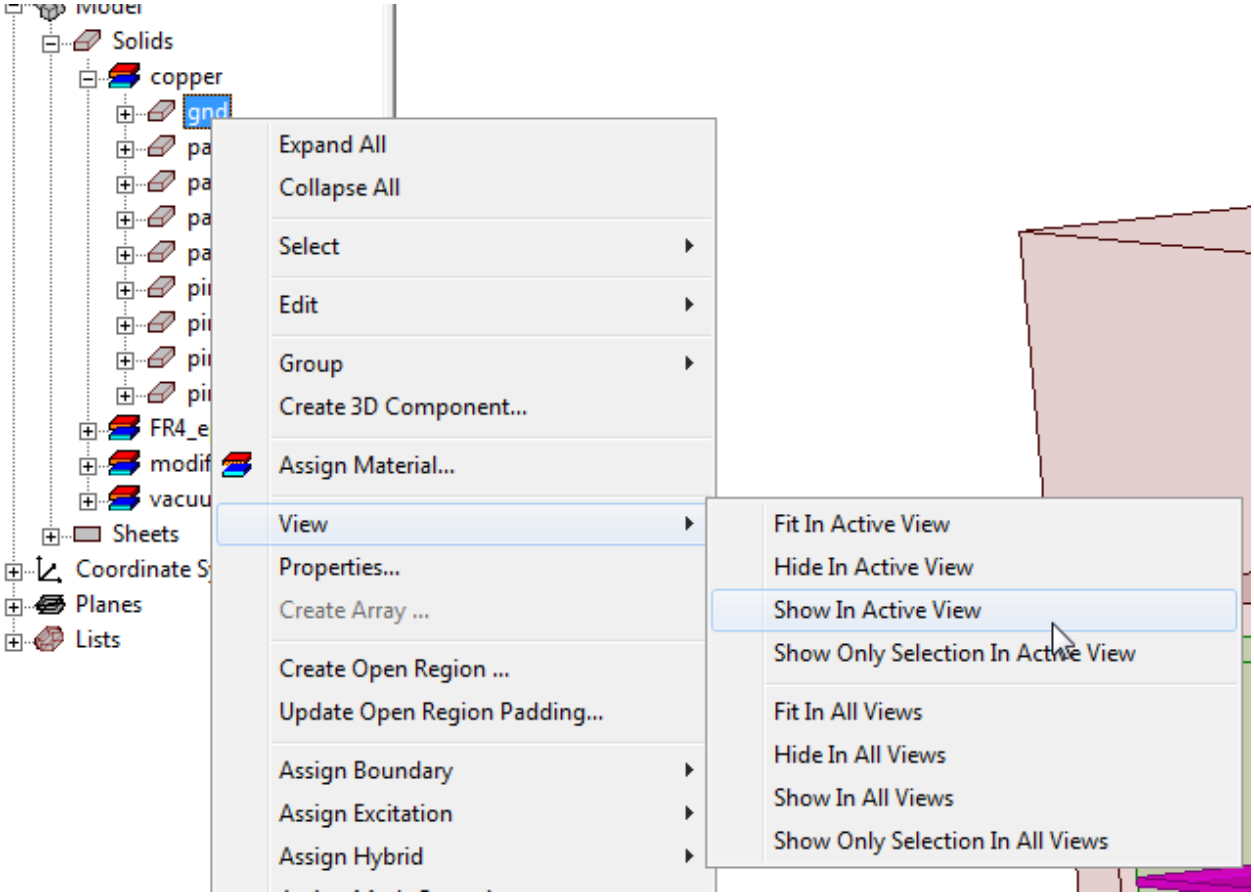
Showing Only Selected Objects in All or Active Views

To show only *selected objects*:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

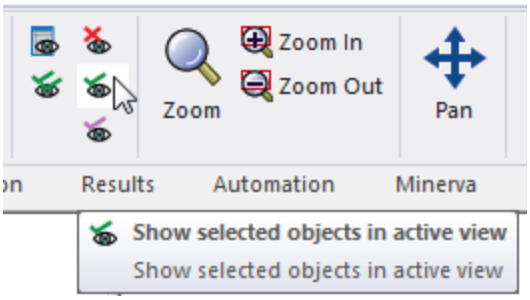
1. Select the object from the Project Manager or History Tree. Hidden items are selected once the node corresponding to them is clicked in the History Tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window

   You can also right-click an object in the History Tree and use the shortcut menu:
3. You can right-click to display the shortcut menu and select **View** > **Show in Active View, Show in All Views,** or **Show Only Selection In Active View**.



You can also use the **View/Draw/Model** ribbon icons to **Show selected objects in all views** and **Show selected objects in active views**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selected objects reappear.

Showing and Hiding Boundaries and Excitations

You can choose to show or hide a boundary or excitation's geometry, name, or vectors, in the active view window or in all view windows.

What do you want to do?

Show or hide a boundary or excitation in the active view window.

Show or hide a boundary or excitation in every view window.

Showing and Hiding Boundaries and Excitations in the Active View Window

1. On the **View** menu, click **Visibility>** **Active View Visibility**.

   The *Active View Visibility* dialog box appears.

2. Select the tab for the objects you want to show or hide. The dialog box contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
4. Click the **Boundaries** tab if you want to show or hide boundaries.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Excitations** tab if you want to show or hide excitations.

- For designs with large numbers of objects, you can resize the dialog for easier selection.
- By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
- You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

The objects you select and designate as Visible (by selecting the property or using Show) appear.

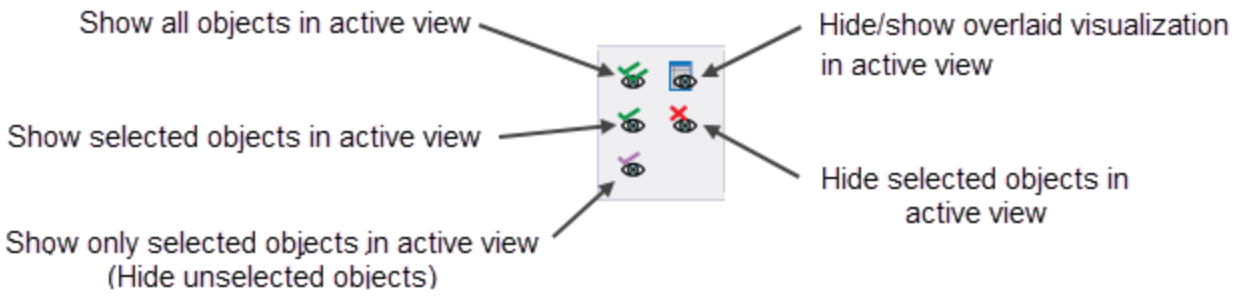
5. Clear the **Visibility** selection of boundaries or excitations that you want to hide from view.

   The boundary or excitation will only be visible in the active view window if it is selected.

6. Select the **Visibility** option for boundaries or excitations that you want to show in the active view window.

   The boundary or excitation will be visible in the active view window when it is selected or when it is not selected.

   You can also use the show/hide icons on the **View, Draw,** and **Model** ribbon tabs:



Showing and Hiding Boundaries and Excitations in Every View Window

1. Click **Mechanical> Boundaries> Visualization** if you want to show or hide boundaries. Or:

   Click **Mechanical> Excitations> Visualization** if you want to show or hide excitations.

2. Clear the **View Geometry**, **View Name**, or **View Vector** option for boundaries and excitations that you want to hide from view. Select the options you want to show.

   The options affect all view windows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Change Object Visualization

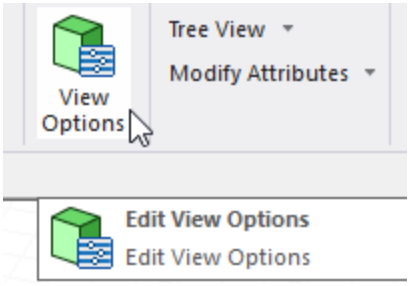
- Render Objects as Wireframes, Flat-Shaded, or Smooth-Shaded Solids
- Set the Enhanced Display of Material Color and Transparency
- Set the Surface Visualization
- Modify the View Orientation
- Modify the Lighting
- Set the Projection View

Change the Background

- Setting the Background Color
- Modify the appearance of the Coordinate System Axes
- Modify the appearance of the Grid

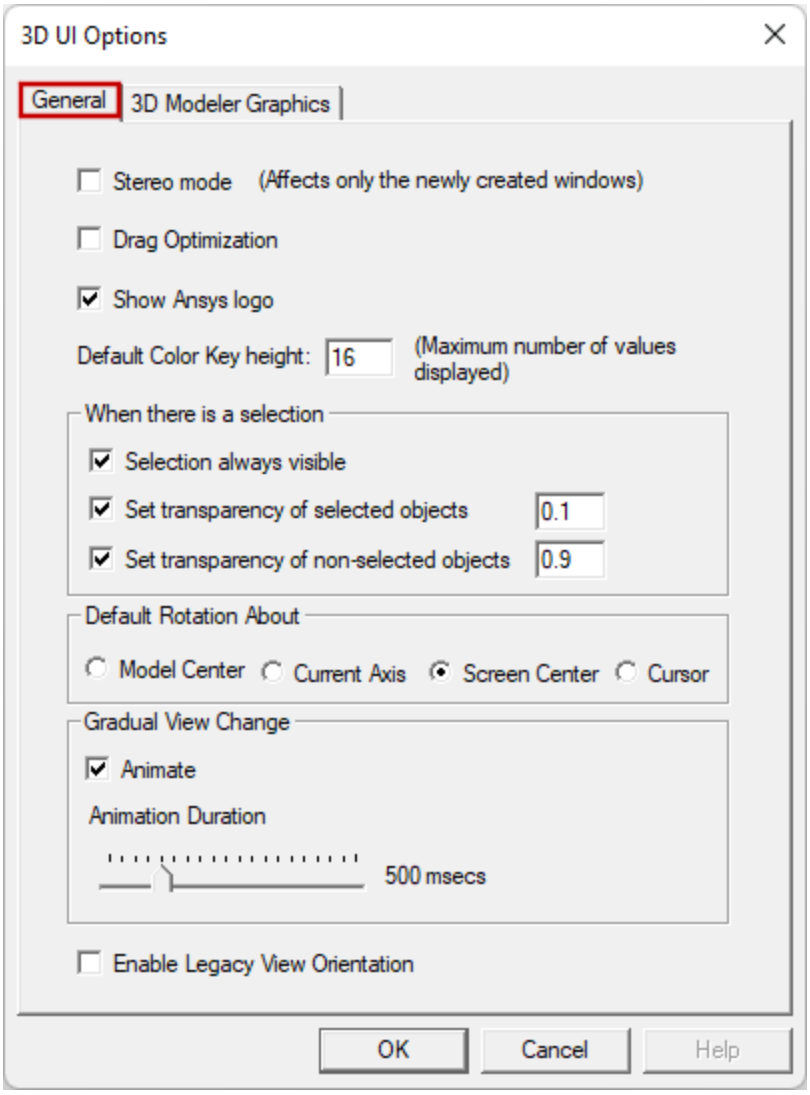
View Options: 3D UI Options

To access View Options, use the **View** > **Options** command, or click **View Options** on the **View** ribbon.



The **3D UI Options** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **General** tab lets you set defaults for the following view options:

- Stereo Mode (default, disabled) – This option is only applicable to systems with support for stereographic viewing for greater 3D realism.
- Drag Optimization (default, disabled)
- Show Ansys logo (default, enabled)
- Default Color Key height (default = 16)

When there is a selection option:

- Selection always visible (default, enabled)
- Set transparency of selected objects
- Set transparency of non-selected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Rotation About options:

- Model Center
- Current Axis
- Screen Center (default)
- Cursor

  When you select one of these as the default, the **View** > **Rotate** selection menu changes to show that the **Alt+drag** combination attaches to your selection.

Gradual View Change settings:

- Animate
- Animation Duration

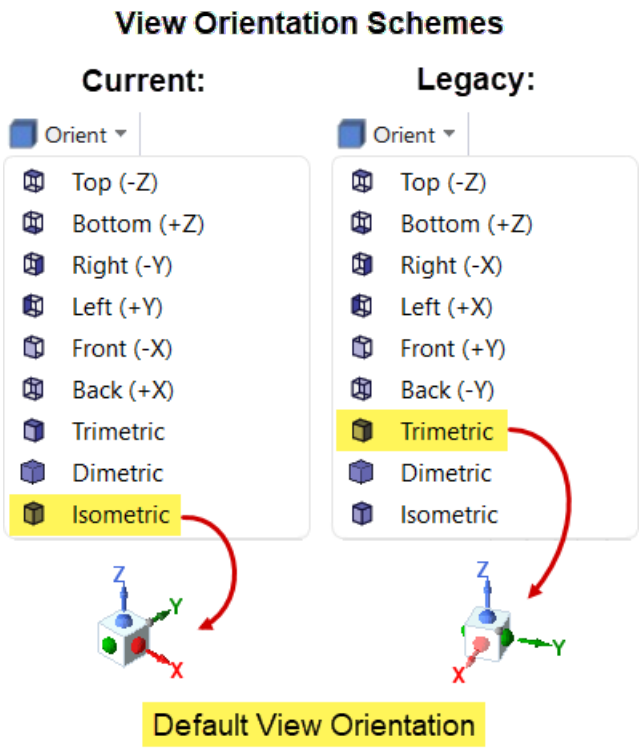
  For operations such as Fit Selection, Fit All, and Zoom, these settings determine whether Electronics Desktop animates the transition between images and how smoothly it does so.

Enable Legacy View Orientations option:

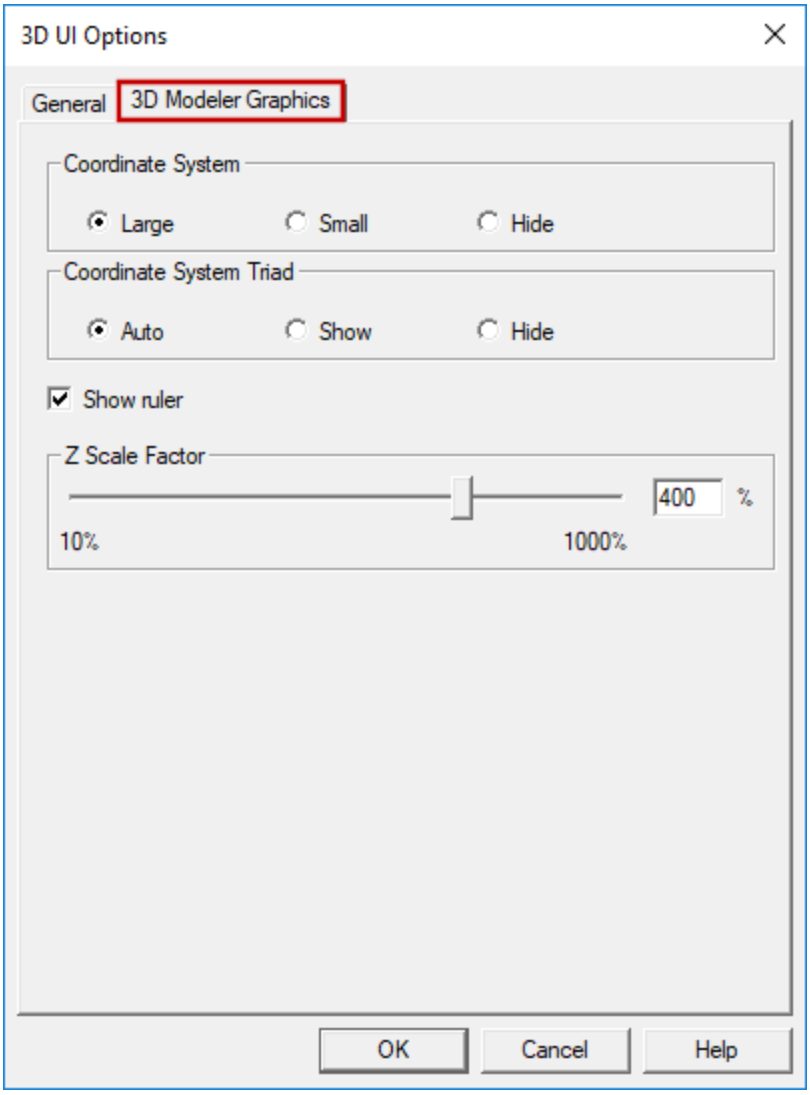
- When selected, the view orientations scheme and the axis directions for named views follow the legacy conventions (software version 2023 R2 or older):
- When cleared, the view orientations scheme and the axis directions for named views follow the current view orientation scheme

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following view is a comparison of the two options:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **3D Modeler Graphics** tab lets you set the following options which are invoked for the next Project and design created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

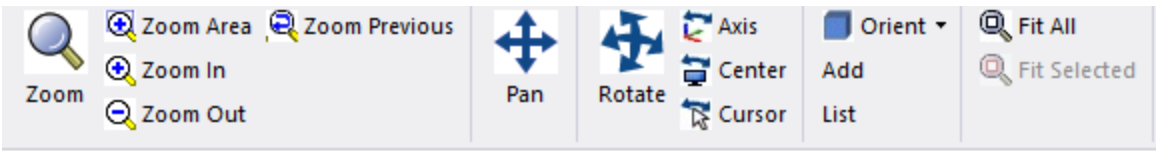
Settings	Description of Choices
Coordinate System	• **Large** (extends to the window edges) • **Small** (centered, not extending toward edges with smaller arrows) • **Hide** (do not display)
Coordinate System Triad (A secondary, smaller coordinate system appearing near the lower right corner of the Modeler window)	• **Auto** • **Show** • **Hide**
Show Ruler (To indicate the scale of the model, based on current units. The ruler appears at the lower center of the Modeler window)	Select to enable; clear to disable
Note: You can also toggle the ruler visibility by clicking **Ruler** on the **Draw** ribbon tab or by right-clicking in the *Modeler* window and choosing **View** > **Ruler** from the shortcut menu.	
Z Scale Factor	The default is 400%. You can use the slider or the text field to specify a different value.

Panning the View

To move (pan) the view:

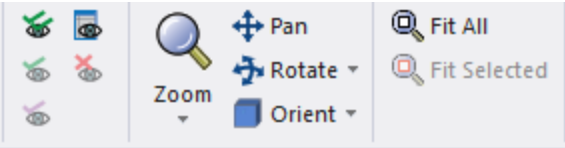
1. Click **View** > **Interaction** > **Pan** or click the **Pan** icon on the **View, Draw,** or **Model** ribbon tab.

   View ribbon tab:



   Draw or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click and drag the mouse in the direction you want to pan the view.

   The view will pan until you release the left mouse button.

3. To exit **Pan** mode, click **Pan** on the **View** menu again or press **Esc**.

> **Tip:**
>
> Alternatively, pan the view using one of the following methods:
>
> - Hold down **Ctrl** as you click and drag using the **middle mouse button**.
> - If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy combination **Shift** + **drag** (with the left mouse button) to pan the view.
> - Press **Ctrl** + ***arrow keys*** to pan vertically or horizontally. The movement is 10 screen pixels per press of the arrow key.
> - In addition to the menu bar and ribbon tabs, you can access *Pan* from a shortcut menu. Right-click in the Modeler window and click **View** > **Pan**. Click the command again or press **Esc** to terminate panning.

Zooming in and Out

You can magnify (zoom in) or shrink (zoom out) the contents in the view window using the following methods:

- Hot Keys (that work without the mouse)
- Hot Keys (used in conjunction with clicking and dragging the mouse)
- **Zoom In** and **Zoom Out** commands, which produce 5% zoom increments
- **Zoom Previous** to undo recent zoom commands, as described below
- Zoom into a Rectangular Area
- Zoom in or out Using a Mouse Wheel
- Zoom to a Selected Boundary or Excitation

> **Note:** Some international keyboards may not support the following hotkeys. If the Shift key is required for a character, such as the plus sign (+), the hotkey may not function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To zoom in using hotkeys:

- Press **Ctrl** + "**+**"

  If using the "=\+" key (not the "+" key on the numeric keypad) do **not** press the *Shift* key.

  The view zooms in 5 percent.

To zoom out using hotkeys:

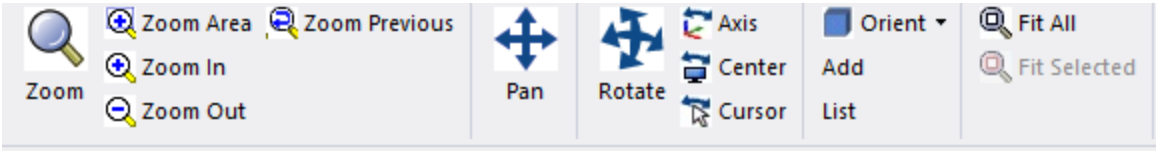
- Press **Ctrl** + "**-**"

  The view zooms out 5 percent.

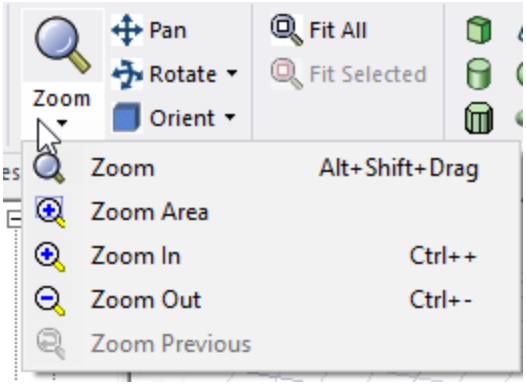
To zoom by dragging the mouse.

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom** commands.



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for Zoom commands:



2. To zoom in, drag the mouse toward the top of the view window. The objects in view expand as you drag.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the Zoom icon, or press **Esc**.

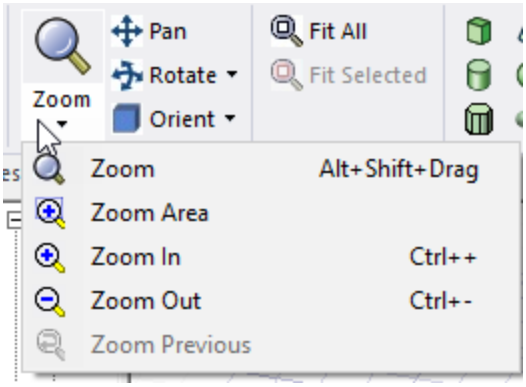
Zoom In and Zoom Out Commands

To use **Zoom In**, **Zoom Out** to zoom in or out in 5% increments:

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom In** or **Zoom Out** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom In** and **Zoom Out**.

   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom In** and **Zoom Out**:



2. Click **Zoom In** or **Zoom Out** for a 5% increment zoom in or out respectively.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

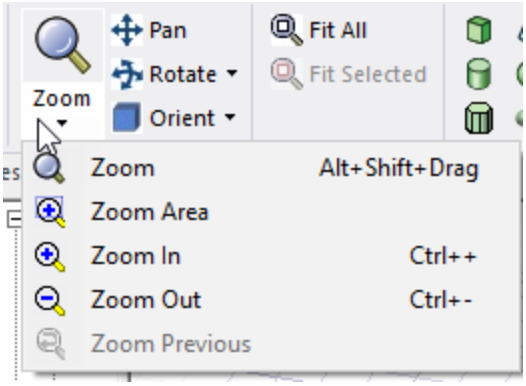
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Zoom Previous Command

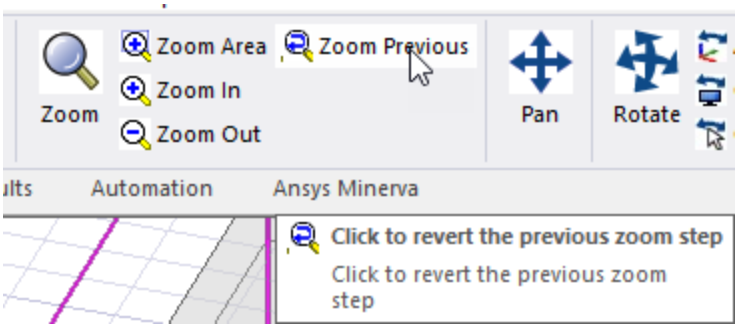
The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.



It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

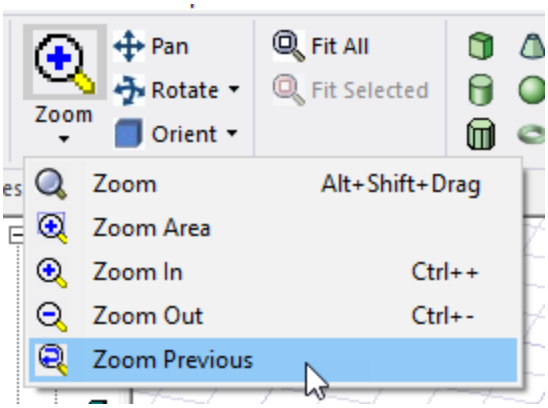
1. Click **View** > **Interaction** > **Zoom Previous**, click **View** > **Zoom Previous** or Zoom Out on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with .



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom Previous**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Zoom Previous** undoes the last executed Zoom command, and then, if others exist, can be used to undo them in reversed order of execution.

   When zooming on a view of model objects the absolute size of the model does not change.

   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Tip:

Alternatively, zoom in or out using one of the following methods:

- Hold down **Shift** as you click and drag using the **middle mouse button**. Moving the cursor upward zooms in and moving it downward zooms out.
- If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy mouse-button and hotkey for zooming. Hold down **Alt** + **Shift** as you click and drag (using the left mouse button).
- Roll the mouse wheel. Rolling the top of the wheel forward (away from you) moves the model view further away from you (zooming out). Conversely, rolling the top of the wheel rearward (towards you) zooms in.
- Right-click in the Modeler window and then click **View** > **Zoom** on the shortcut menu. Click and drag using the left mouse button to zoom in or out. Repeat the command or press **Esc** to cancel zooming.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

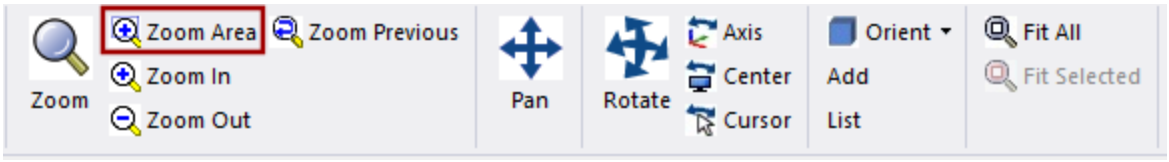
If zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

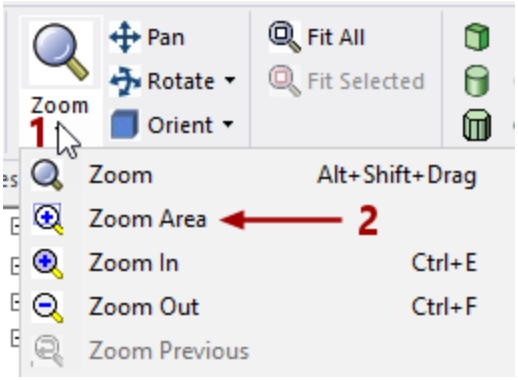
Zooming into a Rectangular Area

Zoom to expand a specified rectangular area to fill the view window, as follows:

1. Use one of the following methods to begin the *Zoom Area* mode:
   - Using the menu bar, click **View** > **Interaction** > **Zoom Area**.
   - From the **View** ribbon tab, click **Zoom Area**:



   - From the **Draw** or **Model** ribbon tab, click **Zoom** > **Zoom Area**:



   - Right-click in the Modeler window and choose **View** > **Zoom Area** from the shortcut menu.
2. Click and drag the mouse to draw a rectangle enclosing the area that you want to zoom into.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you release the mouse button, the rectangular area will come close to filling the display area of the window.

The absolute size of the zoomed geometry does not change, only the scaling of the model view is affected.

3. To end the **Zoom Area** mode, click the command again or press **Esc**.

   The rotate and zoom drop-down menu icons on the ribbon update dynamically to indicate the current rotate/zoom mode. These ribbon buttons are also "sticky." That is, the last active rotate or zoom mode becomes the button default until a different command is chosen or the application is restarted. Default commands are restored the next time you launch the Ansys Electronics Desktop program.

   For 2D or 3D reports, axis labels and tick marks are adjusted after the zoom operation is complete.

Zooming in or out Using a Mouse Wheel

You can zoom the active model view in and out by rotating the mouse wheel. This action works in the Modeler window, for 2D and 3D plots, and more. The cursor location is the center point whenever using the mouse wheel for zooming, and this point remains fixed. The grid and ruler, when visible, are adjusted dynamically as you zoom in and out.

Rotating the wheel forward, away from you, moves the model away from you (zooming out). Rotating the wheel backward, towards you, pulls the model closer (zooming in).

> **Note:**
>
> If Zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

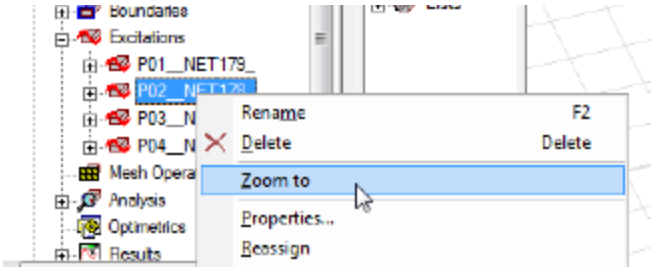
Zoom to a Selected Boundary or Excitation

Zoom to is an option that can enlarge the view of a boundary or excitation. This command can be very useful for locating small items with boundaries or excitations assigned or to inspect any problem areas.

You can use the **Zoom to** command as follows.

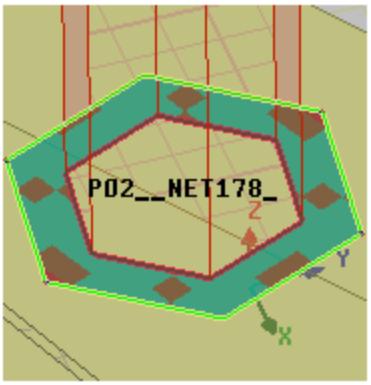
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If needed, expand the *Boundary* or *Excitation* branch in the Project Manager to see the previously assigned conditions.
2. Right-click the boundary or excitation of interest to highlight it in the model view and to access the shortcut menu, as shown below:



3. Select **Zoom to**.

   Notice that the command instantly increases magnification, filling the screen with the object(s) or face(s) to which the boundary or excitation is assigned, as shown in the following figure:

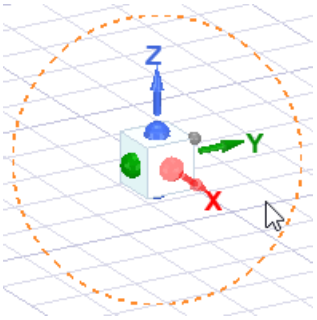


Modifying the View Orientation

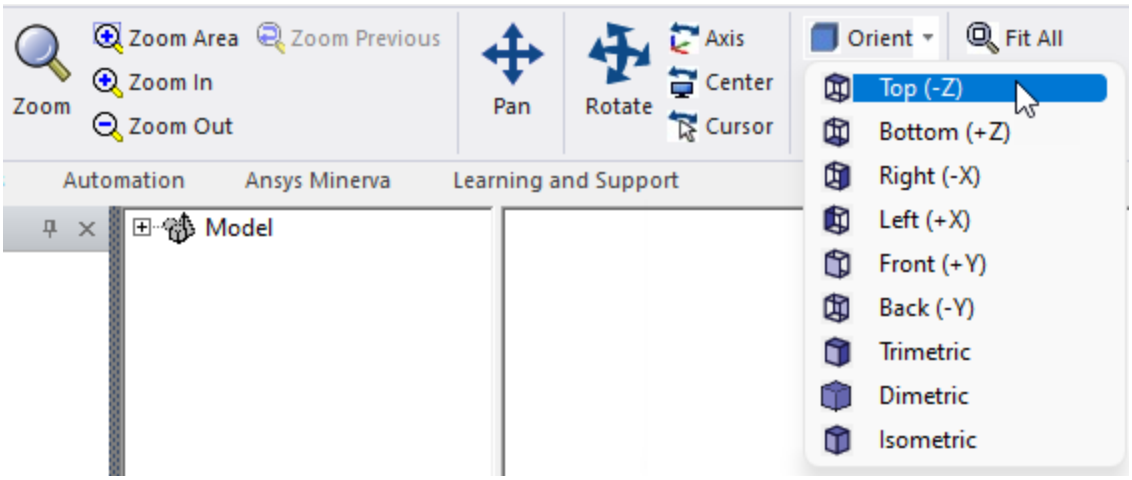
There are several places you can access commands to change the view orientation in a *Modeler* or *3D Plot* window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the View Orientation Gadget:



- Select the **Orient** drop-down Menu on the **View** ribbon tab and choose one of the predefined view orientations:



The **Orient** drop-down menu can also be located on the **Draw** and **Model** ribbon tabs.

The sign and letter in parentheses that follows each standard view indicates the viewing direction, not the model side. For example, consider a simple box aligned with the global planes. The right side of the box is the +X side, but the viewing direction is -X (you look leftward to see the right side of an object).

- Right-click in the Modeler window and, from the shortcut menu that appears, point to **View** > **Apply Orientation**, and select the desired viewing direction from the submenu. The choices include those shown in the preceding image plus any custom views you have defined (see the next bullet).
- From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

  A dialog box with orientation settings appears. It includes a table with the names of nine basic orientations (Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric), any additional orientations that you have added, and a section for managing orientations (adding/removing orientations or setting the default view).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The table includes columns that show the viewing angles relative to the global axes and the equivalent vector components. These vector components are listed based on the global coordinate system and any applicable local coordinate systems.

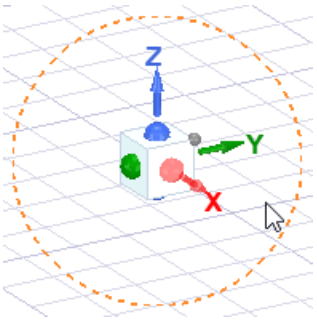
1. Select a default orientation or create a new orientation.
2. Click **Apply to View** for the selected view to appear in the window.
3. Optionally, click the **Reset View Orientation** button to restore the selected view to the original angles.
4. Click **Make Default** if you want the selected viewing direction to be the initial viewing direction when a **3D Modeler** window is opened, either in the current project or future projects.
5. Click **Close** to dismiss the dialog box.

   The orientation you set will be saved with the design. New orientations assigned to other designs after this point will not affect this orientation.

Apply an Orientation to the Current View

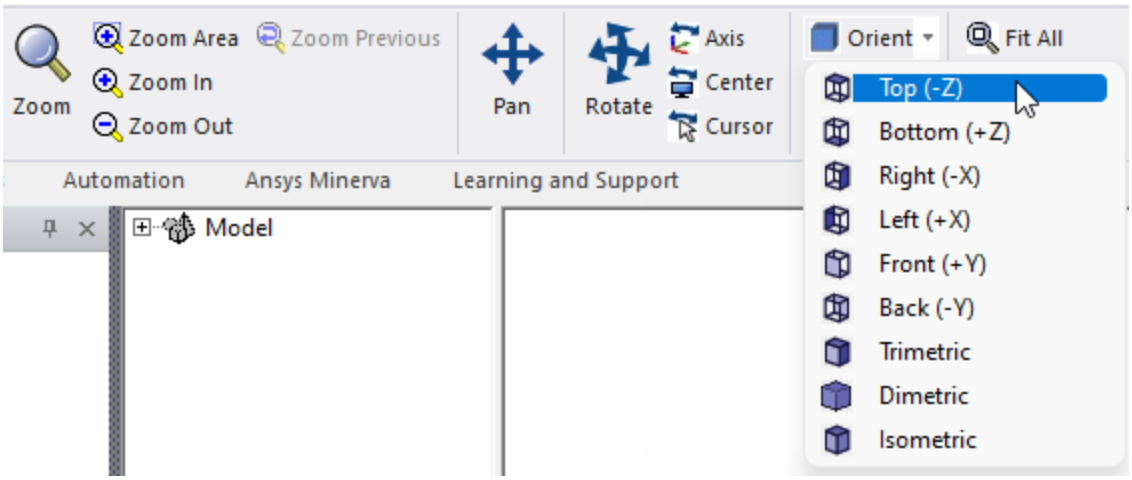
Apply view orientation settings using any one of several available methods:

- Use the Orientation Gadget:

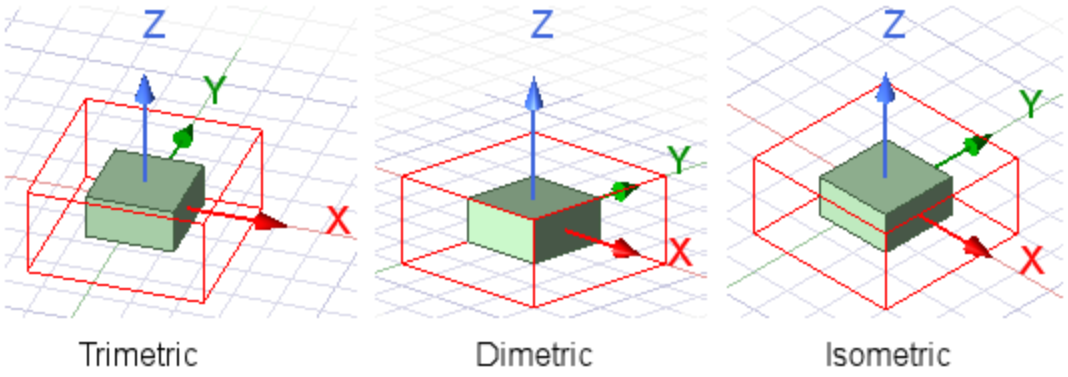


- Use the commands in the **Orient** drop-down menu located on the **View, Draw,** or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

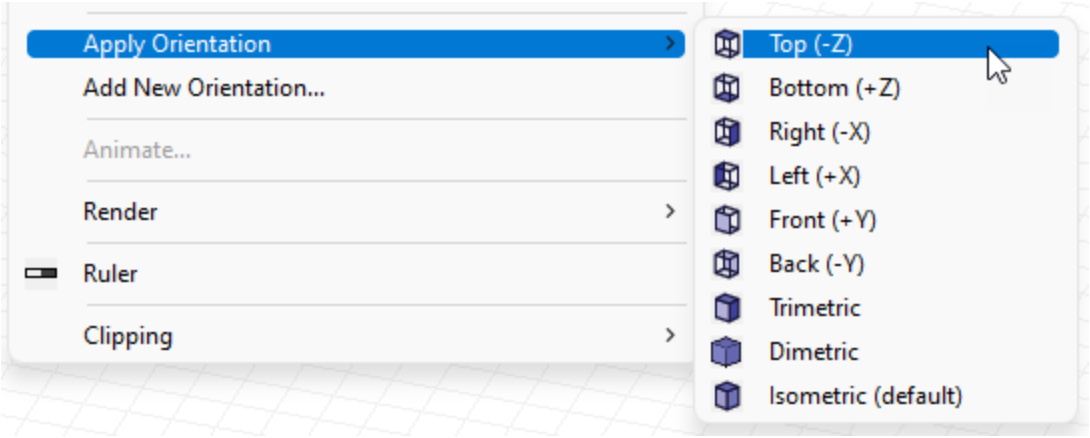


Possible orientations are Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric. These last three differ as shown below:



- Right-click in the Modeler window. In the shortcut menu that appears, go to the **View** > **Apply Orientation** submenu:

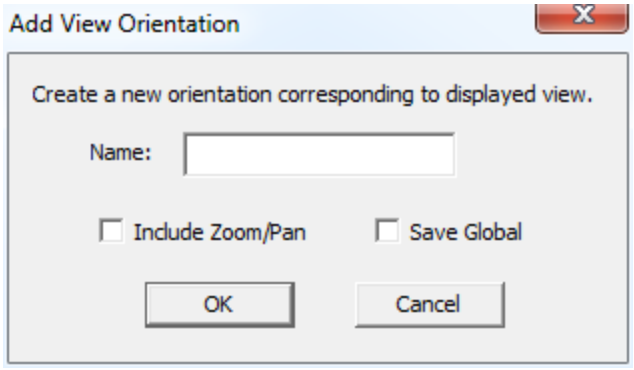
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In addition to the standard views, any custom views that you have defined will appear in this submenu.

After applying an orientation, you can click **View** > **Undo View : *orientation***.

You can also right-click in the Modeler window and choose **View** > **Add New Orientation** to create a custom viewpoint.

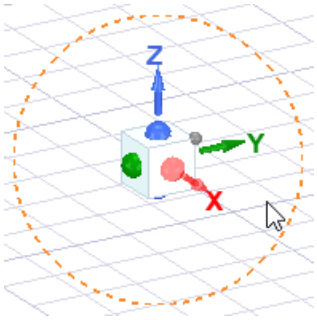


The **Name** that you assign will appear in the **View** > **Apply Orientation** submenu within the Modeler window's shortcut menu. You can specify whether to **Include Zoom/Pan** (in addition to the viewpoint orientation) and whether to **Save Global**, which makes the custom orientation available for all projects.

Using the Orientation Gadget

The **Orientation Gadget** is a tool for quickly manipulating the view orientation of 3D models and 3D plots graphically. It is a convenient alternative to using the **Orient** drop-down menu on the **View** ribbon tab. The Orientation Gadget is enabled by default and appears near the lower left corner of the display area in *Modeler* and *3D Plot* windows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The orange dashed circle appears only when the cursor is near the gadget, and its purpose is discussed later in this topic.

There is a colored dot at the center of each face of the cube and red (X), green (Y), and blue (Z) axes, indicating the axis positive directions for the currently active coordinate system. Additionally, there is a smaller gray dot at the corner where the +X, +Y, and +Z faces meet.

The following list summarizes the results of clicking different parts of the gadget (and optionally dragging the mouse while clicking):

- **Click the large dot at the center of any of the six cube faces:**

  This action rotates the view orientation to place the side of the model that the clicked face represents towards you, parallel with the screen. For example, if you click the dot on the -Y face, the -Y side of the model faces you, which is the *Left* view (+Y viewing direction).

  > **Note:**
  >
  > If the current viewing direction corresponds to one of the global or active user-defined coordinate system axes (±X/Ux, ±Y/Uy, or ±Z/Uz), clicking the axis label or dot at the center of the Orientation Gadget (on the face that is towards you) reverses the view orientation. So, with a single click of the mouse, you can switch from front-to-back, right-to-left, and top-to-bottom (or the inverses of these view orientation changes).

- **Click on one of the gadget axes:**

  This action is equivalent to clicking the dot at the center of the same positive cube face. The axis orientation becomes perpendicular to the screen and towards you. Therefore, the resulting three views are looking at the +X (front), +Y (right), or +Z (top) side of the model (-X, -Y, or -Z viewing direction, respectively).

- **Click the small gray dot at the cube's +X, +Y, +Z corner:**

  This action produces the standard *Isometric* view orientation.

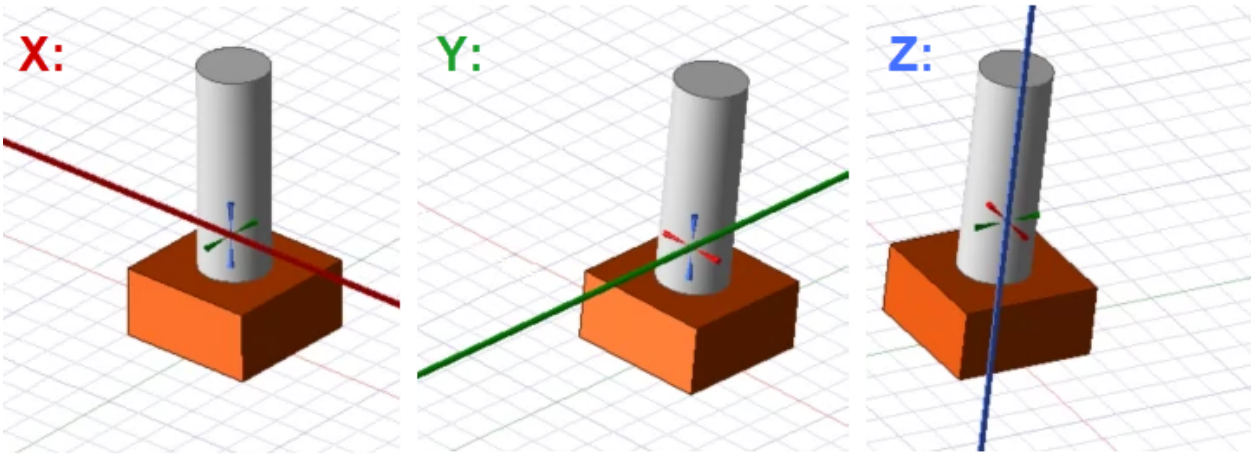
- **Click on or very close to the cube (but not on an axis or dot) and drag the mouse:**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

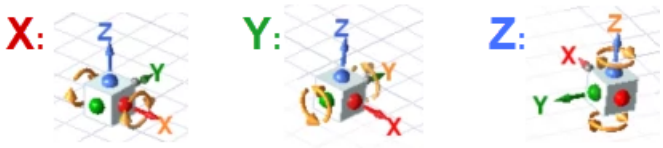
This action freely rotates the view orientation. The rotation center depends on the **Default Rotation About** option selected in the *3D UI Options* dialog box. The behavior is equivalent to clicking and dragging in the display area using the middle button / wheel but is performed with the left mouse button. Additionally, you can constrain rotation to be about any of the three active coordinate system axes by pressing one of the following keys while clicking and dragging the mouse:

- **X + click and drag** to rotate the view orientation about the active **X** axis.
- **Y + click and drag** to rotate the view orientation about the active **Y** axis.
- **Z + click and drag** to rotate the view orientation about the active **Z** axis.

During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When dragging the view orientation gadget to rotate the view, the center of rotation is the centerpoint of the drawing canvas. The following image shows the appearance of the rotation axes and center markers:



Additionally, a pair of gold, circular arrows appear on the view orientation gadget indicating the axis of constrained rotation:



While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

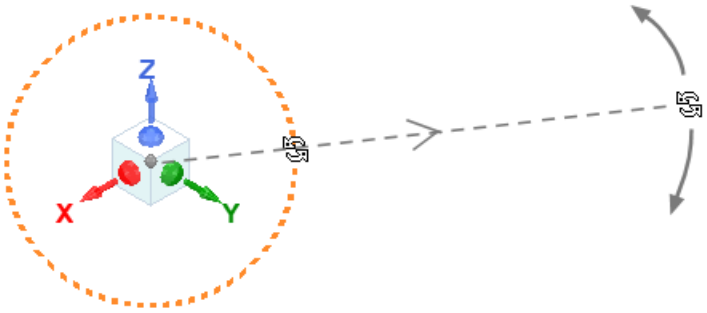
For more information and tips about the usage of this feature, see Rotating or Spinning the View.

- **Orange dashed circle:**

  This circle is used to rotate the current view about an axis normal to the screen. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

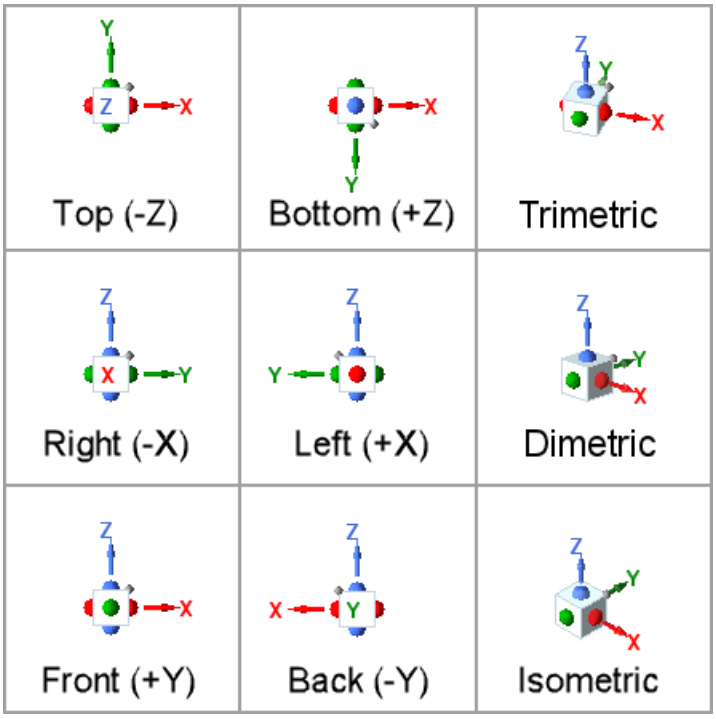
  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl+click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise rotation. The greater the distance from the gadget, the finer the rotational control.



The appearance of the Orientation Gadget for each of the standard view orientation is as shown in the following image:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

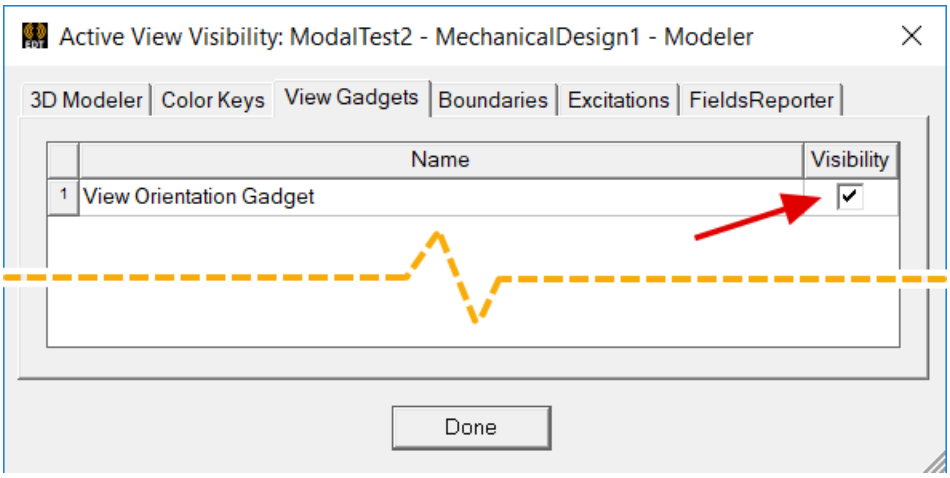
The orientation, colored dots, and axes of the Orientation Gadget correspond to the active coordinate system, whether **Global** or **user-defined**. All standard views are defined in the same manner as they are for the global CS, but Ux, Uy, and Uz directions are used instead of global X, Y, and Z.

Controlling Visibility of the Orientation Gadget

You can enable or disable the Orientation Gadget visibility as follows:

1. Access the *Active View Visibility* dialog box using one of the following two methods:
   - From the menu bar, click **View > Visibility > Active View Visibility**.
   - On the **View, Draw,** or **Model** ribbon tab, click **Hide/Show overlaid visualization in the active view**.
2. Select the **View Gadgets** tab.
3. Select or clear the checkbox in the **Visibility** column to the right of **View Orientation Gadget** to control the gadget visibility.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Done** to close the dialog box.

Rotating or Spinning the View

You can rotate the view relative to the Model Center, the Screen Center, the Current Axis (that is, the active coordinate system origin), or the Cursor location. You can activate the rotation modes via the menu bar, ribbon tabs, or shortcut menu. When a *Rotation* command is active, you can click and drag using the **left mouse button** to manipulate the model viewpoint.

Rotation Using Only the Mouse: You can use the **middle mouse button** to rotate the view *without* selecting a *Rotation* command. From the menu bar, click **View** > **Options** to access the *General* tab of the *3D UI Options* dialog box. Here, you can choose the desired **Default Rotation About** option. The view rotates about the **Screen Center** by default. The other available options are rotation about the **Model Center**, **Current Axis** (active coordinate system origin), or **Cursor** location. This setting also controls the rotation center for the **Spin** command.

Rotation using Only the Keyboard: You can use the key combination **Alt** + ***arrow keys*** to rotate the model view about a vertical or horizontal axis. Similar to using the middle mouse button, the center of rotation, through which the vertical or horizontal rotation axis runs, is determined by the **Default Rotation About** selection in the *3D UI Options* dialog box.

For each active *Rotation* command, or when using the middle mouse button, the behavior differs depending on where you click and drag, as follows:

- **Click and drag the cursor near any border of the Modeler window's display area:**

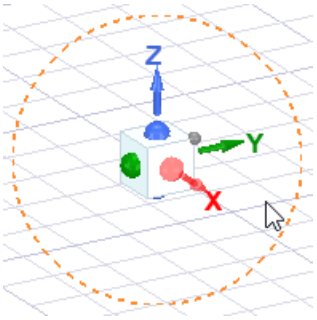
  The view is constrained to rotate about an axis that is perpendicular to the screen. The location of the axis, which is the rotation centerline, depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag the cursor within the interior of the display area (not close to a border):**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the view rotates freely about any axis, depending on the direction you drag and the relative positions of the cursor and center of rotation. The rotation center depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag on or near the View Orientation Gadget to freely rotate the view:**



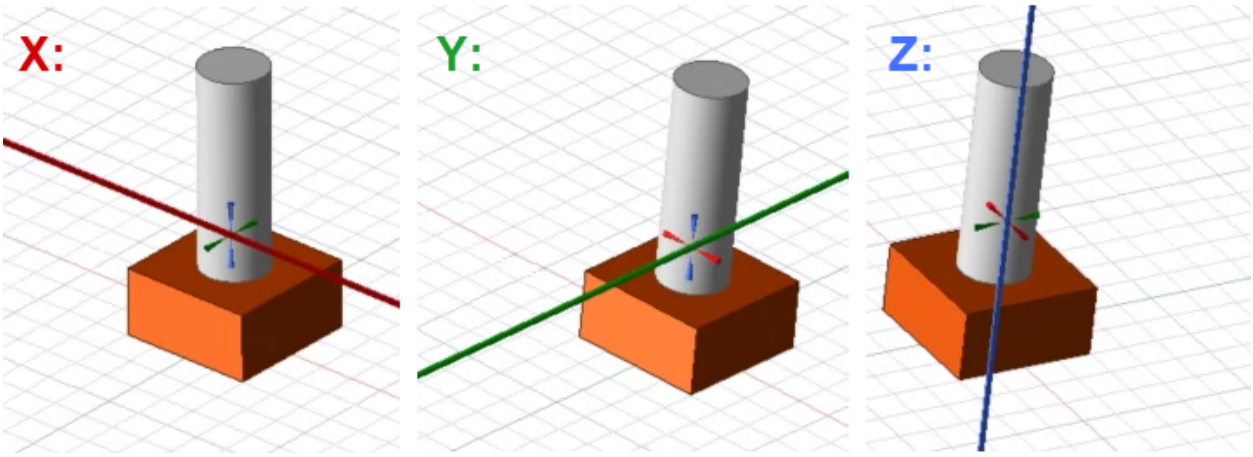
- **Constrained rotation about the X, Y, or Z axis**:

  Use the following hotkeys to constrain rotation of the model viewpoint to be about a coordinate system axis (global or user-defined). These hotkeys work whether you perform the rotation using the **middle mouse button**, any of the **Rotate** commands available on the ribbon (under the *View, Draw,* or *Model* tabs), or any of the **Rotate** commands in the menu bar (**View** > **Interactions** > **...**).

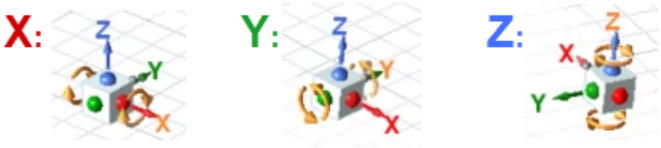
  - **X + click and drag** to rotate the view orientation about the active **X** axis.
  - **Y + click and drag** to rotate the view orientation about the active **Y** axis.
  - **Z + click and drag** to rotate the view orientation about the active **Z** axis.

  During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When using the middle mouse button, the center of rotation is the centerpoint of the drawing canvas. For menu and ribbon commands, you can choose alternative rotation centerpoints (model center, cursor location, or origin), which are covered further down in this help topic. The following image shows the appearance of the rotation axes and center markers:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Additionally, a pair of gold, circular arrows appear on the view orientation gadget, indicating the axis of constrained rotation:



While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

This feature provides a very convenient means of achieving the ideal model viewpoint for setup operations and for presentation purposes, giving you much more control than free rotation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tip:

- For best results, drag the cursor in a roughly circular path around the rotation centerpoint. The model orientation will follow the cursor movement, with the angle of rotation matching the angle of the cursor's arc about the same centerpoint.
- Hold down the X, Y, or Z key throughout the constrained rotation operation. The axial rotation constraint is canceled as soon as you release the key, though the rotation mode remains active. Depending on how you initiated rotation, cancel it by either releasing the middle mouse button, pressing **Esc**, or clicking the command again.
- During an active constrained rotation operation, you can release the current hotkey and press one of the other hotkeys to choose a different rotation axis, without first canceling the operation.

- **Rotating the view about an axis normal to the screen:**

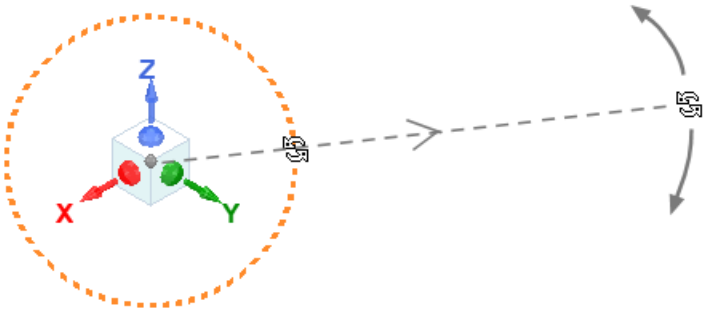
  This orange dashed circle that appears around the view orientation gadget when the cursor is nearby is used to rotate the current view about the screen-normal axis. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl + click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise

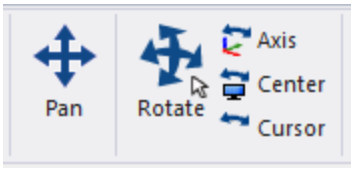
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotation. The greater the distance from the gadget, the finer the rotational control.



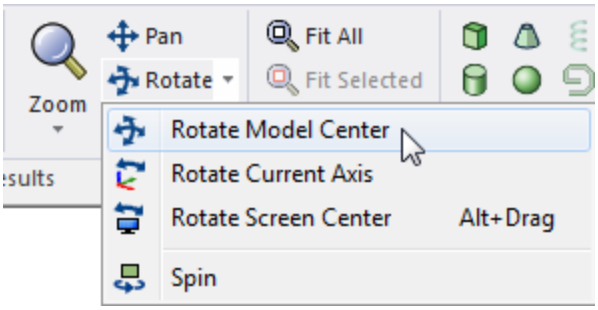
To rotate the view about the **Model Center**:

1. Choose the *Model Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, choose **Rotate**:



> **Note:**
>
> The *Center* icon on the *View* ribbon tab is for rotation about the *screen center*, not the model center.

   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Model Center**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> After selecting a rotation mode, the selected option becomes the default operation for the remainder of the application session or until a different rotation option is selected. To repeat the last rotation command, you do not have to access the drop-down menu; just click **Rotate**:

   - From the menu bar, click **View** > **Interaction** > **Rotate Model Center**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Model Center** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

Current Axis option – To rotate the view about the origin of the currently active coordinate system axes:

1. Choose the *Current Axis* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Axis**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Current Axis**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Current Axis**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Current Axis** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Screen Center**:

1. Choose the *Screen Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Center**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Screen Center**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Screen Center**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Screen Center** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Cursor** location:

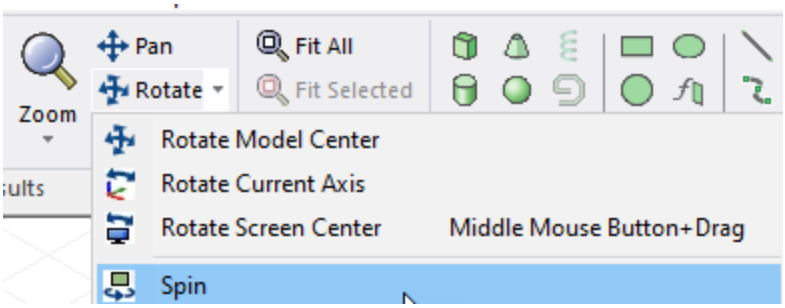
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose the *Cursor* rotation mode using one of the following three methods:
   - From the **View** ribbon tab, click **Cursor**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Cursor**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Cursor** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

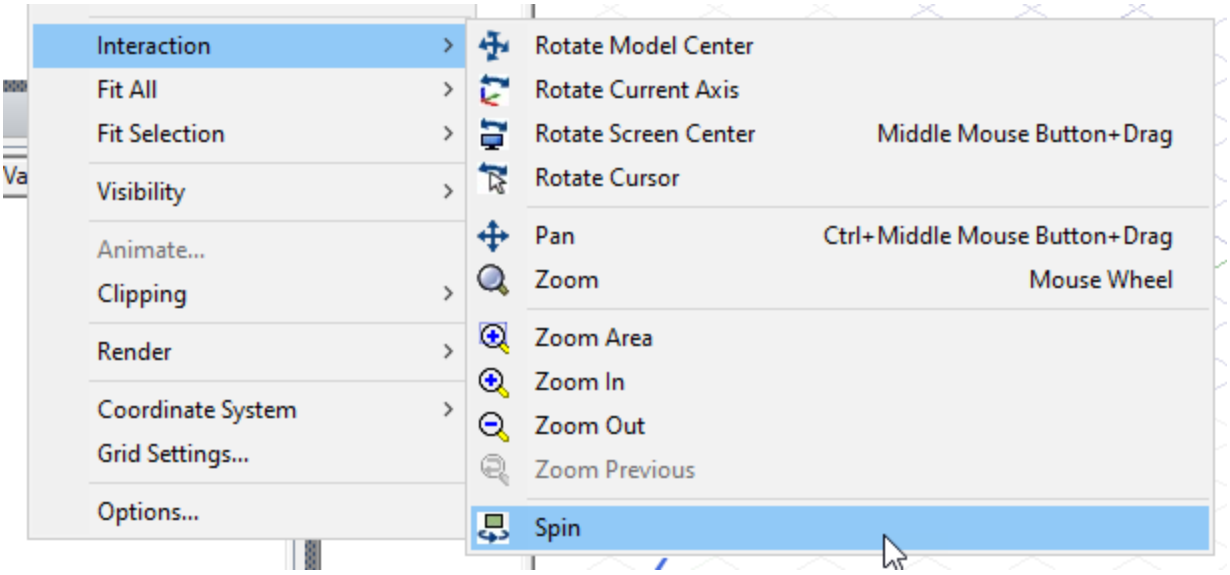
Spin Mode: This tool is useful for capturing animations (for presentation purposes) that show a 360 degree view of the model geometry, mesh, or simulation results.

To **Spin** the model view (that is, to keep it rotating after you release the mouse button):

1. Choose the *Spin* mode using one of the following two methods:
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Spin**.



   - From the menu bar, click **View** > **Interaction** > **Spin**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

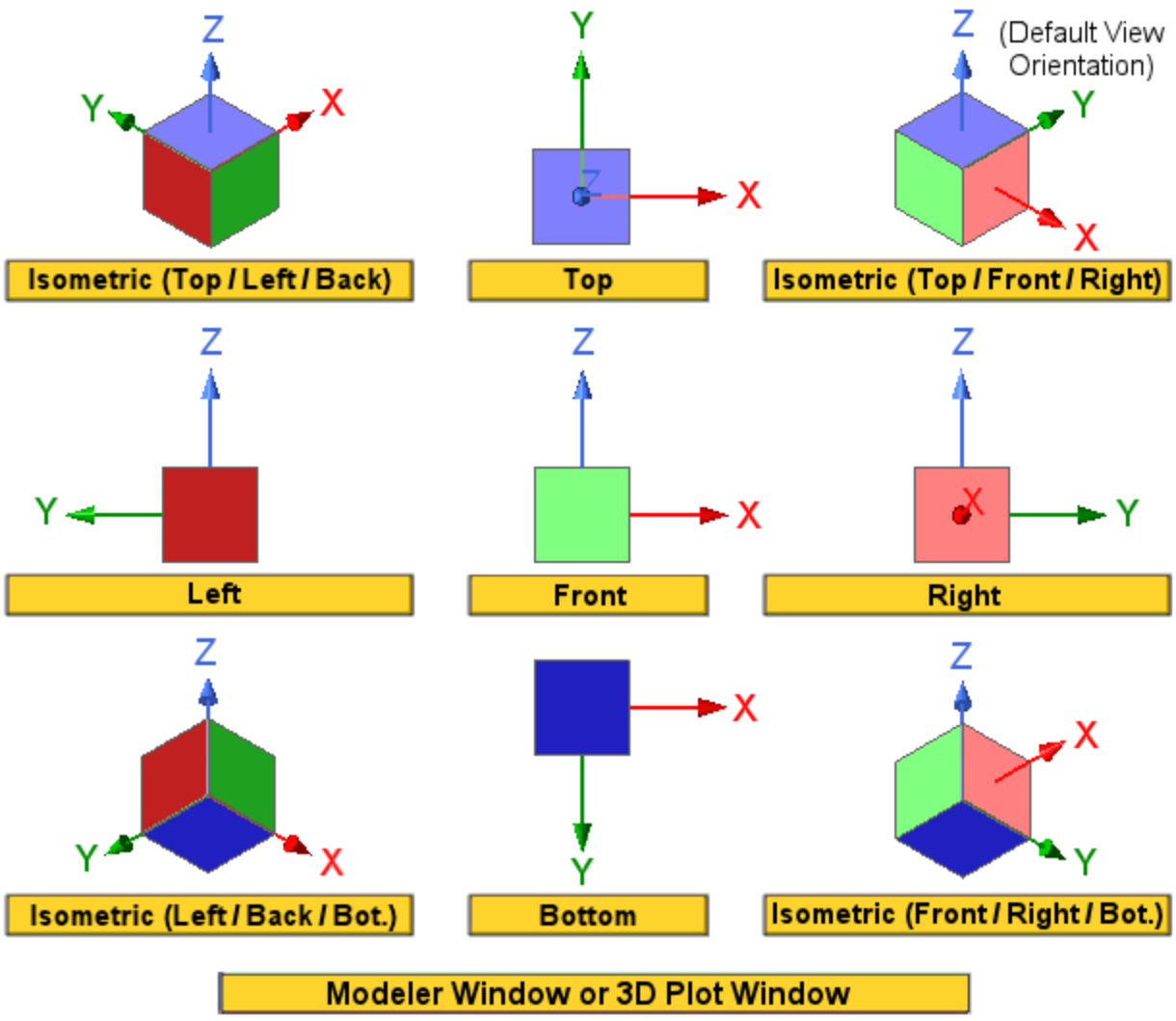
2. Click and drag the mouse in the direction you want to spin the view. Release the mouse button while still dragging.
    - The model will continue to spin in the specified direction. The spinning speed will be proportional to how fast you were dragging the cursor when you released the mouse button.
    - The center of rotation for the *Spin* mode is controlled by the *Default Rotation About* option selected in the 3D UI Options dialog box.
    - The *X, Y,* and *Z* keys also work as modifiers for the *Spin mode* (for constrained spinning about one of the active coordinate system axes).
3. To exit the *Spin* mode, click the command again (per step 1) or press **Esc**.

> **Note:**
>
> If rotation response is slow, especially for complex models:
>
> - For some graphics cards, you can improve the performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**.
>
>   For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

Changing the Model View with Alt+Double-Click Areas

In the following figure, the colored boxes show the nine view orientations you can obtain by using **Alt + double-click** in the corresponding areas of the *Modeler* window. Each face is a unique color and is named according to the standard view orientation names. The **Alt + double-click** view navigation method is also applicable to report windows containing *3D Plots* (such as a 3D Polar Gain plot).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Back, Trimetric,* and *Dimetric* standard view orientations are not available via *Alt + double-click* zones. For these three views, use other available ribbon commands or shortcut menus. For user-defined views, use the *Update View Orientation* dialog box or the Modeler window's shortcut menu (*Right-click* > *View* > *Apply Orientation*).

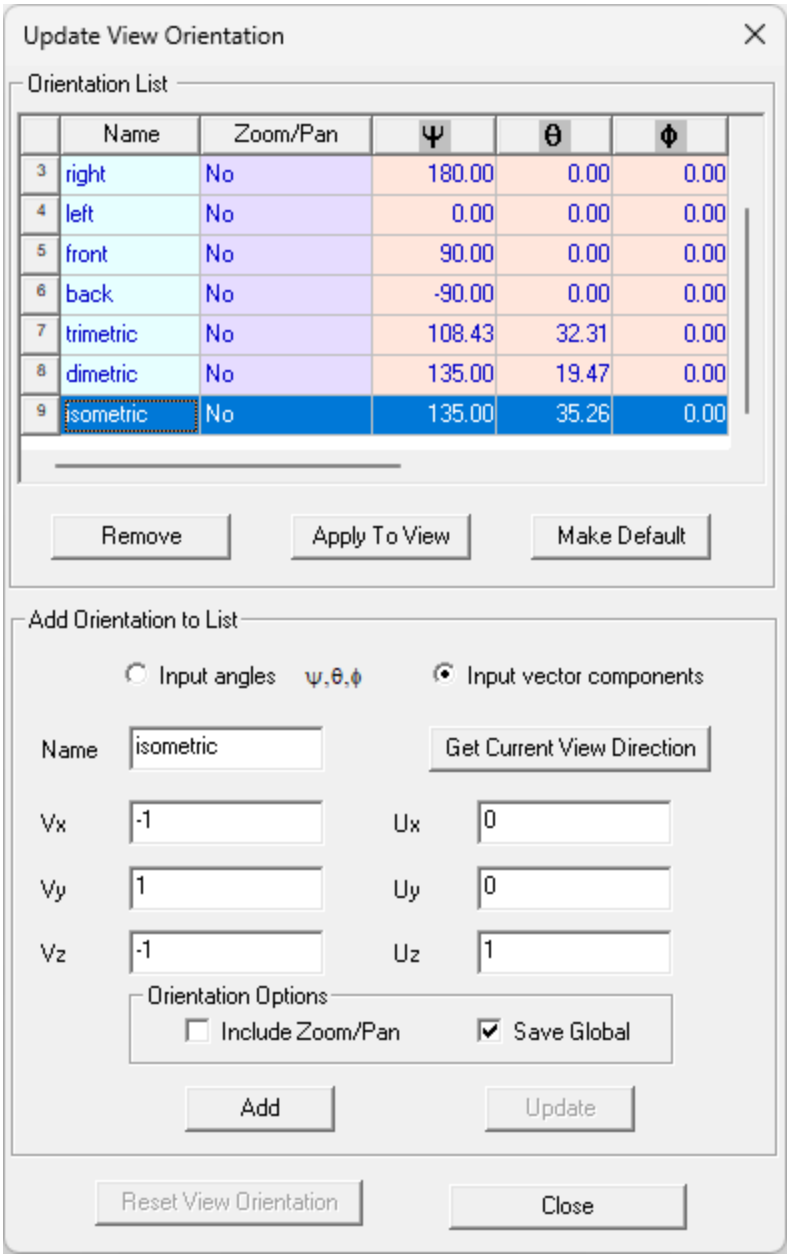
Applying a Default View Orientation

To apply a default viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click one of the orientation names listed in the orientation list.
3. Click one of the orientation names listed in the viewing directions list.
4. Click **Apply**.

   The viewing direction will be applied to the active view window.

   You can use the **Reset View Orientation** button to restore the view to the original angle.
5. Click **Close**.

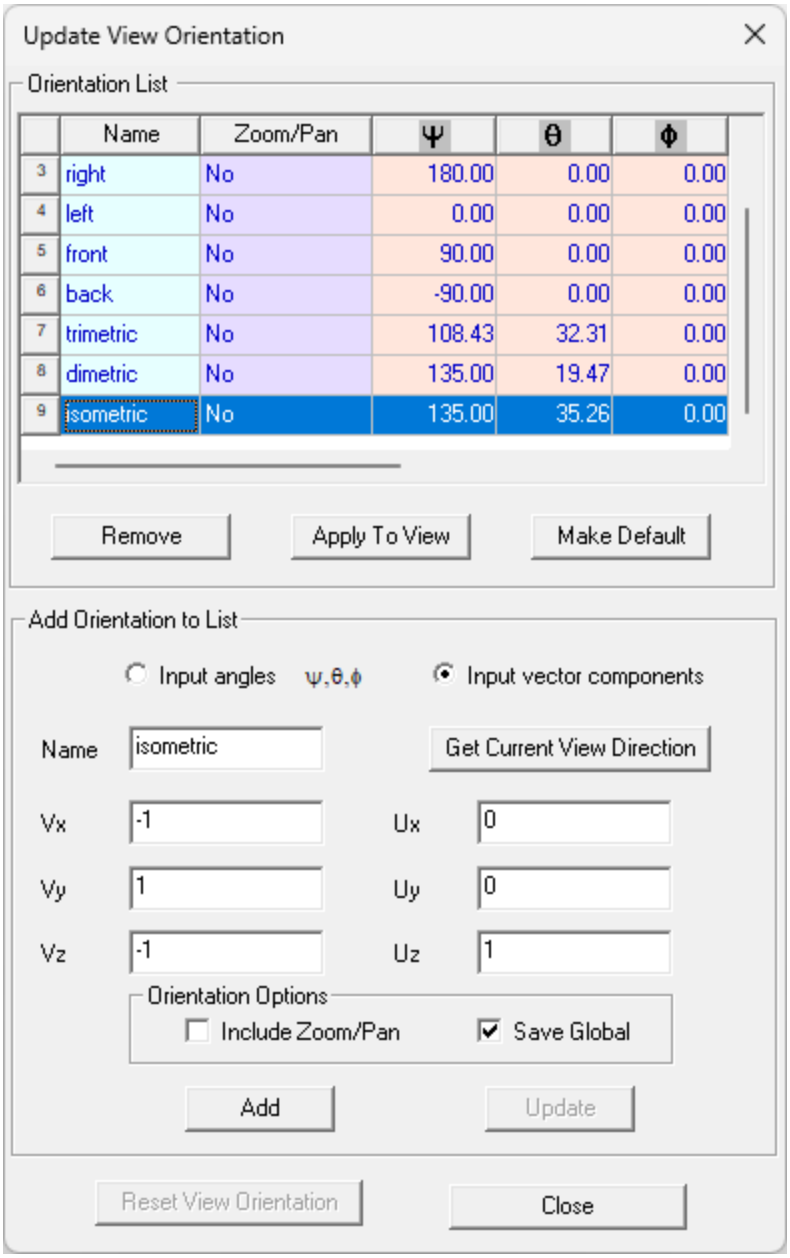
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Applying a New View Orientation

To create and apply a new viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To create a viewing direction that is based on a default viewing direction, click the existing orientation name in the viewing directions list.

   To create a viewing direction based on the current view in the **Modeler** window, click **Get Current View Direction**.

   - To modify the selected orientation's vector components, select **Input vector components** under **Add Orientation to List** and then modify the values in the **Vx, Vy**, or **Vz** text boxes, and the **Up vector** boxes for **Ux, Uy,** and **Uz**.
   - To modify the selected orientation's input angles, select **Input angles** under **Add Orientation to List** and then modify the values in the **psi, phi** and **theta** text boxes.
3. Type a name for the new orientation in the **Name** text box.
4. Click **Add/Edit**.

   The new orientation is added to the list of viewing directions.
5. Click **Make Default** if you want the new viewing direction to be the initial viewing direction when a **3D Modeler** window is opened in the current project or future projects.
6. Click **Close**.

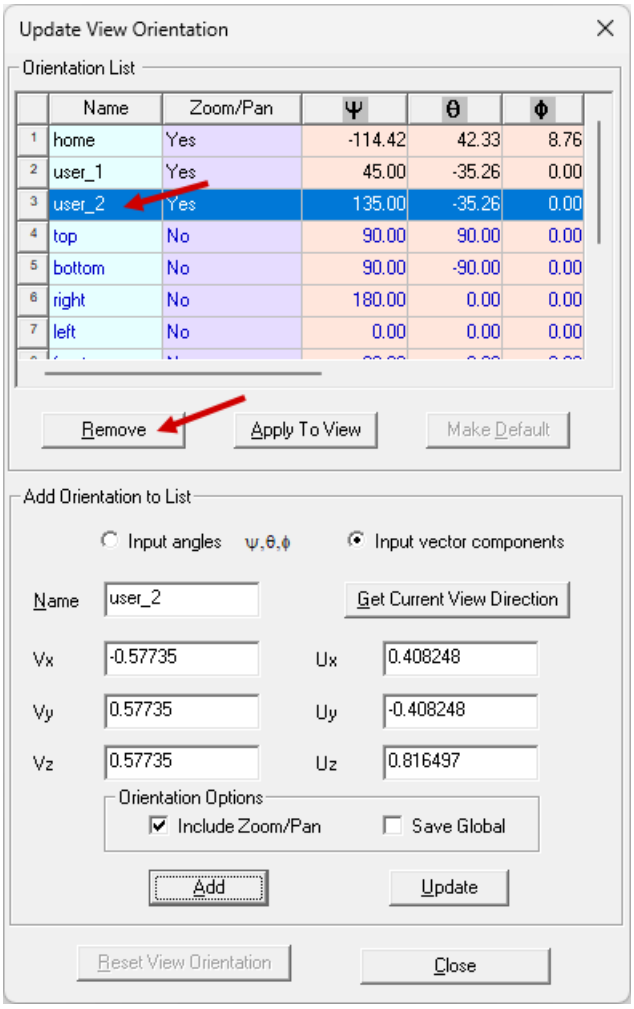
Removing an Orientation

To remove a viewing direction from the list in the orientation settings dialog box:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the viewing orientation you want to delete from the list of names.
3. Click **Remove**.

   The viewing direction is removed from the list.

> **Note:**
>
> This operation cannot be undone.

Fitting Objects in the View Window

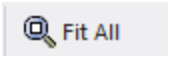
What do you want to do?

- Fit all objects or traces in a view window
- Fit selected objects in a view window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fitting All Objects in a View Window

To fit all the views: click **View** > **Fit All** > **All Views** or use the equivalent icons in the **Draw** or **Model** ribbon tabs.



All view windows displaying the active design change to include all model objects.

To fit only the active view: click **View** > **Fit All** > **Active View**.

The view in the active Modeler window changes to include all model objects.

> **Tip:**
>
> Alternatively, fit all objects in the active view window using one of the following methods:
>
> - Press **Ctrl+D**
> - Right-click in the Modeler window and then click **View** > **Fit All** on the shortcut menu.

When **Fit All** is used in a **Report** view, the window is automatically rescaled to fit all traces in the window and the axis label and ticks are rescaled.

Fitting Selected Objects in a View Window

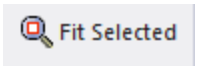
To fit one or more selected objects, 3D Components, or User Defined Models in the Modeler window or traces selected in the Reporter:

1. When you are working on a model view, select the objects you want to fit in the view. When you are working on a report, select the traces you want to fit.
   - To fit the selection in the active view window: Click **View** > **Fit Selection** > **Active View**.
   - To fit the selection in every open view window of the active design: Click **View** > **Fit Selection** > **All Views.**

     > **Tip:**
     >
     > Alternatively, fit the selection in the active view window by clicking **View** > **Fit Selection** on the shortcut menu.

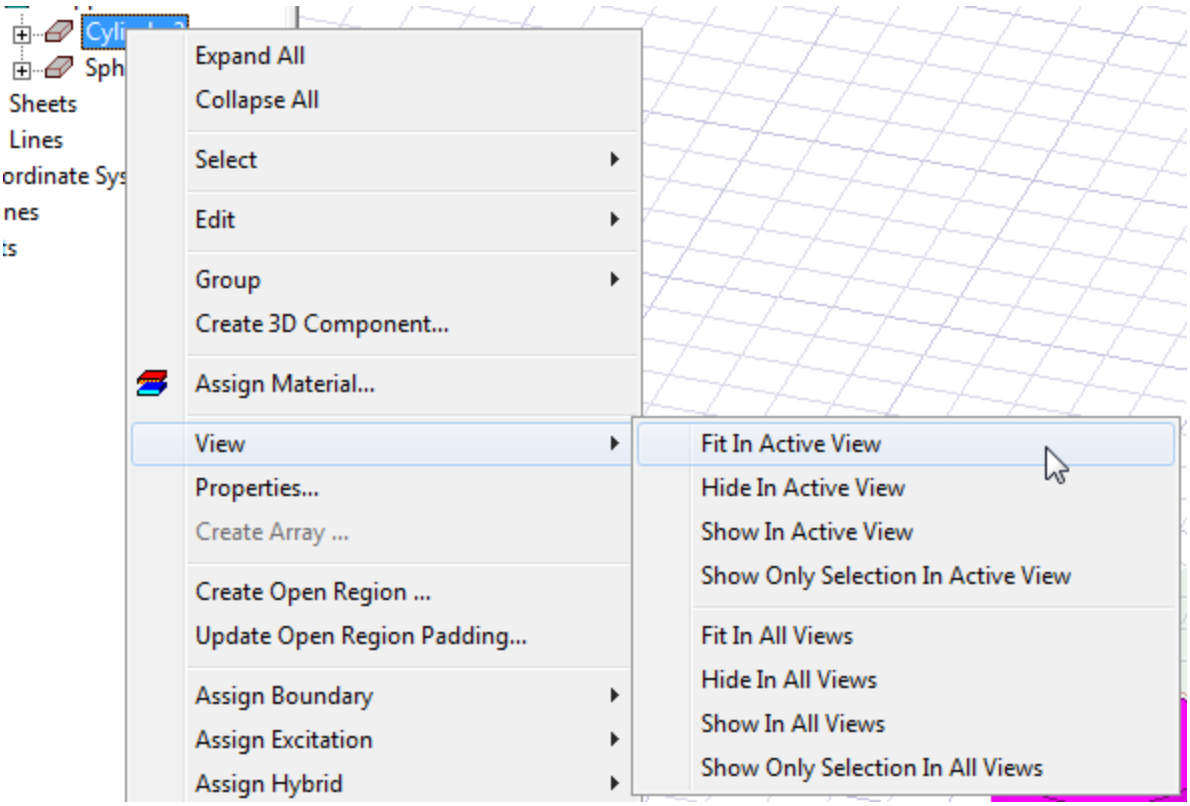
   You can also use the Fit Selected icon in the **Draw** and **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To fit one or more objects or components selected in the History Tree:

1. Select the objects or components of interest and right-click to display the shortcut menu.
2. Select **View** > **Fit in Active View** or **Fit in All Views**:



.The view adjusts to fit the select objects.

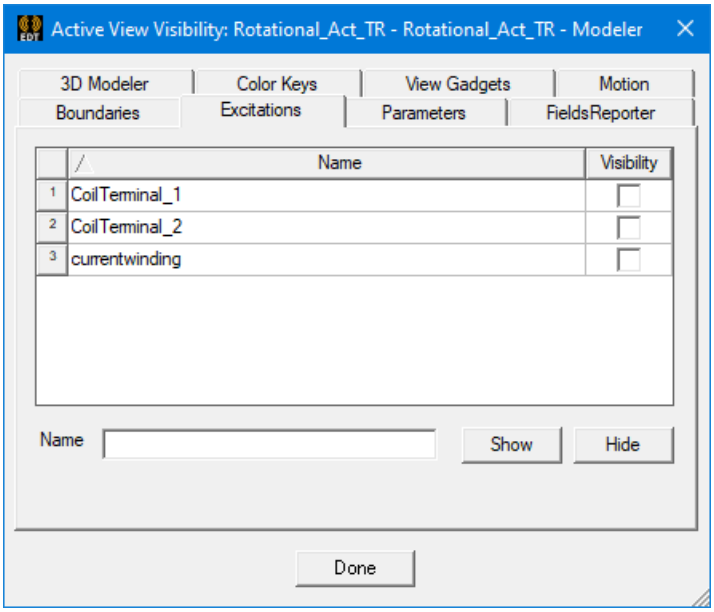
Active View Visibility Dialog Box

If there are many objects, it may be easier to show or hide objects using the **Active View Visibility** dialog box.

1. Click **View** > **Visibility** > **Active View Visibility**, or click the equivalent **Hide/Show** icon on the **View, Draw,** or **Model** ribbon tabs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Active View Visibility** dialog box appears.



2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
    - For designs with large numbers of objects, you can resize the dialog for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.
3. Under the tab, clear the **Visibility** option for the objects you want to hide in the active view window.

    The objects you designate are hidden.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Rendering Objects as Wireframes or Solids

To render (display) *all* objects in the view window as wireframe outlines, flat-shaded solids, or smooth-shaded solids:

1. Click **View**> **Render** and click one of the following:
   - **Wireframe**:

     The objects in the view window are displayed as skeletal structures, enabling you to see all sides of the objects at one time.

     You can also use the **F6** key to display a wireframe view.
   - **Smooth Shaded**:

     The objects in the view window are displayed as shaded objects with smooth edges.

     You can also use the **F7** key to display a smooth shaded view.

To render a *single* object in the view window as a *wireframe outline*:

1. Select the object you want to render as a wireframe.
2. In the **Attribute** tab of the docked **Properties** window, select **Display Wireframe**.

Setting the Default View Rendering Mode

To set a default rendering mode for all objects created in the active design and in future designs:

1. Click **Tools** > **Options** > **Modeler Options**.
2. Click the **Display** tab.
3. Select one of the following options from the **Default View Render Mode** drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wireframe**.

  The objects in the view window will be displayed as skeletal structures, enabling you to see all sides of the objects at one time.

- **Smooth Shaded**.

  The objects in the view window will be displayed as shaded objects with smooth edges.

4. Click **OK**.

   The rendering mode will be applied to all new objects you create.

Setting the Enhanced Display of Material Color and Transparency

You can specify the use of material color and transparency display by using the **View** > **Render** > **Enhanced Display** command. Any changes reset the default.

To set the Enhanced Display for the active modeler window:

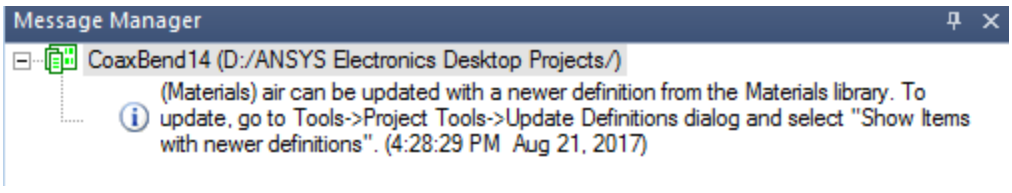
1. Click **View** > **Render** > **Enhanced Display** or press **F8**.

Enhanced Display overrides part color and transparency properties with material settings for all the parts visible in the active view. By default it is off (unchecked on the **View> Render** menu) and part attribute color and transparent values are applied.

It is possible for the view to be in the “Enhanced Display” mode but some of the parts in the model could have “Use Material Appearance” unchecked. In this case you can edit the **Color** or **Transparent** values, or select **Use Material Appearance** in the docked **Properties** window, but they are not applied to the active view visualization. This resembles how the **Wire Frame** setting is handled currently.

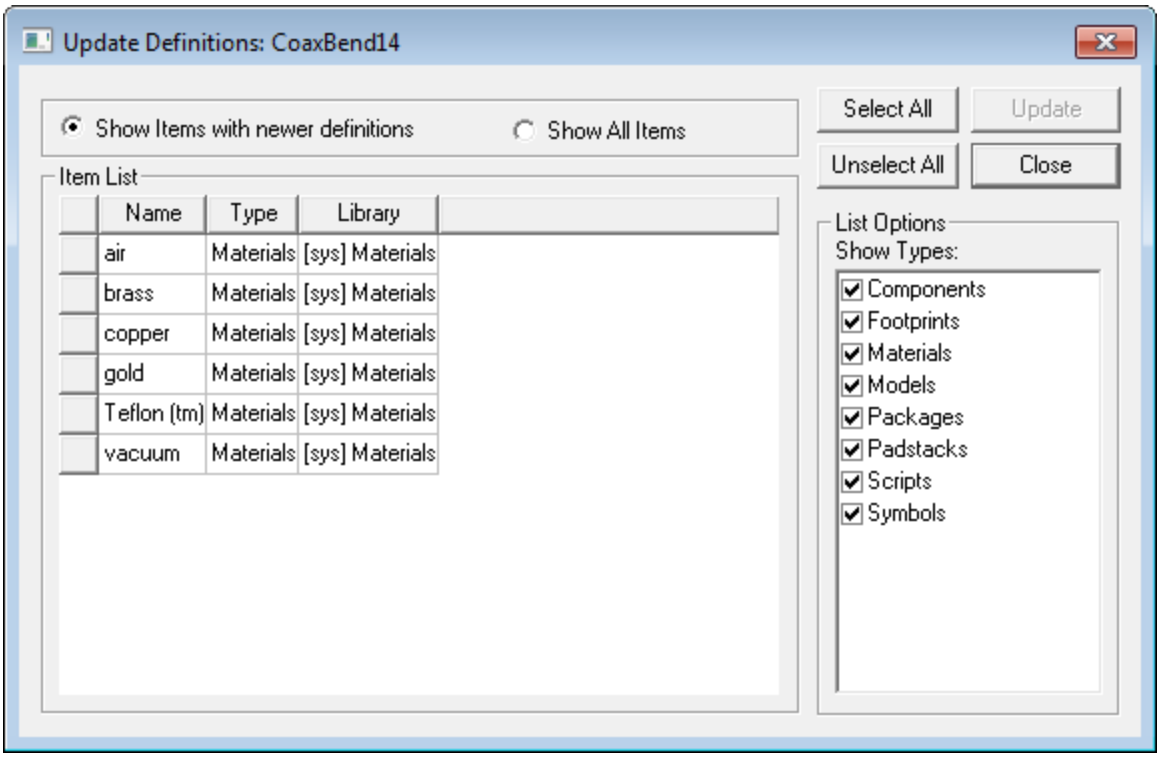
Sheet objects in the 3D modeler do not have a material assignment, but they could have a boundary assignment with material assigned to the boundary. In this case sheet object visualization resembles solid object visualization, where the material appearance from the assignment is used.

When you are working with a legacy project and use **View** > **Render** > **Enhanced Display**, if there are updates to the materials definitions for appearance, you will receive an info message informing you that the material definition was updated.



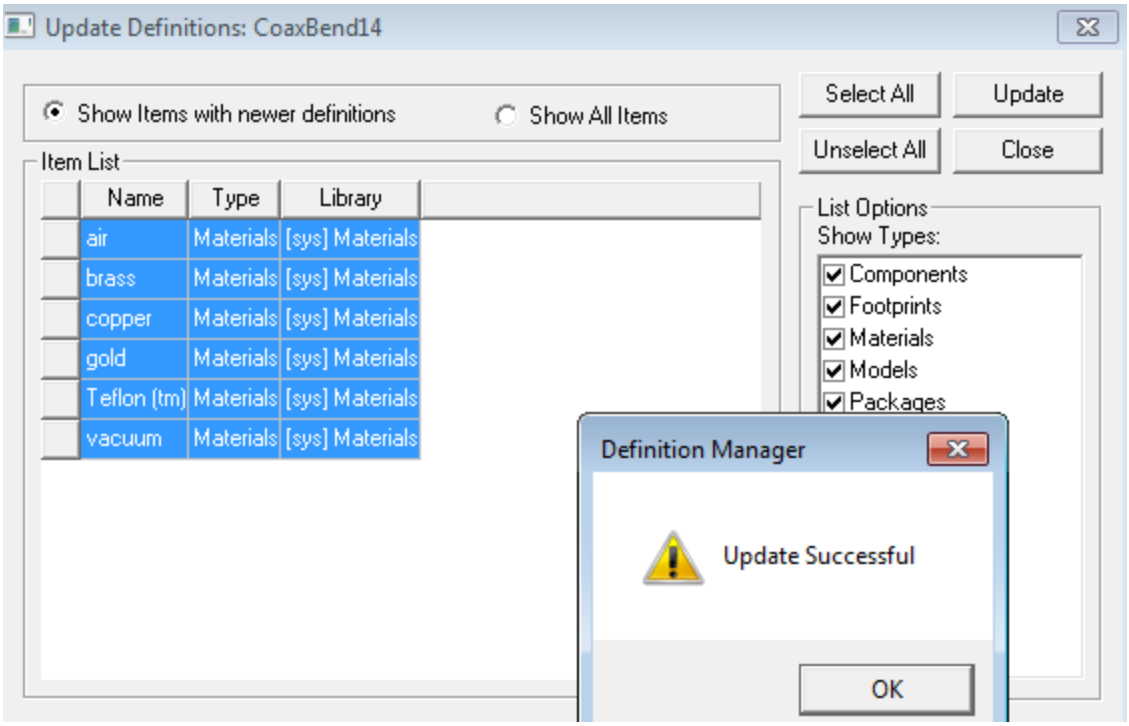
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to update, click **Tools** > **Project Tools** > **Update Definitions** to open the **Update Definitions** dialog box.



Here you can multi-select rows of materials to update them. In case of updates available from multiple libraries, you can choose the library location. Selecting material rows enables the **Update** button. Selecting **Update** then produces a message on a successful update.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting the Curved Surface Visualization

You can specify the faceting for rendering true curves by using the **View** > **Render** > **Curved Object Visualization** command. There are two options for control--Maximum surface deviation and Maximum normal deviation. This resembles the Mesh surface approximation settings. Reduce either or both of the allowed deviations to improve the image quality. Improved image quality comes at the cost of increased CPU consumption. Changes apply to the current model until they are changed again. Any changes reset the default. The default gives satisfactory results (cpu/memory consumption vs. graphical display) for various model complexities.

When you change Curved Object Visualization settings and apply them to a design, those settings are saved with design unless you change it again. That means when you open the design again, it will apply saved visualization settings and NOT the default settings. Because this affects the CPU and memory required to open the project, typically, you should not save a project with other than the default settings.

To set the Curved Object Visualization for the active modeler window:

1. Click **View** > **Render** > **Curved Object Visualization**.

   This command displays the **Curved Object Visualization**dialog for the active modeler window. The dialog contains areas for setting the Maximum Deviation, and the Maximum Normal deviation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the Maximum Deviation by first selecting from the radio buttons for **Ignore**, set as **Relative Deviation** or set as **Absolute Deviation**. Selecting the later two radio buttons enables the value field.

   When set as **Relative Deviation**, the actual surface deviation depends on the model size. For example, sphere with a radius of 10 has same number of facets as a sphere with a radius of 1. This means that CPU cost does not increase based on the model dimension.

   When set as **Absolute Deviation**, the maximum surface deviation for both the spheres will be approximately same since a bigger sphere has more facets than a smaller one. This means that the most CPU cost applies to the larger objects.

3. If you selected the radio buttons for Relative or Absolute Deviation for Maximum Deviation, enter a value in the field.
4. To change the **Maximum Normal Deviation**, enter a value in the text field. Units are degrees.

   > **Note:**
   >
   > Wire bodies cannot be rendered with a Maximum Normal Deviation value less than 1 degree. When using a setting less than 1 degree all wire bodies will be rendered with a setting of 1 degree and all closed bodies will be rendered with the dialog box setting.

5. The **Save As Default** button lets you Save any values you change to the drop-down menus for the fields.
6. The **Restore Default**s button lets you return to the original values. Any values you provided through **Save As Default** remain on the drop-down menus for the fields for surface and normal deviations.
7. Click **Apply** to apply the current values to the active modeler window, and **Close** or **Cancel** to close the dialog box without changing settings.

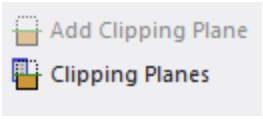
Using Clip Planes

The **View** > **Clipping** > **Clip Plane** command lets you define a clip plane that you can use to interactively make any desired cut-away view of a model. If you use **Edit** > **Copy Image** or **Modeler** > **Export**> **[image format]** with the clip plane active, the image shows the clipped plane. When parts of the model are hidden by a clip plane, model selection works as though only the visible parts are present.

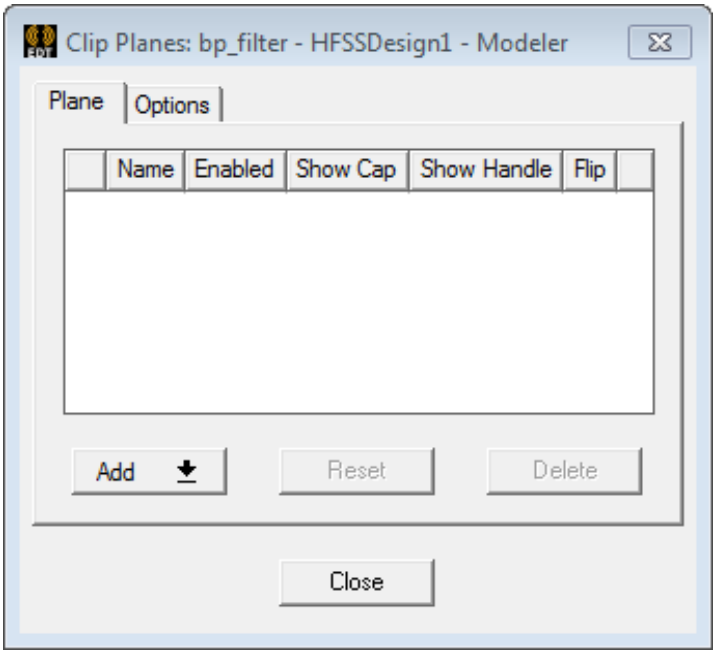
To add a clip plane:

1. Click **View** > **Clipping** > **Clip Plane** or click the **Clipping Planes** icon on the **View** ribbon tab.

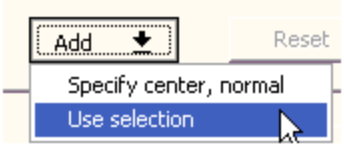
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This displays the **Clip Plane** dialog box with the **Plane** tab selected.

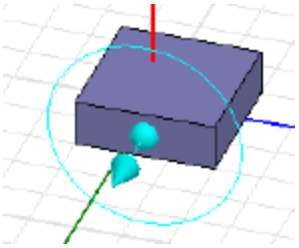


2. The **Add** button contains a drop-down menu with choices for **Specify center, normal**, and **Use selection**. If you want to use a selection, you must first select a face or an existing cut plane.

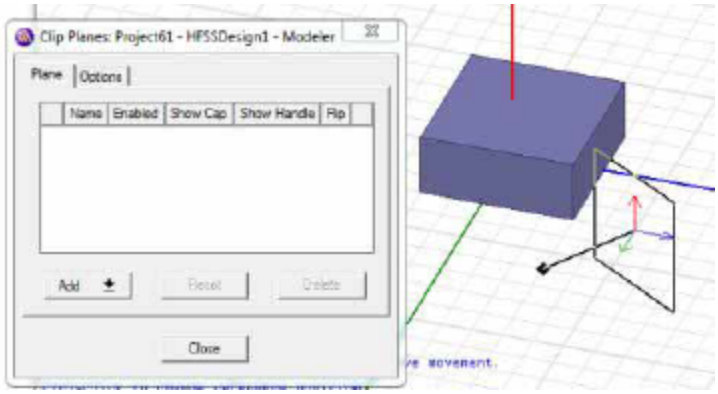


- If you first select a face or cut plane, and then click **Add** > **Use Selection**, the clip plane is added on that face.

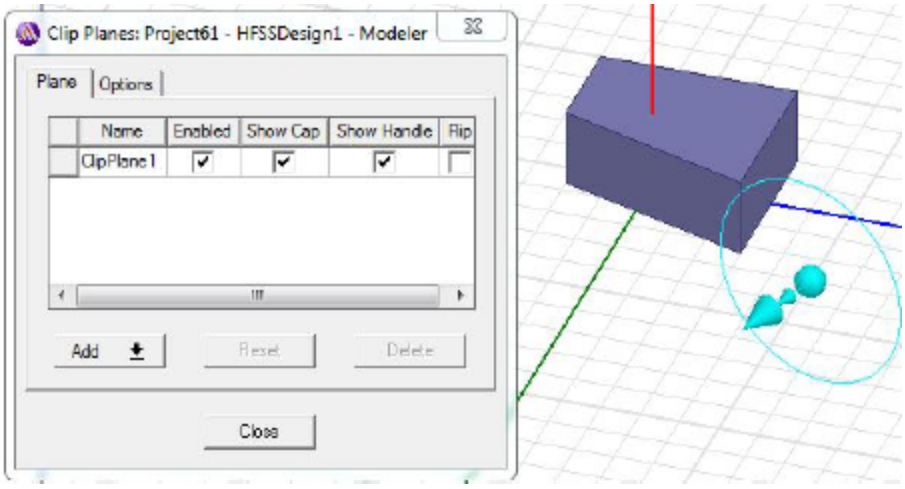
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you select **Add** > **Specify center, normal**, this launches a Measure dialog box and enters a mode for you to click to define the start location (shown as a triad).
    a. When you move the cursor, a rectangle represents the clip plane, and a vector the current direction.



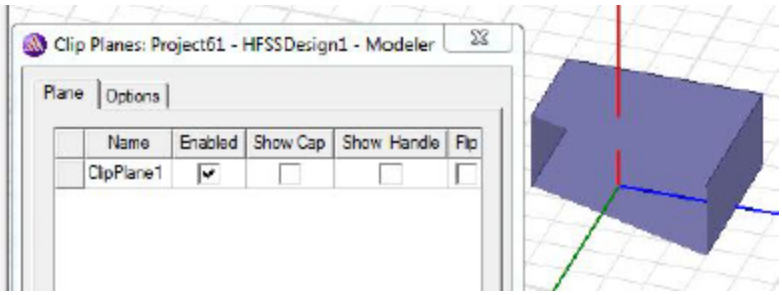
    b. Click again to set the reference position.



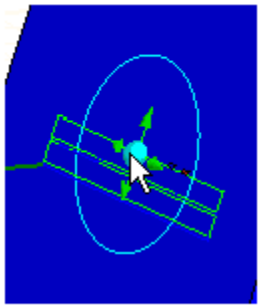
    After the second click, the clip plane is active. The handle is visible as a circle with a sphere at the center, and an arrow pointing the normal for the plane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

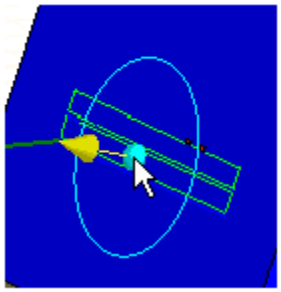
The **Clip Planes** dialog box shows the clip plane name, that it is enabled, the cap (which is the plane surface), and the handle. **Flip selection** lets you reverse the direction of the clip plane. If you deselect **Show cap** and/or **show handle**, they disappear from the display.



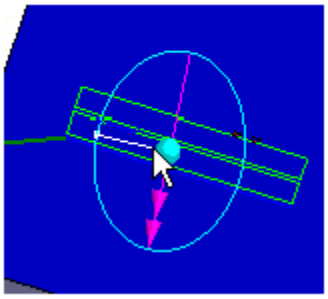
3. With **Show Handle** enabled, you can use the handle to manipulate the location and orientation of the clip plane. The handle changes appearance and function relative to the position of the cursor. Dragging the cursor makes use of the current function.



Move Handle independently of clip plane. This lets you move the handle away from the model.



Move Handle while dragging clip plane



Rotate clip plane by dragging the cursor. The white line is parallel to the clip plane rotation. The double arrow line is the axis.

Move the cursor around the center of the handle to change the axis.

4. The **Options** tab of the **Clip Planes** dialog box contains four options:
   - Force opaque for the unclipped portion.
   - Disable clip plane when drawing a new clip plane.
   - Plane handle color

     The button shows the current color. Click the button to display a color selection dialog box. Select a default or custom color and click OK.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plane handle radius.

  This slider lets you resize the radius of the handle to the most convenient size. The radius resizes dynamically. When you close and reopen the modeler window, the last selected size persists.

You can save your choices as new defaults.

Modifying the Lighting

You have the option to emit the following types of light on a design:

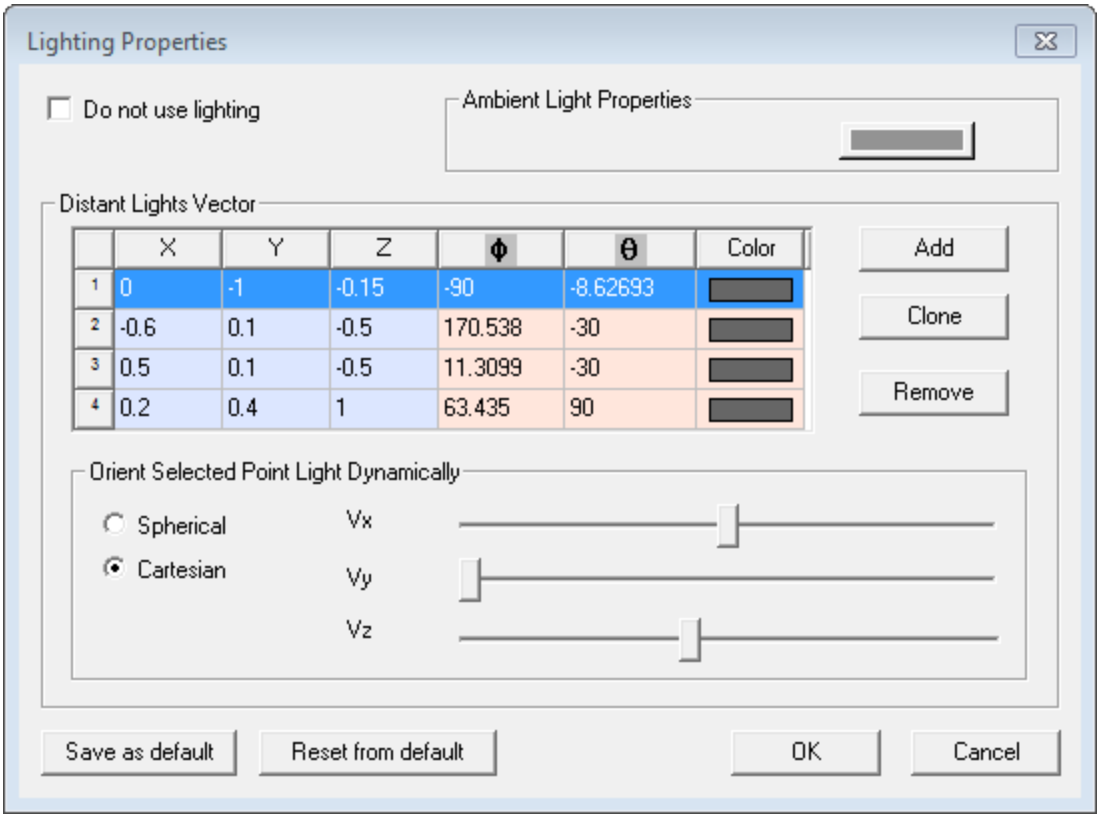
- *Ambient* lighting surrounds the model evenly with light. All objects are lit evenly in every direction by a color of light that you specify.
- *Distant* lighting directs a ray of light at the model in a direction you specify. By default, two distant light vectors are in effect for every new view window.

To modify the lighting:

1. Click **View** > **Modify Attributes** > **Lighting** or, on the **View** tab of the ribbon, click on **Modify Attributes** and select **Lighting** from the drop-down menu.

   The **Lighting Properties** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. To turn off ambient and distant lighting, select **Do Not Use Lighting**.
3. To surround the model with light, click the **Ambient Light Properties** color button and then select a color for the surrounding light from the *Color* palette.
4. To modify the distant light on a model, do one of the following:
    - Add a new distant light by clicking **Add**.
    - Copy an existing distant light that you intend to modify by first selecting it in the **Distant Light Vectors** table and then clicking **Clone**.
    - Select a default distant light to modify by selecting it in the **Distant Light Vectors** table.
5. For the selected distant light vector, specify the vector direction:
    a. To modify the direction by specifying Cartesian coordinates, do one of the following:
        - Enter the new Cartesian coordinates in the **X**, **Y**, and **Z** boxes.
        - Use the **Vx**, **Vy**, and **Vz** sliders to specify the Cartesian coordinates dynamically.
    b. To modify the direction by specifying the spherical coordinates, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter the new spherical coordinates in the Φ (phi) and θ (theta) boxes.
- Use the Φ and θ sliders to specify the spherical coordinates dynamically.

6. To revert to the default ambient and distant light settings, click **Reset**.
7. Click **Save As Default** if you want the new lighting settings to be the defaults for all **3D Modeler** windows, either in the current project or future projects.
8. Click **OK** to dismiss the dialog box.

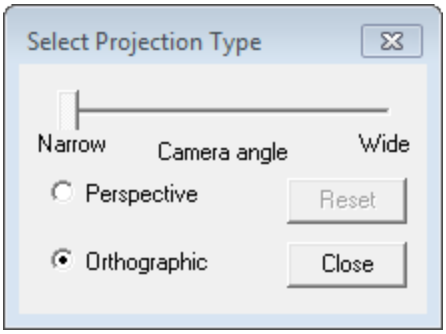
The lighting settings will be saved with the design. New lighting applied to other designs after this point, including new default settings, will not affect these lighting settings.

Setting the Projection View

To modify the projection of model objects (the camera angle) in the view window:

1. Click **View**> **Modify Attributes**> **Projection** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Projection** from the drop-down menu.

   The *Select Projection Type* dialog box appears:



2. Select **Perspective** to enable the slider to change the angle of the view.
   - Move the slider to the right to increase the proximity, or widen, the view. Move the slider to the left to decrease the proximity, or flatten, the view.

   Objects that are closer appear larger relative than objects that are farther away.

3. Select **Orthographic** to view the model without distortion.

   The slider is disabled because a distortion scale is no longer applicable.

4. Click **Reset** to return the model to its original view.
5. Click **Close** to accept the projection setting and dismiss the window.

   The *Select Projection Type* dialog box closes. The last view you specified in the projection window remains visible in the view window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

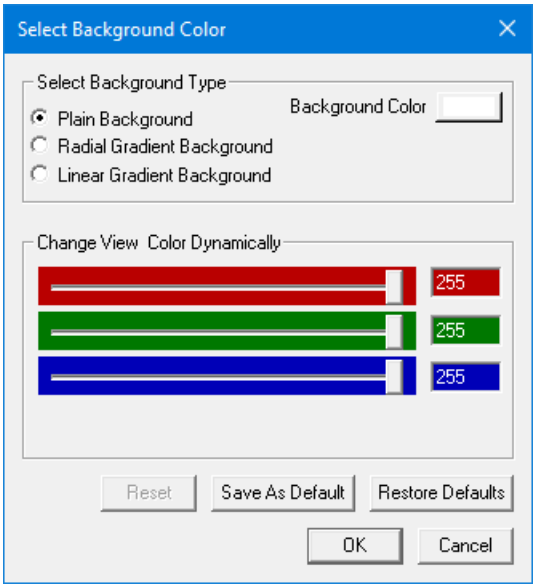
The projection view you set will be saved with the design. New projection views assigned to other designs after this point will not affect this projection setting.

Setting the Background Color

To set the color of the background in the view window:

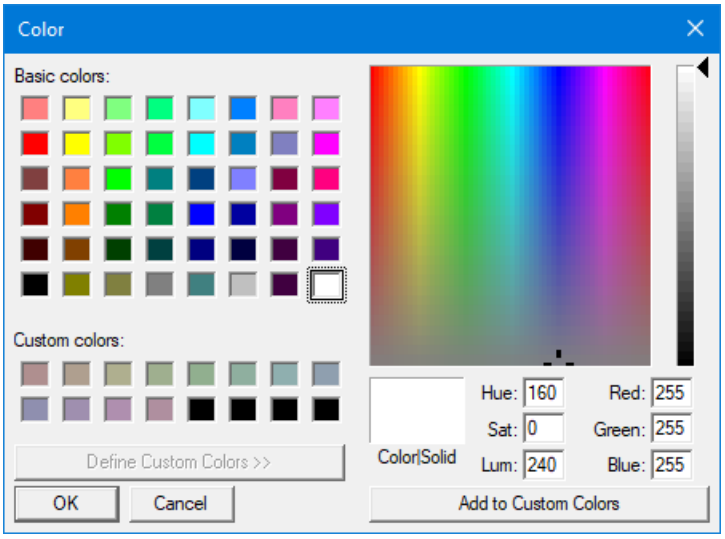
1. Click **View** > **Modify Attributes** > **Background color** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Background Color...** from the drop-down menu.

   The **Select Background Color** window appears.



2. To assign a solid background color, do the following:
   a. Select **Plain Background**.
   b. Modify the background color in one of the following ways:
      - Click the **Background Color** button and then select a color from the **Color** palette.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Use the RGB sliders under **Change View Color Dynamically** to specify the color's red, green, and blue values.

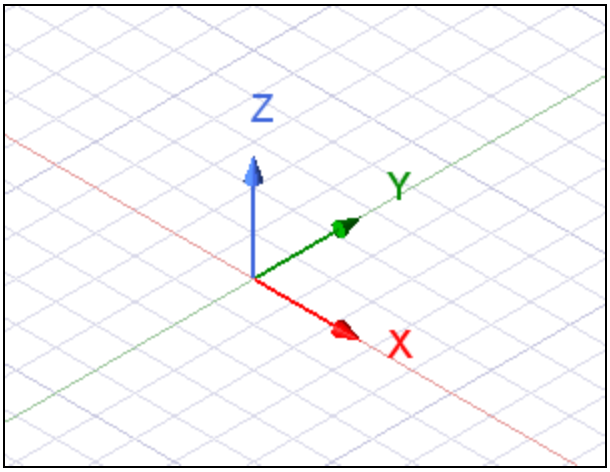
3. To assign a background color that gradually changes from one color to another, do the following:
    a. Select **Gradient Background**.
    b. Specify the background color at the top and bottom of the view window in one of the following ways:
        - Under **Select Background Type**, click the **Top Color** button and select a color from the **Color** palette. Then click **Bottom Color** and select a color from the **Color** palette.
        - Under **Change View Color Dynamically**, click **Top Color** or **Bottom Color** and use the RGB sliders to specify the color's red, green, and blue values.
4. Click **Reset** to revert to the default background colors.
5. Click **Save As Default** if you want the new background color to be the background color for all *3D Modeler* windows in either the current project or future projects.
6. Click **OK**.

The background color you set will be saved with the design. New background color settings assigned to other designs after this point, including new default settings, will not affect this design.

Modifying the Coordinate System Axes View

The coordinate system axes displays the x, y, z orientation from the origin point for the current working coordinate system. The axes use an XYZ = RGB coloring scheme.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By option, you can also use a triad which is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system.

What do you want to do?

- Show or hide the coordinate system axes
- Show the coordinate system axes for selected objects.
- Enlarge or shrink the size of the coordinate system axes
- Show or hide the triad axes

Showing or Hiding the Axes

1. Click **View** > **Coordinate System**, then click one of the following:
   - **Hide** to hide the x-, y-, and z-axes in the active view window.
   - **Show** to display the x-, y-, and z-axes in the active view window.

Show the Axes for Selected Objects

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

   This command displays the *Options* dialog box.
2. Expand the **3D Modeler** branch of the options tree.
3. Expand the **Display** sub-branch.
4. Select the **General** subgroup.
5. Select **Show orientation of selected objects**.
6. Click **OK** to close the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enlarging or Shrinking the Axes

1. Click **View**> **Coordinate System**, then click one of the following:
    - **Large** to display the x-, y-, and z-axes as extending to the edges of the active view window.
    - **Small** to display the x-, y-, and z-axes in a smaller size in relative to the edges of the active view window.

Showing or Hiding the Triad Axes

The triad is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system. It can be shown or hidden separately from the selected coordinate system.

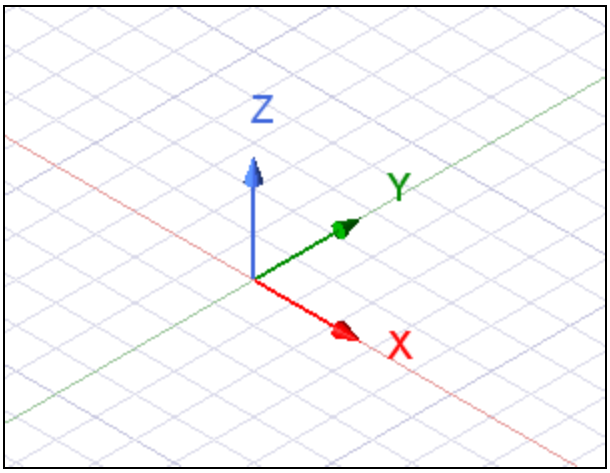
To show the triad:

1. Click **View**> **Coordinate System**> **Triad**, then click one of the following:
    - **Hide** to hide the triad x-, y-, and z-axes at the lower right of the active view window.
    - **Show** to display the triad x-, y-, and z-axes in the lower right active view window.
    - **Auto** to generally hide the triad axes.

Choosing Grid Settings

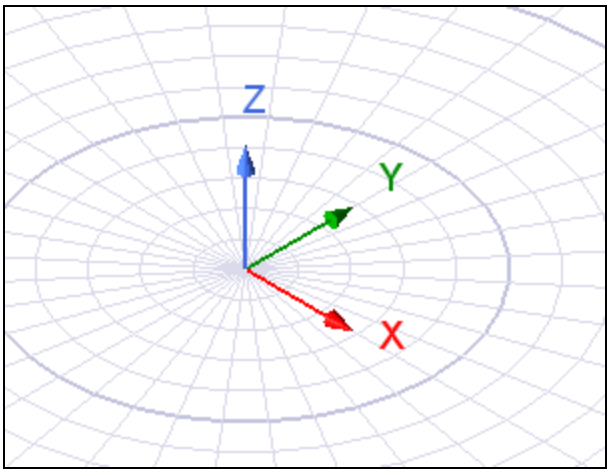
The grid displayed in the **Modeler** window is a drawing aid that helps to visualize the location of objects.

For Cartesian grids, the location of points on the grid are defined by intersections of planes that are perpendicular to and along the x-, y-, and z-axes. The division (the distance between neighboring parallel planes perpendicular to the same axis) can be set.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For polar grids, the location of points on the grid are defined by intersections of planes that are perpendicular to the local radius and angle coordinates. The division (the distance between neighboring parallel planes perpendicular to the same radius and angle) can be set.



Grid spacing is set according to the current project's drawing units. You can control the following aspects of the grid:

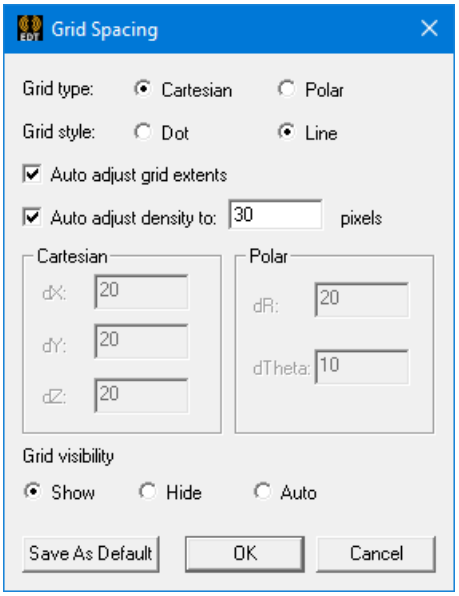
- Type (rectangular or circular)
- Style (dots or lines)
- Grid Extent (minimal for existing objects or as a plane)
- Density
- Spacing
- Visibility
- Snap settings
- Grid plane

Setting the Grid Type

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a grid type for the active Modeler window: **Cartesian** for a rectangular grid or **Polar** for a circular grid.

   The grid in the active view window is centered at the origin of the working coordinate system.

For Cartesian grids, you define a coordinate by specifying its distance from the origin along each axis in the **X**, **Y**, and **Z** text boxes or its relative distance from the previously selected point in the **dX**, **dY**, and **dZ** text boxes.

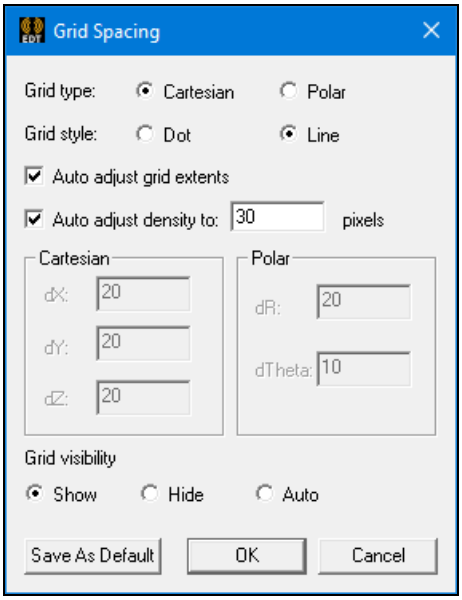
For polar grids, you define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

Setting the Grid Style

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select one of the following grid styles for the active view window:

   Dot Displays each grid point as a dot.

   Line Displays lines between grid points.

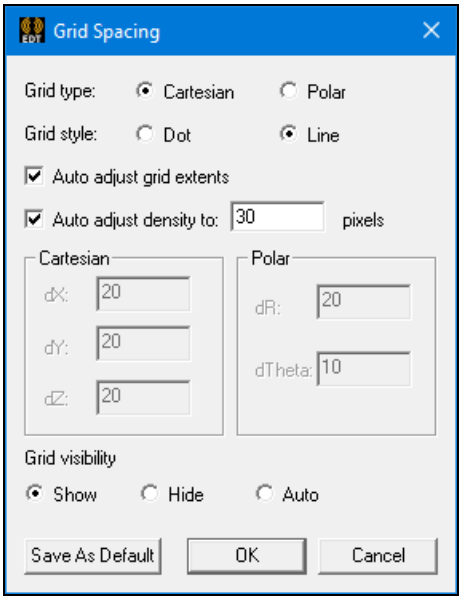
3. To show a Minimal grid around existing objects, check **Auto adjust grid extends**. To display a grid as a plane, deselect **Auto adjust grid extends**.

Setting the Grid Density and Spacing

1. Click **View** > **Grid Settings**.

   The **Grid Settings** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you want to change the density of the grid in the active Modeler window as you zoom in or out on objects, do the following:
    a. Select **Auto adjust density to**.
    b. Specify a distance between grid points by typing a value in the **pixels** box.

    The default is set to 30 pixels, which is generally the best setting for displaying objects.

3. If you do not want the grid density to change when you zoom in or out, but instead want to specify a constant grid spacing, do the following:
    a. Clear the **Auto adjust density to** option.
    b. Specify the grid's spacing in the active design's units.

    If you selected a Cartesian grid, type the values of **dX**, **dY**, and **dZ**. These values represent the difference between one grid point and the next in the x, y, and z directions, respectively.

    If you selected a polar grid, type the values for **dR** and **dTheta**. **dR** represents the difference between each radius. **dTheta** is the difference between angles.

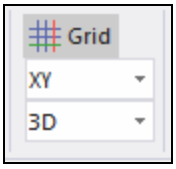
    The distance between grid points will increase and decrease proportionately as you zoom in and out in the active Modeler window.

4. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Grid's Visibility

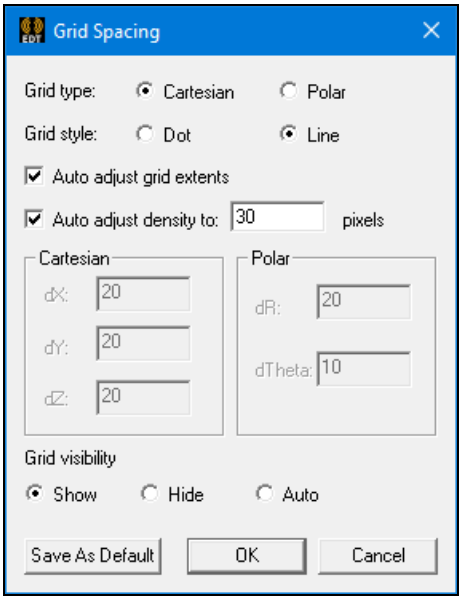
To hide the grid in the Modeler window, on the **Draw** ribbon tab, click the Grid toolbar icon. Click it again to show the grid.



Alternatively:

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



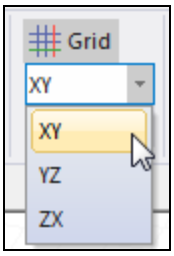
2. Select **Show** to make the grid always visible in the active **Modeler** window. Select **Hide** to make the grid always invisible in the active **Modeler** window. Select **Auto** so the grid appears only while you are drawing an object in the **Modeler** window.

Setting the Grid Plane

To specify the plane on which you want to display the grid in the active view window, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Modeler** > **Grid Plane** and then select a grid plane: **XY**, **YZ**, or **XZ**.
- Click a grid plane on the drop-down menu under Grid on the **Draw** ribbon tab:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14 - Specifying Solution Settings for Mechanical

The solution settings are dependent on the mechanical Solution Type that you specify. Use the appropriate link below for more information:

- ***Modal*** Solution Setup
- ***Thermal*** Solution Setup
- ***Structural*** Solution Setup

Editing an Existing Solution Setup:

To review or change any of the settings for a previously added solution setup, do the following:

- Under *Analysis* in the Project Manager, right-click the setup you wish to modify and choose **Properties** from the shortcut menu.

  Either the *Modal Solve Setup, Thermal Solve Setup,* or *Structural Solve Setup* dialog box will appear, depending on the *Solution Type* of the analysis.

Setup Link Window

For Mechanical designs, the *Setup Link* window is used to import certain types of data from a source design.

For *Modal, Thermal,* and *Structural* solutions, you can import a source design's mesh. See the Mesh Linking topic for more information.

For *Thermal* solutions, you can import electromechanical (EM) losses from several design types. You can also import non-uniform heat transfer film coefficients from an Icepak design (for convection boundaries). Choose either of the following links for more information:

- Importing EM Losses
- Importing Non-Uniform Film Coefficients

Clearing Linked Data

Linked data can be a mesh, field results, or some other post-processing data that the source design generated. The target design for the link caches the data internally to minimize the need to activate the source design.

If you have previously setup links to a design, the **Mechanical**> **Analysis**> **Setup** shortcut menu contains an option to **Clear Linked Data**. This removes the linked data for all links in a design, therefore invalidating the solutions. You can also clear linked data through **Mechanical**>

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results> **Clean Up Solutions**, which displays a dialog box that includes options to let you selectively delete linked data only, or as part of other deletions.

Clearing linked data for some link types requires Mechanical to revert to the initial mesh. Thus in some cases, this command removes the current mesh of the target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - Optimetrics

Optimetrics enables you to determine the best design variation among a model's possible variations. You create the original model, the *nominal design*, and then define the design parameters that vary, which can be nearly any design parameter assigned a numeric value in Mechanical. For example, you can parameterize the model geometry or material properties. You can then perform the following types of analyses on your nominal design:

optiSLang	optiSLang simulations can be integrated with Ansys Electronics Desktop, such that you can create a setup very much like an Optimetrics setup to run through an optiSLang installation.
Parametric	In a parametric analysis, you define one or more *variable sweep definitions,* each specifying a series of variable values within a range. For example, you can parameterize component values. (See Variables for more information.) Optimetrics solves the design at each variation. You can then compare the results to determine how each design variation affects the performance of the design. Parametric analyses are often used as precursors to optimization solutions because they help to determine a reasonable range of variable values for the optimization analysis.
Design of Experiments	Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems.
DesignXplorer	An optimization tool for studying a range of design variations, used with Design of Experiments. This permits you to manage an Optimetrics simulation from the Ansys Workbench.
Optimization	For an optimization analysis, you identify the cost function and the optimization goal. Optimetrics changes the design parameter values to meet that goal. The cost function can be based on any solution quantity that can be computed.
Sensitivity	In a sensitivity analysis, you use Optimetrics to explore the vicinity of the design point to determine the sensitivity of the design to small changes in variables.
Tuning	Tuning allows you to change variable values interactively while monitoring the performance of the design. If you want to ensure that tuning does not resolve variations already solved by parametric setup, you must check **Save Fields Mesh** in the **Options** tab of the Optimetrics setup.
Statistical	In a statistical analysis, you use Optimetrics to determine the distribution of a design's performance, which is caused by a statistical distribution of variable values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Sensitivity, Statistical, Design Xplorer, and some Optimizers have been placed in legacy mode because they are no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Note:

Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

The HPC and Analysis Options dialog can be accessed from the setup dialog for each type of Optimetrics analysis.

Parametric Overview

Running a parametric analysis enables you to simulate several design variations using a single model. You define a series of variable values within a range, or a variable sweep definition, and Mechanical generates a solution for each design variation. You can then compare the results to determine how each design variation affects the performance of the design.

You can vary design parameters that are assigned a quantity, such as geometry dimensions, material properties, and boundary and excitation properties (See the help topic for the specific parameter you want to vary). The number of variations that can be defined in a parametric sweep setup is limited only by your computing resources.

To perform a parametric analysis, you first create a nominal design. A nominal design is created like any other design, except that variables are assigned to those aspects of the model you want to change. You can create a parametric setup before defining variables but all variables must be defined before you start the parametric analysis. Although you are not required to solve the nominal design before performing a parametric analysis, doing so helps ensure that the model is set up and operates as intended. Alternatively, you can perform a validation check on the nominal design before performing a parametric analysis.

Parametric analyses are often used as precursors to optimization analyses because they enable you to determine a reasonable range of variable values within which optimal conditions will occur.

Setting Up a Parametric Analysis

A *parametric setup* specifies all of the design variations that Optimetrics drives the software to solve. A parametric setup is made up of one or more variable sweep definitions, which are a set

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

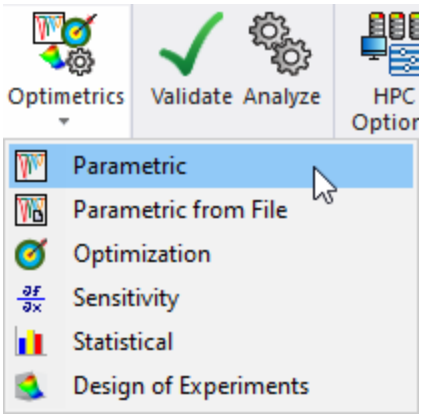
of variable values within a range that you want the software to solve when you run the parametric setup.

You can define more than one parametric setup per design.

> **Note:**
>
> Once you have created a parametric setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To add a parametric setup to a design:

1. Click **Mechanical**> **Optimetrics Analysis**> **Add Parametric** .
   - Alternatively, right-click **Optimetrics** in the Project Manager and then click **Add** > **Parametric** on the shortcut menu.
   - Select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down menu under the Optimetrics icon:



   The **Setup Sweep Analysis** dialog box appears.

2. Add a variable sweep definition.

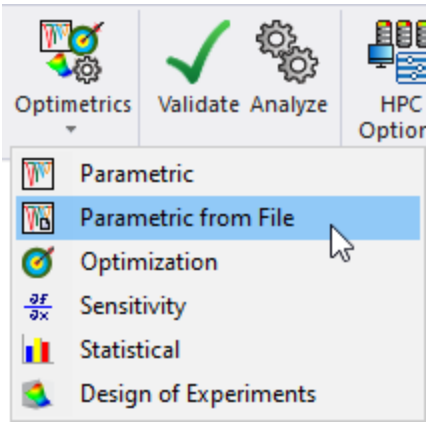
   After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name in the Project Manager.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Parametric Sweep from a File

You can specify the parameters for a parametric sweep in a spreadsheet that uses either a .csv (comma delimited) or .txt (tab delimited) format. You can then import the parametric sweep using the **Mechanical** > **Optimetrics Analysis** > **Add Parametric from File** command, or with the **Simulation** tab selected, use the drop-down menu under the Optimetrics icon and select **Parametric from File**:



These methods open a file browser for a comma delimited file (.csv) or a tab delimited .txt file.

For example, a .txt spreadsheet file could resemble the following:

```
a $b $c[in] d[m] $e $f
0.1 mil 2mm 11 21 0.6in 8
0.2mil 3mm 1.3 2.6mm 3 9cm
...
```

The first row lists the Project and Design Variable names, and when followed by parentheses, the units. When unit not present, SI unit is assumed. The following rows provide the variable values and units, where SI is assumed if not specified. Project or Design variables must be defined before they are accepted from a file. The characters in variable names are not case sensitive. Consecutive separators are treated as one separator.

The header row also takes units in ( ) as well as the conventional [ ].

After you have imported a valid file, the detailed information regarding sweep values can be found at the lower portion of the Parametric **Setup Sweep Analysis** dialog**Sweep Definitions** tab (Operation/Description panel). It is treated as: first row as single point. then each time a new sweep point is added.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Variable Sweep Definition

A parametric setup is made up of one or more *variable sweep definitions*. A variable sweep definition is a set of variable values within a range that Optimetrics drives to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

> **Note:**
>
> Sweeping a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

1. Click **Mechanical** > **Optimetrics Analysis** > **Add Parametric** .
   - Alternatively, right-click **Optimetrics** in the project tree, and then click **Add** > **Parametric** on the shortcut menu.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.

   All the independent variables associated with the design are listed in the **Variable** drop-down menu of the **Add/Edit Sweep** dialog.
3. Click the variable for which you are defining the sweep definition from the **Variable** drop-down menu.

   If you do not define a sweep definition for a variable in the list, the variable's current value in the nominal design is used in the parametric analysis.
4. Specify the variable values to be included in the sweep.
5. Click **Add** and then click **OK**.

   You return to the **Setup Sweep Analysis** dialog box. The variable sweep is listed in the top half of the window.
6. View the design variations that are to be solved in table format under the **Table** tab. Viewing the sweep definition in table format enables you to visualize the design variations that are to be solved and manually adjust sweep points if necessary.
7. Click **OK**.

Specifying Variable Values for a Sweep Definition

To specify the variable values to include in a sweep definition:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select one of the following in the **Add/Edit Sweep** dialog box:

Single value	Specify a single value for the sweep definition.
Linear step	Specify a linear range of values with a constant step size.
Linear count	Specify a linear range of values and the number, or count of points within this range.
Decade count	Specify a logarithmic (base 10) series of values, and the number of values to calculate in each decade.
Octave count	Specify a logarithmic (base 2) series of values, and the number of values to calculate in each octave.
Exponential count	Specify an exponential (base e) series of values, and the number of values to calculate.

2. If you selected **Single value**, type the value of the sweep definition in the **Value** box.

   If you selected another sweep type, do the following:

   a. Type the starting value of the variable range in the **Start** text box.
   b. Type the final value of the variable range in the **Stop** text box.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

3. If you selected **Linear step** as the sweep type, type the step size in the **Step** box.

   The step size is the difference between variable values in the sweep definition. The step size determines the number of design variations between the start and stop value. The model is solved at each step in the specified range, including the start and stop values. The step size can be negative, when the **Stop** value is less than the **Start** value

   If you selected another sweep type, type the number of points, or variable values, in the sweep definition in the **Count** text box. For **Decade count** and **Octave count**, the **Count** value specifies the number of points to calculate in every decade or octave. For **Exponential count**, the **Count** value is the total number of points. The total number of points includes the start and stop values.

Synchronizing Variable Sweep Definitions

By default, variable sweep definitions are nested. Alternatively, you can synchronize the variable sweep definitions if they have the same number of sweep points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if you synchronize a sweep definition that includes values of 1, 2, and 3 inches with a second sweep definition that includes values of 4, 5, and 6 inches, 3 design variations are solved. The first variation is solved at the variable values of 1 and 4; the second variation is solved at the variable values 2 and 5; and the third variation is solved at the final variable values 3 and 6.

To synchronize variable sweep definitions:

1. Under the **Sweep Definitions** tab of the **Setup Sweep Analysis** dialog box, select the rows containing the sweep definitions you want to synchronize.
2. Click **Sync**.

   The synchronized sweeps are given a group number, which is listed in the **Sync #** column.

   Optionally, view the design variations that are to be solved in table format under the **Table** tab.

Creating Parametric Animations

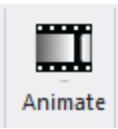
Parametric animations can be based on parametric sweeps. Following is the general procedure for creating an animation of a plot based a parametric sweep.

Prerequisites

- For animation of a field plot/overlay plot over a geometric variable, you must define and solve at least one optimetric sweep with same variable values and Save Fields selected before you can create a parametric animation of a field solution. See Creating Geometry Animations.
- For animation of geometry and VRT plots, solving parametric setup is not required.
- Animations can be based on time variables and objects whose location is linked to the time variable. See the animated example below.

Procedure:

1. Right-click in the **Modeler** window, and click **View**> **Animate...**, or select the **View** tab of the ribbon, and click the **Animate** icon.

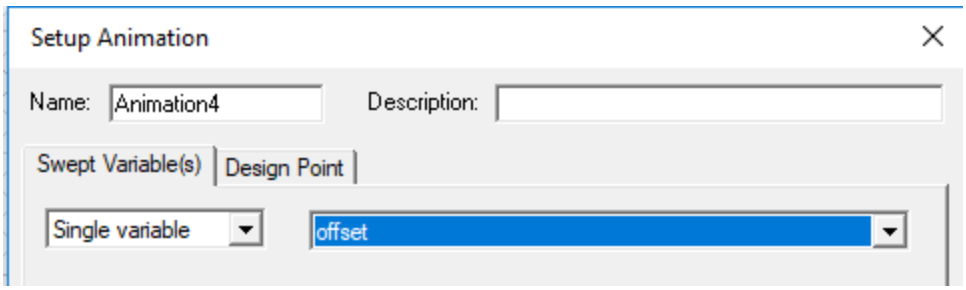
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the *Select Drawing* dialog box:
   a. Select the geometry variable to vary in the animation.
   b. Select the object you want to animate.

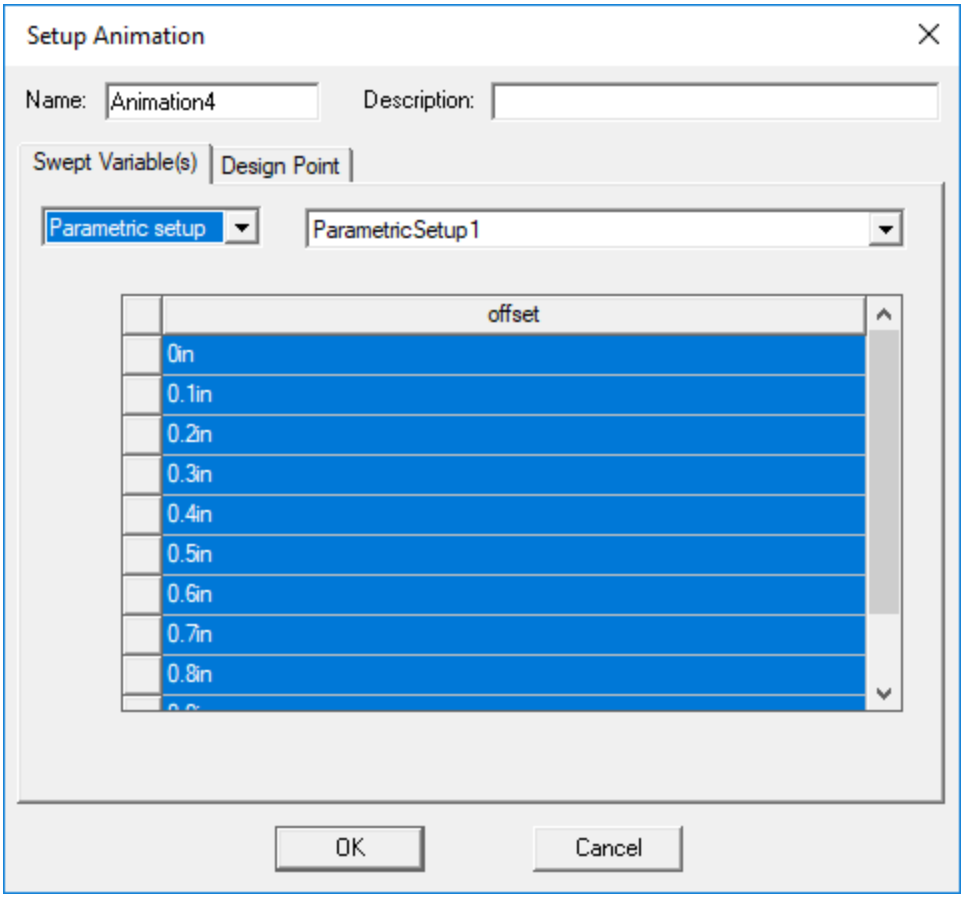
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

The *Setup Animation* dialog box appears.



3. If your have created one or more Parametric Sweeps for one or more geometric variables, you can select **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

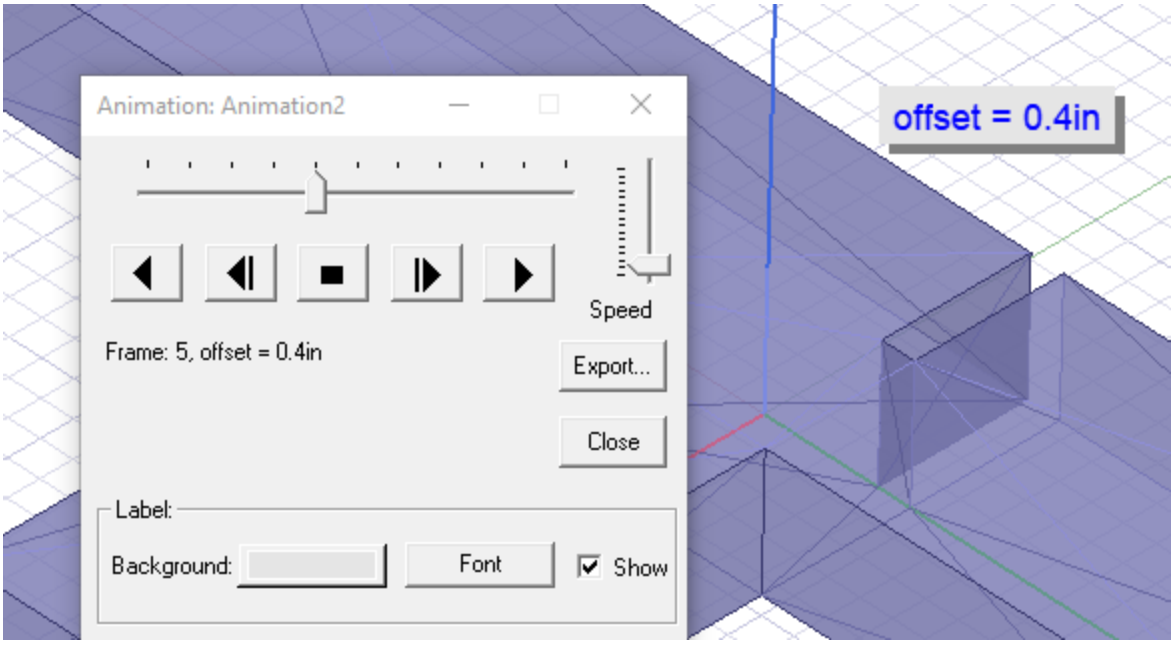


4. Click **OK**.

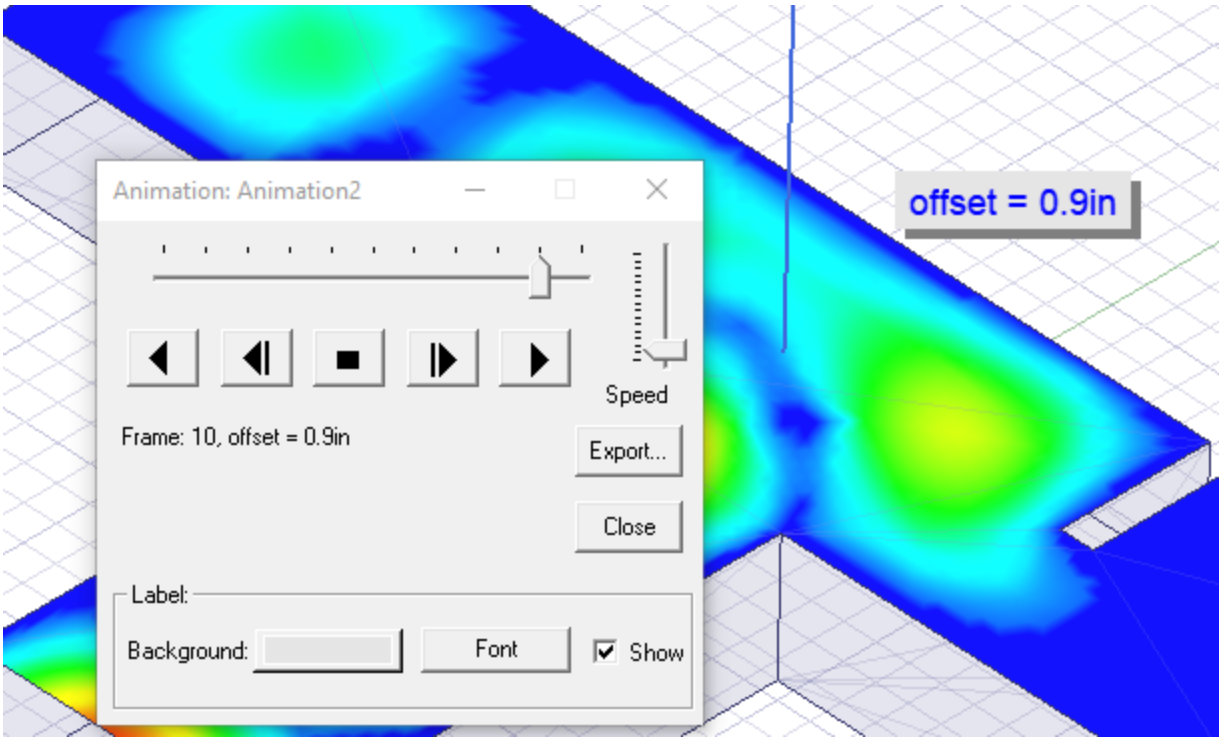
   The animation begins in the view window. It will display one frame for each variable value.

   The **Animation** control panel lets you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While a parametric animation is running, you can also turn on other field plots.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying a Variable Sweep Definition Manually

You can manually modify the variable values that are solved for a parametric setup by explicitly changing, adding, or deleting existing points in a variable sweep definition under the **Table** tab of the **Setup Sweep Analysis** dialog box.

To manually modify a variable sweep definition:

1. Click the **Table** tab of the **Setup Sweep Analysis** dialog box.

   The design variations that will be solved for the parametric setup are listed in table format.

2. Do one of the following:
   - To modify a variable value, click a value text box in the table and type a new value.
   - To delete a variable value from the sweep definition, click the row you want to delete, and then click **Delete**.
   - To add a new variable value to the sweep definition, click **Add**. Then click in the value text box and type a new value.

     > **Warning:**
     >
     > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Your modifications are tracked and available for viewing at the bottom of the **Setup Sweep Analysis** dialog box under the **Sweep Definitions** tab. The operations you performed are listed with descriptions.

> **Warning:**
>
> If you modify an original sweep definition using the **Add/Edit Sweep** dialog box after you have manually modified its table of design variations, your manual modifications become invalid and are removed. A warning is displayed to inform you that your manual values are about to become invalid, so you can decide whether or not to proceed.

Overriding a Variable's Current Value in a Parametric Setup

If you choose not to sweep a variable, the variable's current value set for the nominal design is used when it solves the parametric setup. To override the current variable value for a parametric setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.

   Under **Starting Point**, all of the current independent design variable values are listed.

2. Click the **Override** box of the design variable with the value you want to override for the parametric setup.
3. Type a new value in the **Value** box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for the parametric setup. For this parametric setup, the new value will override the current value in the nominal design.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new variable value in the **Value** box.

4. Optionally, click a new unit in the **Units** box.

To revert to the current variable value, clear the **Override** option.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Specifying a Solution Setup for a Parametric Setup

To specify the solution setup that the software analyzes when it solves a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.
2. Select the solution setup you want the software to use when it solves the parametric setup.

   The parametric setup is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Specifying the Solution Quantity to Evaluate for Parametric Analysis

When you add a parametric setup, you can identify one or more solution quantities to be presented in the *Post Analysis Display* dialog box. The solution quantities are specified by mathematical expressions that are composed of basic quantities, such as output variables. When you view the results, Mechanical extracts the solution quantities and lists them in the results table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the *Setup Sweep Analysis* dialog box, click the **Calculations** tab.

   This displays a table that will show Solutions and associated Calculations. Below the table, are control buttons to **Setup Calculations...** and **Delete**.

2. Click **Setup Calculations**.

   This displays the *Add/Edit Calculation* dialog box. The dialog contains panes to set the **Context**, the **Trace** tab for the **Calculation Expression**, and the **Calculation Range** tab for the **Calculation Range**.

   Follow the procedure to Setup Calculations for Optimetrics.

3. Click **Add Calculation** to add the expression in the *Add/Edit Calculation* dialog box's **Calculation Expression** field to the Calculations tab of the *Setup Sweep Analysis* dialog box.
4. Click **Done** to close the *Add/Edit Calculation* dialog box.

Setting up Calculations for Optimetrics

The **Setup** dialog boxes for each of the Optimetrics types include a **Setup Calculations** button. Clicking this button displays the **Add/Edit Calculation** dialog box, which contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**.

The **Context** pane contains fields for the Report Type to use, the Solution, and depending on the Report Type selection, the Geometry.

The **Trace** tab contains fields for the Calculation expression and, to build the expression, a Category list, a Quantity list with a Text Filter field, and a list of Functions available for the selected Category. The Range function button opens a dialog box in which you can define a range function to apply a function to the expression.

The Category list for the **Trace** tab includes Variables and Output Variables. An Output Variables... button lets you open a dialog box to define and edit the Output Variables.

To set up an Optimetrics calculation:

1. Click **Setup Calculations** to open the **Add/Edit/Calculation** dialog box.
2. In the **Report Type** text field in the **Context** pane, select from the drop-down menu of available types.

   Selecting Fields as the Report type causes the **Geometry** field to display.

3. In the **Solution** text box, select from the drop-down menu of available solutions.
4. If the **Geometry** field is available, select from the drop-down menu.
5. In the **Trace** tab, specify the solution Category, a Quantity, and Functions. The resulting expression will be displayed in the *Calculation Expression* field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Select the **Category** from the list.

   The selection appears in the **Calculation Expression** field, and the Quantity and Function fields list what is available for the corresponding selection.

b. Select the **Quantity** from the list.

   The selected quantity appears in the **Calculation Expression** field.

   If the **Quantity** list is long, you can filter it for easier selection by typing in the text filter field. Only quantities that contain those alphanumeric characters anywhere in their name will remain visible in the list.

   If you want to create an output variable that represents the solution quantity, do the following:

   i. Click **Output Variables**.

      The **Output Variables** dialog box appears.

   ii. Add the expression you want to evaluate, and then click **Done**.

      The recently created output variable appears in the **Quantity** list.

   iii. Click a new output variable in the **Quantity** drop-down menu.

   > **Note:**
   >
   > The calculation you specify must be able to be evaluated into a single, real number.

   The selected Quantity appears in the **Calculation Expression** field.

c. Select the **Function** from the list.

   The selected function is applied to the **Quantity** in the **Calculation Expression** field.

6. To apply a **Range function** to the **Calculation Expression**, see Setting a Range function.
7. Click **Add Calculation** to add the expression in the **Add/Edit Calculation** dialog box's **Calculation Expression** field to the Calculations tab of the **Setup Sweep Analysis** dialog box.
8. Click **Done** to close the **Add/Edit Calculation** dialog box.

Specifying a Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the value of intrinsic variables such as frequency (F) at which the solution quantity will be extracted. For a parametric setup, the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you selected to extract the solution from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you selected to extract the solution quantity from a frequency sweep solution, Optimetrics by default will use the starting frequency in the sweep.

1. In the **Setup Sweep Analysis** dialog box, click the **Calculations** tab.
2. Click the **Setup Calculations** button.

   The **Add/Edit/Calculation** dialog box appears.
3. Select the **Calculation Range** tab.
4. In the **Variable** list, click an intrinsic variable.

   Depending on the variable and Report Type, either a single value or All appears in the Value field.
5. In the **Value** box, click the ellipsis button to display a dialog box that lets you select particular values, or specify a range of values (for Time sweeps). By default, **Use all values** is selected.
6. Click **Update**, and then click **Edit**.

Using Distributed Analysis

If you have purchased the appropriate license, the Electronics Desktop supports distributed solve, which involves distributing rows of a parametric table during Optimetrics solve.

If you do a distributed solve, the Electronics Desktop launches solver engines on multiple machines, assuming that you have configured your HPC and Analysis Options correctly. Also see Large Scale DSO for Parametric Analysis.

To run a distributed analysis:

1. Under **Optimetrics** in the project tree, right-click the specific parametric setup.

   A shortcut menu appears.
2. Select **Analyze** from the shortcut menu.

   > **Note:**
   >
   > After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name.

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

   To show the child progress bars:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click the progress window, and select **Show Subtask Progress Bars**.

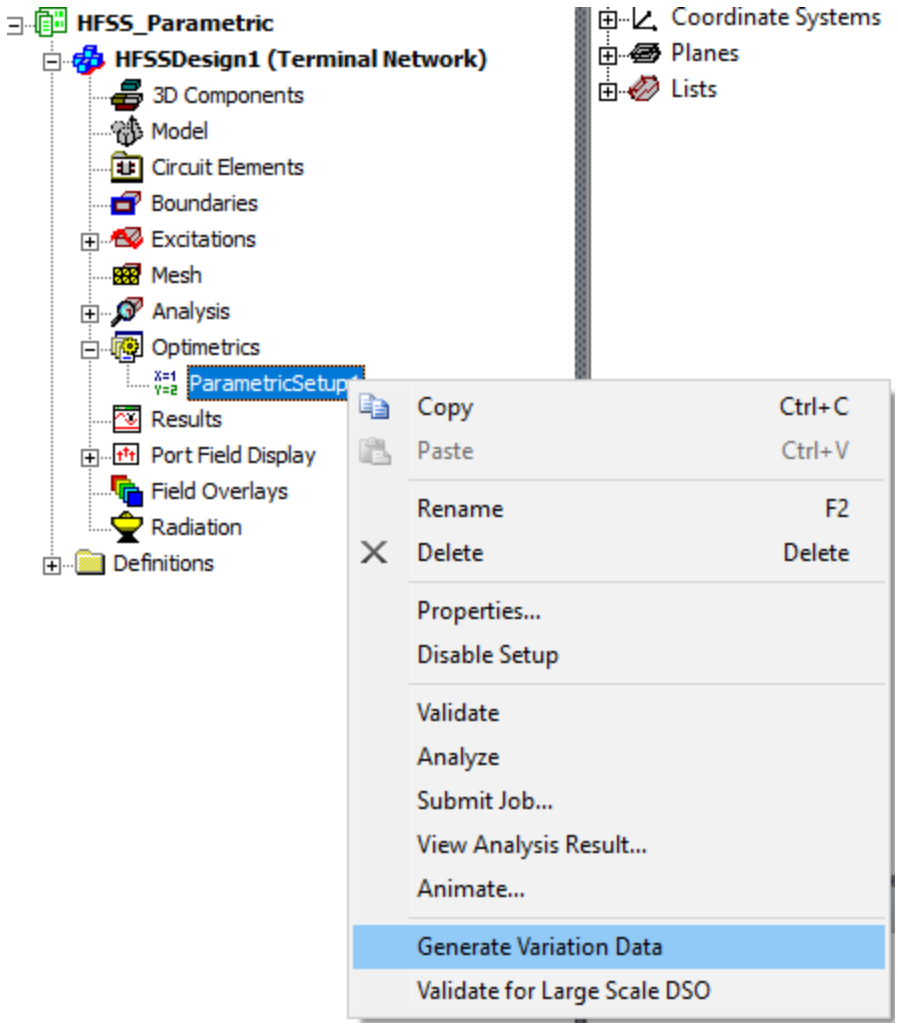
  To hide the child progress bars:

- Right-click the progress window, and select **Hide Subtask Progress Bars**.

Solving a Parametric Analysis on Another Machine

If you would like to run a parametric analysis on another machine, for example, to run a Discovery-linked model on Linux, you can generate the variation data and then export an archive of the project.

To generate the variation data, right-click on the **ParametricSetup** entry in the Project Manager tree, and select **Generate Variation Data**.



When this option is selected, Mechanical will generate geometry for all variations and cache it in the results folder. Users can archive all the data, and then solve on other machines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If the parametric analysis uses a Discovery-linked model, the machine that the archive is moved to does not have to have Discovery installed.

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the parametric setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation. The variation column in the table lists the entries in order. Clicking the Vision header inverts the order. Clicking other headers sorts the entries by value, and clicking again inverts the order.

4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.

5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

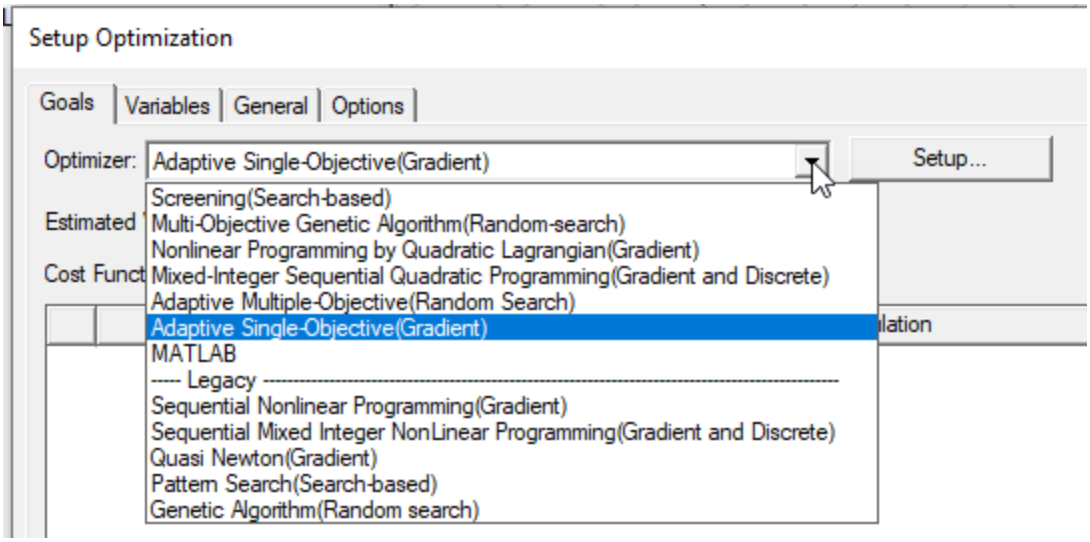
Optimization Overview

Optimetrics interfaces with Ansys Electromagnetics products to help you optimize a wide variety of design parameters based on variable geometry, materials, excitations, component values, etc. Optimization is the process of locating the minimum of a user-defined cost function. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Choosing an Optimizer

Conducting an optimization analysis allows you to determine an optimum solution for your problem. In optimization analyses, there are many available optimizers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Goal Driven Optimizers

These use a Decision Support Process (DSP) based on satisfying criteria as applied to the parameter attributes using a weighted aggregate method. In effect, the DSP can be viewed as a post-processing action on the Pareto fronts as generated from the results of the various optimization methods.

- Screening (Search based) – a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design, which can lead you to apply one of the other approaches for more refined optimization results.
- Multi-Objective Genetic Algorithm – an iterative random search algorithm that can optimize problems with continuous input parameters. It is better for calculating the global optima. You can start with MOGA to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value.
- Nonlinear Programming by Quadratic Lagrangian (Gradient) – a gradient-based, single-objective optimizer based on quasi-Newton methods. Ideally suited for local optimization.
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) – a gradient-based, single-objective optimizer that solves mixed-integer non-linear programming problems by a modified sequential quadratic programming (SQP) method. Ideally suited for local optimization.
- Adaptive Multiple Objective – an iterative, multi-objective optimizer that employs a Kriging response surface and MOGA. In this method, the use of a Kriging response surface allows for a more rapid optimization process because all design points are not evaluated except when necessary and part of the population is simulated by evaluations of the Kriging

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

response surface, which is constructed of all design points submitted by Multi-Objective Genetic Algorithm (MOGA).

- Adaptive Single Objective (Gradient) – a gradient-based, single-objective optimizer that employs an OSF (Optimal Space-Filling) DOE, a Kriging response surface, and MISQP.
- MATLAB

All optimizers assume that the nominal problem you are analyzing is close to the optimal solution; therefore, you must specify a domain that contains the region in which you expect to reach the optimum value.

All optimizers allow you to define a maximum limit to the number of iterations to be executed. This prevents you from consuming your remaining computing resources and allows you to analyze the obtained solutions. From this reduced range, you can further narrow the domain of the problem and regenerate the solutions.

All optimizers also allow you to enter a coefficient in the **Add Constraints** window to define the linear relationship between the selected variables and the entered constraint value. For the SNLP and SMINLP optimizers, the relationship can be linear or nonlinear. For the Quasi Newton and Pattern Search optimizers, the relationship must be linear.

Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Legacy Optimizers

These include

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi-Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random Search)

Adaptive Single Objective Optimization

Adaptive Single-Objective (ASO) is a mathematical optimization method that combines an OSF (Optimal Space-Filling) DOE, a Kriging response surface, and MISQP. It is a gradient-based algorithm based on a response surface, which provides a refined, global, optimized result.

ASO supports a single objective and multiple constraints. It is available for continuous parameters, including those with manufacturable values. It does not support the use of parameter relationships in the optimization domain and is available only for a Direct Optimization system.

Like MISQP, ASO solves constrained nonlinear programming problems of the form:

Minimize:

$$f = f(\{x\})$$

Subject to:

$$g_k(\{x\}) \leq 0, \forall k = 1,2,\ldots,K$$

$$h_l(\{x\}) = 0, \forall I = 1,2\ldots,L$$

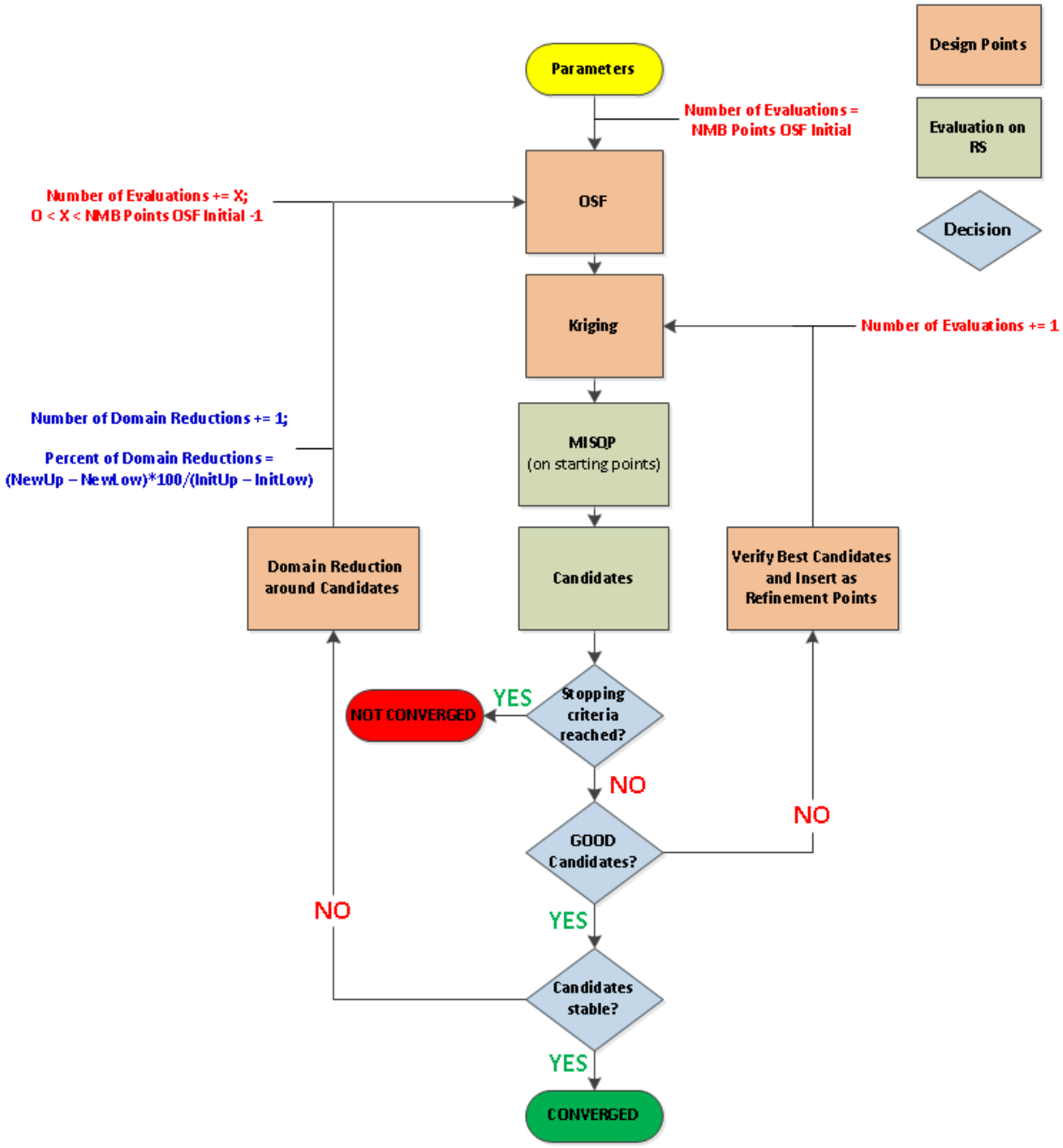
Where:

$$\{x_L\} \leq \{x\} \leq \{x_U\}$$

ASO Workflow

The workflow of ASO follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

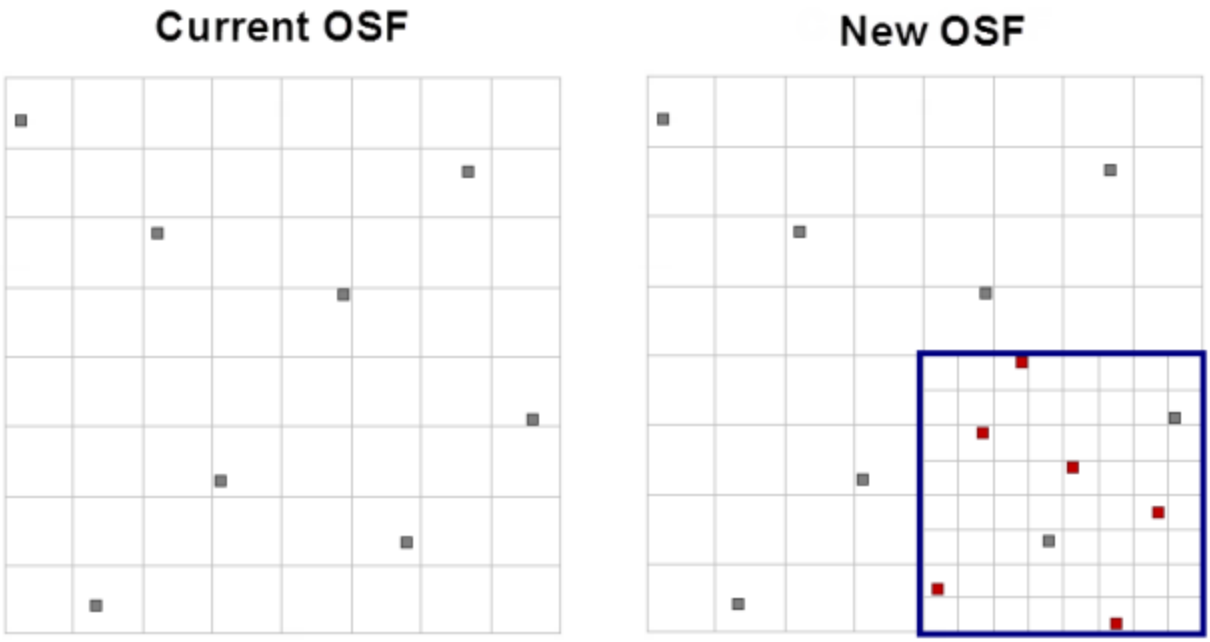
AMO Steps

1. **OSF Sampling**

   OSF (Optimal Space-Filling Design) is used for the Kriging construction. In the original OSF, the number of samples equals the number of divisions per axis and there is one sample in each division.

   When a new OSF is generated after a domain reduction, the reduced OSF has the same number of divisions as the original and keeps the existing design points within the new bounds. New design points are added until there is a point in each division of the reduced domain.

   In the following example, the original domain has eight divisions per axis and contains eight design points. The reduced domain also has eight divisions per axis and includes two of the original design points. To have a design point in each division, six new design points need to be added.



2. **Kirging Generation**

   A response surface is created for each output, based on the current OSF and consequently on the current domain bounds.

   A Kriging response surface is created for each output, based on the first population and then improved during simulation with the addition of new design points.

   For more information on Kriging algorithms, see Kriging Algorithms.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. **MISQP**

   MISQP is run on the current Kriging response surface to find potential candidates. A few MISQP processes are run at the same time, beginning with different starting points, and consequently, giving different candidates.

4. **Candidate Point Validation**

   All the obtained candidates are either validated or not, based on the Kriging error predictor. The candidate point is checked to see if further refinement of the Kriging surface changes the selection of this point. A candidate is considered as acceptable if there aren't any points, according to this error prediction, that call it into question. If the quality of the candidate is not called into question, the domain bounds are reduced. Otherwise, the candidate is calculated as a verification point.

   - **Refinement Point Creation** (If the selection is ***not*** to be changed)

     When a new verification point is calculated, it is inserted in the current Kriging response surface as a refinement point and the MISQP process is restarted.

   - **Domain Reduction**(If the selection is to be changed)

     When candidates are validated, new domain bounds must be calculated. If all of the candidates are in the same zone, the bounds are reduced, centered on the candidates. Otherwise, the bounds are reduced as an inclusive box of all candidates. At each domain reduction, a new OSF is generated (conserving design points between the new bounds) and a new Kriging response surface is generated based on this new OSF.

5. **Convergence and Stop Criteria**

   The optimization is considered to be converged when the candidates found are stable. This occurs when all of the MISQP processes run on the response surface converge to the same verified candidate point. However, there are four stop criteria that can stop the algorithm: Maximum Number of Evaluations, Maximum Number of Domain Reductions, Percentage of Domain Reductions, and Convergence Tolerance.

Setting Up Adaptive Single-Objective (Gradient) Optimizer

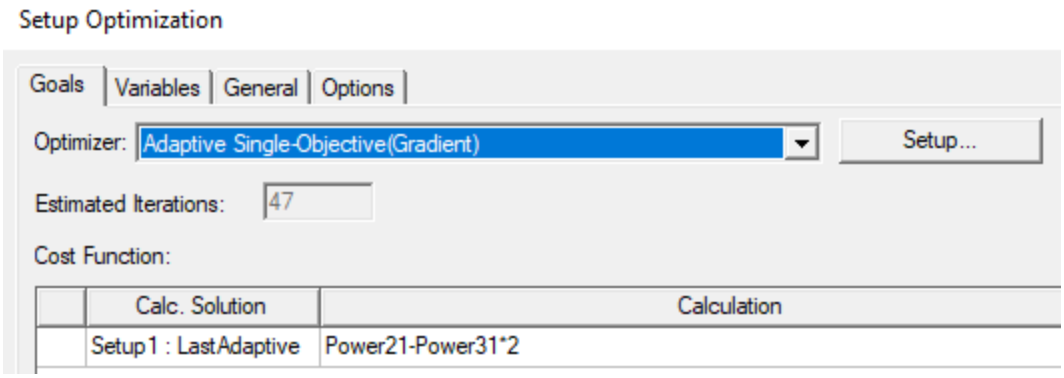
Following is the procedure for setting up an optimization analysis using the Adaptive Single-Objective (OSF + Kriging + MISQP) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

This gradient-based method employs automatic intelligent refinement to provide the global optima. It requires a minimum number of design points to build the Kriging response surface, but, in general, this method reduces the number of design points necessary for the optimization. Failed design points are treated as inequality constraints, making it fault-tolerant.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

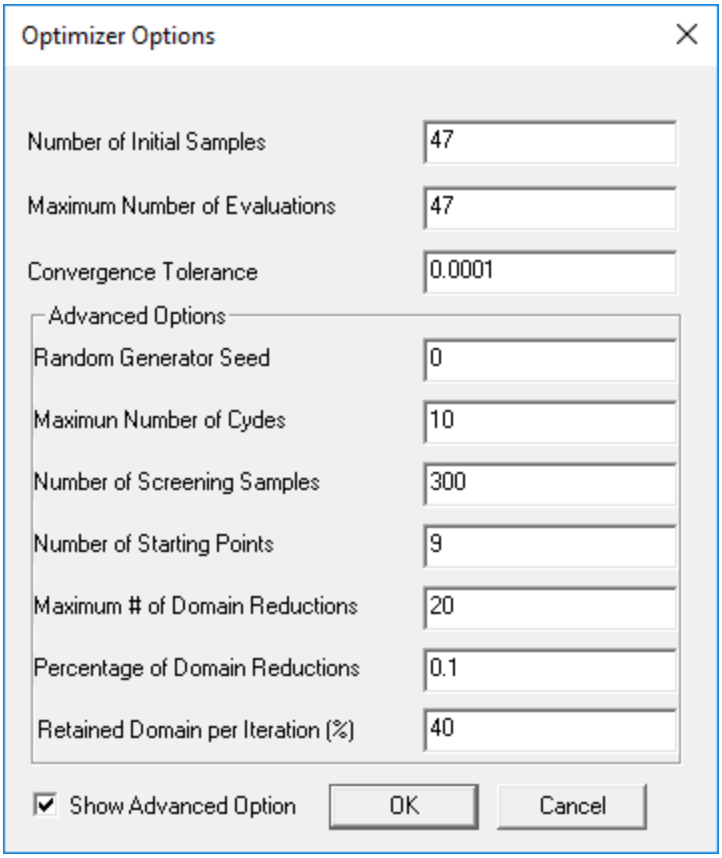
The Adaptive Single-Objective method is available for input parameters that are continuous, including those with manufacturable values. It can handle only one output parameter goal, although other output parameters can be defined as constraints. It does not support the use of parameter relationships in the optimization domain. For more information, see Adaptive Single-Objective Optimization (ASO). It requires advanced options. Ensure that the Show Advanced Options check box is selected.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, choose the optimizer by selecting **Adaptive Single Objective (Gradient)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window and check **Advanced Options**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Number of Initial Samples**: Number of samples generated for the initial Kriging and after all domain reductions for the construction of the next Kriging. You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is (NbInp+1)*(NbInp+2)/2.

  Because of the Adaptive Single-Objective workflow (in which a new OSF sample set is generated after each domain reduction), increasing the number of OSF samples does not necessarily improve the quality of the results and significantly increases the number of evaluations

- **Maximum Number of Evaluations**: Stop criterion. Maximum number of evaluations (design points) that the algorithm is to calculate. If convergence occurs before this number is reached, evaluations stop. This value also provides an idea of the maximum possible time it takes to run the optimization. The default is 20*(NbInp +1).
- **Convergence Tolerance**: Stop criterion. Minimum allowable gap between the values of two successive candidates. If the difference between two successive candidates is smaller than the value for Convergence Tolerance multiplied by the maximum variation of the parameter, the algorithm is stopped. A smaller value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. The default is 1E-06.

- **Random Generator Seed**: The value for initializing the random number generator invoked internally by OSF. The value must be a positive integer. This property allows you to generate different samplings by changing the value or to regenerate the same sampling by keeping the same value. The default is 0.
- **Maximum Number of Cycles**: Number of optimization loops that the algorithm needs, which in turns determines the discrepancy of the OSF. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the OSF smaller. The value must be greater than 0. For practical purposes, 10 cycles is usually good for up to 20 variables. The default is 10.
- **Number of Screening Samples**: Number of samples for the screening generation on the current Kriging. This value is used to create the next Kriging (based on error prediction) and verified candidates.

  You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is 100*NbInp for a Direct Optimization system. There is no default for a Response Surface Optimization system.

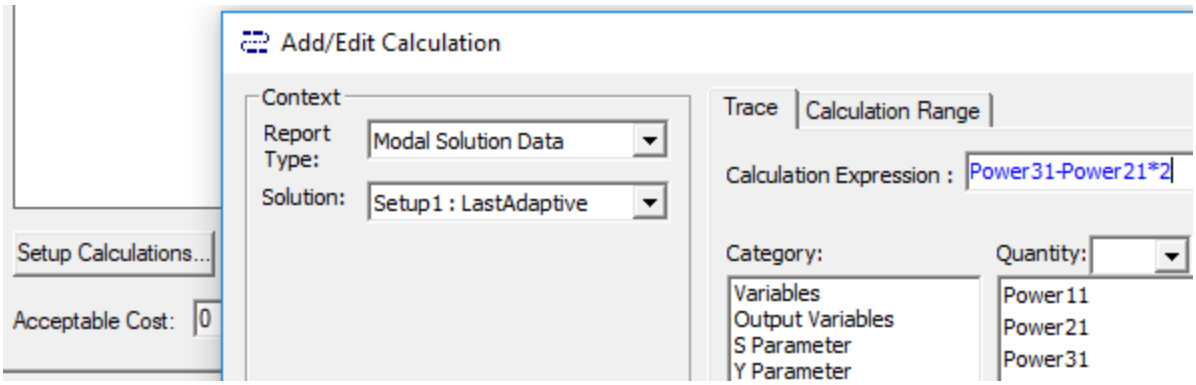
  The larger the screening sample set, the better the chances of finding good verified points. However, too many points can result in a divergence of the Kriging.

- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these samples are selected in this sample. The default is the value for **Number of Initial Samples**.
- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according to the initial domain. For example, with one input ranging between 0 and 100, the domain size is equal to 100. The percentage of domain reduction is 1%, so the current working domain size cannot be less than 1 (such as an input ranging between 5 and 6). The default is 0.1.
- **Retained Domain per Iteration (%)**: Advanced option that allows you to specify the minimum percentage of the domain you want to keep after a domain reduction. The percentage value must be between 10 and 90. A larger value indicates less domain reduction, which implies better exploration but a slower solution. A smaller

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

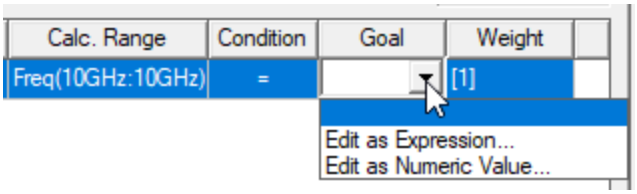
value indicates a faster and more accurate solution, with the risk of it being a local one. The default percentage value is 40.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

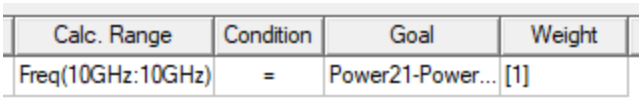


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A norm is a function that assigns a positive value to the cost function.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adaptive Multiple Objective Optimization

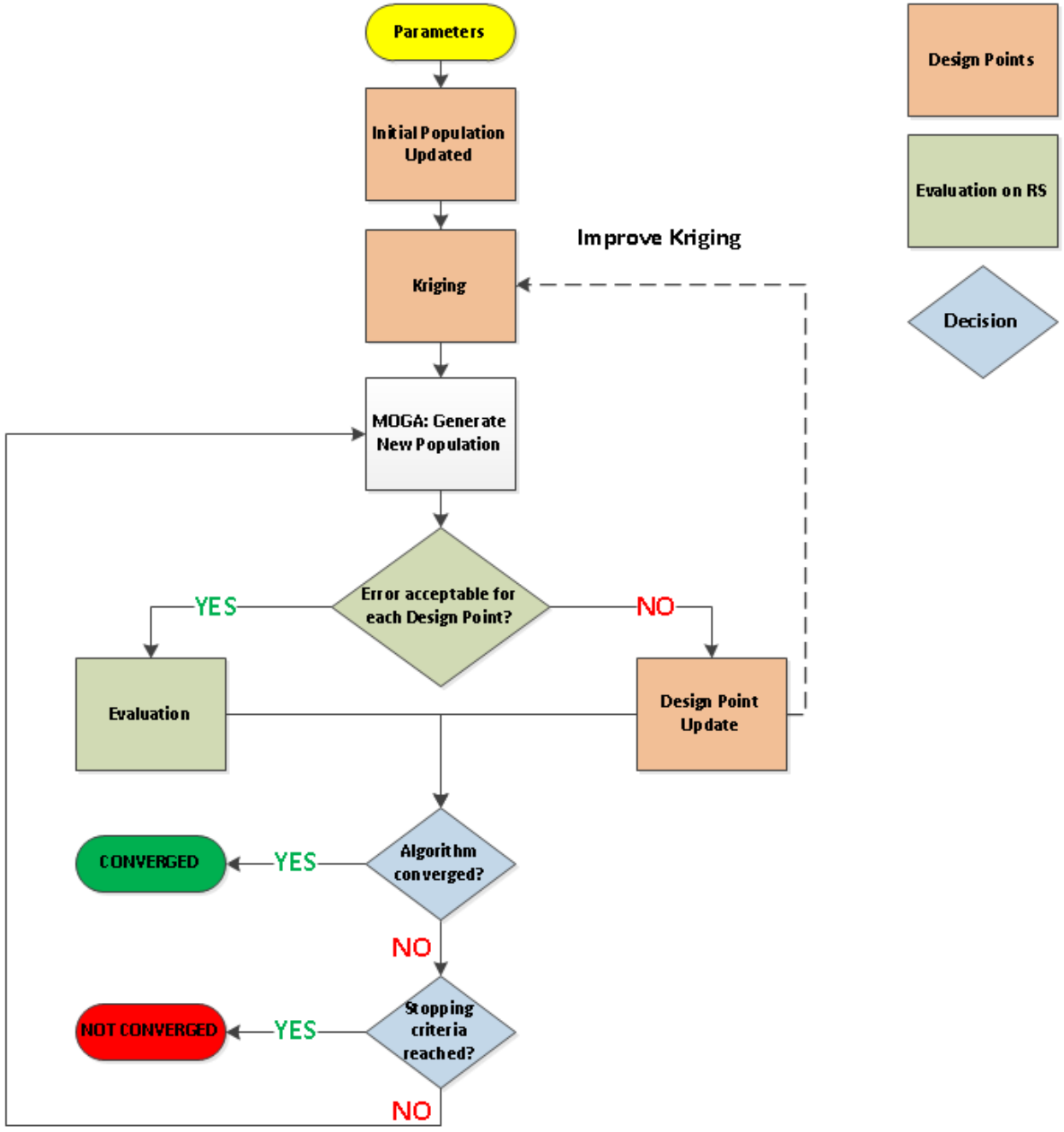
Adaptive Multiple-Objective (AMO) is a mathematical optimization that combines a Kriging response surface and MOGA. It allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. Except when necessary, the optimizer does not evaluate all design points. The general optimization approach is the same as MOGA, but a Kriging response surface is used. Part of the population is "simulated" by evaluations of the Kriging response surface. The Kriging error predictor reduces the number of evaluations used in finding the first Pareto front solutions.

AMO supports multiple objectives and multiple constraints. It is limited to continuous parameters, including those with manufacturable values. It is available only for a Direct Optimization system. When discrete parameters are used, MOGA is the more efficient optimization method. For more information, see Multi-Objective Genetic Algorithm (MOGA).

AMO Workflow

The workflow of AMO follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



AMO Steps

1. **First Population of MOGA**

   The initial population is used to run MOGA.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. **Kirging Generation**

   A Kriging response surface is created for each output, based on the first population and then improved during simulation with the addition of new design points.

3. **MOGA**

   MOGA is run, using the Kriging response surface as an evaluator. After the first iteration, each population is run when it reaches the number of samples defined by the Number of Samples Per Iteration property.

4. **Evaluate the Population**
5. **Error Check**

   The Kriging error predictor is checked for each point.

   - **Yes: Error Acceptable**

     Each point is validated for error. If the error for a given point is acceptable, the approximated point is included in the next population to be run through MOGA (return to Step 3).

   - **No: Error Not Acceptable**

   If the error is not acceptable, the points are promoted as design points. The new design points are used to improve the Kriging response surface (return to Step 2) and are included in the next population to be run through MOGA (return to Step 3)

6. **Convergence Validation**

   The optimization is validated for convergence.

   - **Yes: Optimization Converged**

     MOGA converges when the maximum allowable Pareto percentage has been reached. When this happens, the process is stopped.

   - **No: Optimization Not Converged**

   If the optimization is not converged, the process continues to the next step.

7. **Stopping Criteria Validation**

   If the optimization has not converged, it is validated for fulfillment of the stopping criteria.

   - **Yes: Stopping Criteria Met**

     When the maximum number of iterations has been reached, the process is stopped without having reached convergence..

   - **No: Stopping Criteria Not Met**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the stopping criteria have not been met, the MOGA algorithm is run again (return to Step 3).

8. **Conclusion**

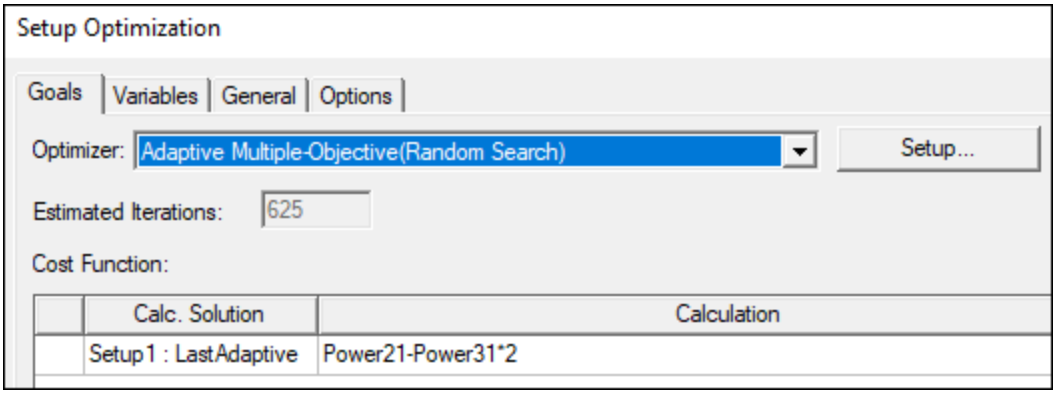
   Steps 2 through 7 are repeated in sequence until the optimization has converged or the stopping criteria have been met. When either of these things occurs, the optimization concludes.

Setting Up Adaptive Multiple-Objective (Random Search) Optimizer

Following is the procedure for setting up an optimization analysis using the Adaptive Multiple Objective (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

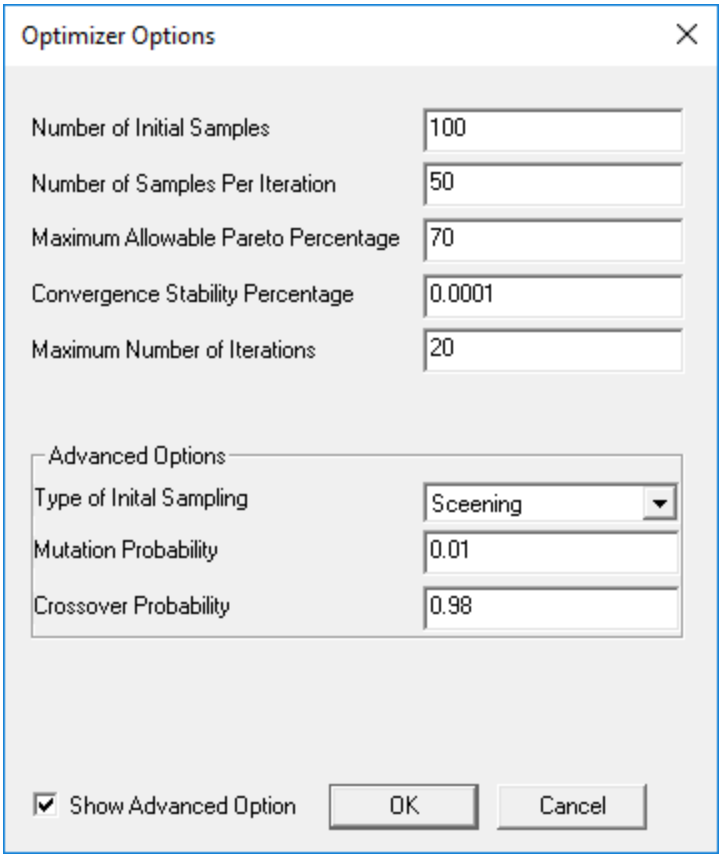
The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce the number of evaluations needed to find the global optimum. The Adaptive Multiple-Objective method is available only for continuous input parameters, including those with manufacturable values. It can handle multiple objectives and multiple constraints. For more information, see Adaptive Multiple-Objective Optimization.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Adaptive Multiple Objective (Random-search)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the *Optimizer Options* window.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

  The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

  If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can enter a minimum of 2 and a maximum of 10000.

- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

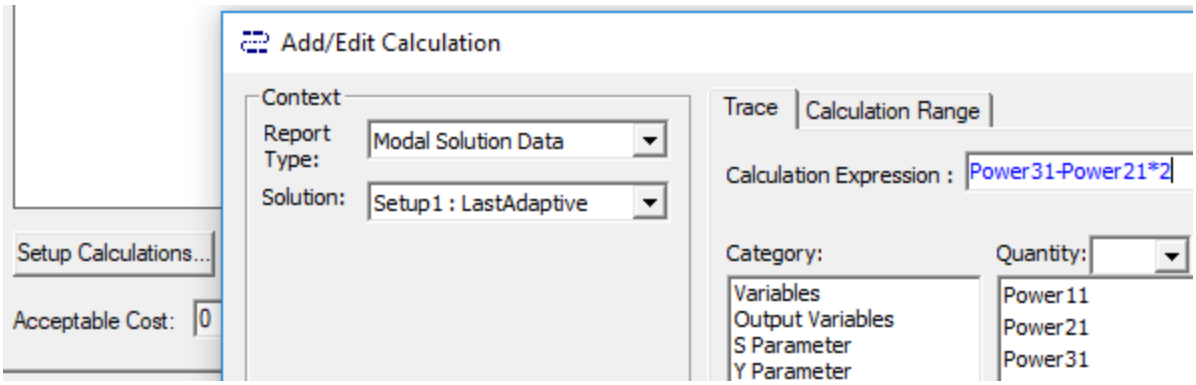
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

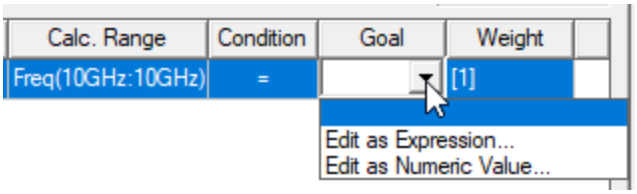
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

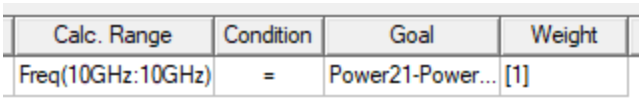


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The *Cost Function Norm Type* pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

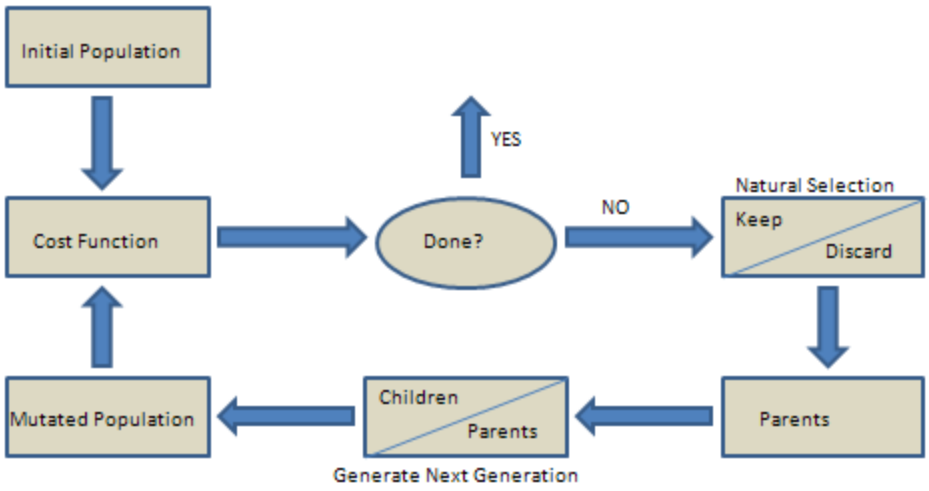
> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Genetic Algorithm (Random Search)

Genetic Algorithm (Random Search) optimizers are part of a class of optimization techniques called stochastic optimizers. They do not use the information from the experiment or the cost function to determine where to further explore the design space. Instead, they use a type of random selection and apply it in a structured manner. The random selection of evaluations to proceed to the next generation has the advantage of allowing the optimizer to jump out of a local minima at the expense of many random solutions which do not provide improvement toward the optimization goal. As a result, the GA optimizer will run many more iterations and may be prohibitively slow.

The Genetic Algorithm search is an iterative process that goes through a number of generations (see picture below). In each generation some new individuals (Children / Number of Individuals) are created and the grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a smaller set of individuals must be created from a bigger set, the GA selects individuals from the original set. During this process, better fit (in relation to the cost function) individuals are preferred. In the elitist selection, simply the best so many individuals are selected, but if you turn on the roulette selection, then the selection process gets relaxed. An iterative process starts selecting the individuals and fills up the resulting set, but instead of selecting the best so many, we use a roulette wheel that has for each selection-candidate divisions made proportional to the fitness level (relative to the cost function) of the candidate. This means that the fitter the individual is, the larger the probability of his survival will be.

Advanced Genetic Algorithm Optimizer Options

The Genetic Algorithm (GA) search for Optimization analysis is an iterative process that goes through a number of generations. In each generation some new individuals (Children / Number of Individuals) are created and the so grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).

If you select the Genetic Algorithm for an Optimization analysis, a **Setup** button is enabled on the **Setup Optimization** page.

1. Click **Setup** to open the *Advanced Genetic Algorithm Optimizer Options* dialog box.
2. Select the Stopping Criteria. Any of the three following, or any combination of these can be selected.
    - **Maximum number of generations**. If checked, this enables a value field.
    - **Elapsed time**. If checked, this enables a drop-down menu with times ranging from five minutes to two weeks.
    - **Slow convergence**.
3. Specify the Parents.

    The first step toward mating is a selection process that determines the participating individuals. Potential parents are selected from the Current Generation. This is a set of individuals that is always a subset of the current generation.

    - **Number of individuals** value field -- specify the number of parents for the optimizer to use. You can set the Number of Individuals to less than or equal to the size of the "Current Generation". One reason to consider fewer parents than the possible maximum is to steer the GA toward improvement by selecting the better portion of the current generation to be able to mate.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the Mating pool.

   The Mating pool is created by selecting randomly from the parents, but with each selection, the parent gets "cloned" so it can be selected again and again.

   - **Number of individuals** field -- specify the number individuals to include in the mating pool.
   - **Reproduction setup**-- this button opens the *Genetic Algorithm Optimizer Reproduction Setup* dialog box.

5. Click **Reproduction setup** for the dialog box to specify the Crossover setup, and the Mutation setup.

   The crossover and mutation operator have different roles: *Crossover* mixes "features" of the parents in a new combination, while *mutation* slightly alters the "features" of the individuals. Both need to be present in a GA. The crossover is a way to discover new combinations while the mutation acts as a local search or fine-tuning step. Mutation also keeps diversity in a population, which is a must for GA.

   The crossover operator has two steps. It first alters the variable values of the parents according to a distribution. This tends to produce one child that looks a lot like one parent, and one child that looks a lot like the other parent. Next, some of the variable values of the two children can be exchanged in order to achieve more variation.

   For crossover there are four possible parameters.

   a. **Individual Crossover Probability** determines, for each pair in the mating pool, the probability that their features will be mixed. Usually, this probability should be close or equal to one. If you set it to less than one, some parents will produce two children which are exact clones of the parents. This means that some children inherit all the features of their parents unchanged.
   b. Parents often have multiple variables. If the parent is a candidate for mixing, the **Variable Crossover Probability** determines, for each variable, the probability of mixing. This is usually set high to ensure that most or all variables mix.
   c. **Variable Exchange Probability**: After the slight change in the variable values has been made, the crossover operation is also able to exchange the values of the variables between the two children that are being constructed. The Variable Exchange Probability governs the likelihood of exchange of any variable.
   d. **Mu** is a general parameter defining the sharpness of the distribution that might be used for the **Variable Crossover Probability**. Mu should be greater than one. There is no theoretical upper limit, but we recommend not exceeding 30.

6. Select one of the four **Crossover types** from the drop-down menu.

   The crossover type selected affects the options available.

   Uniform Individual crossover

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	probability
	Variable crossover probability
One point	Individual crossover probability
Two point	Individual crossover probability
Simulated binary crossover	Individual crossover probability
	Variable crossover probability
	Variable exchange probability
	Mu

7. Select the **Mutation type** – this can be one of three types, which you select from a drop-down menu:
   - **Uniform Distribution**
   - **Gaussian Distribution**
   - **Polynomial Mutation**
8. For the selected mutation type, set the following parameters:
   - **Uniform Mutation Probability**: If this is more than zero (recommendation is to have still a small probability here), then there will be some children whose features are simply a completely random design (design variables randomly selected over the domain).
   - **Individual Mutation Probability** controls, for each child, the likelihood of a mild mutation.
   - **Variable Mutation Probability**. If the child will be mutated, this probability controls at the variable level the likelihood of a mutation of the variables.
   - **Standard Deviation** is the standard deviation of the selected distribution that is being used for the mutation and it is measured relatively to the optimization-domain.
9. When you have completed the Reproduction setup in the *Genetic Algorithm Optimizer Reproduction Setup* dialog box, click **OK** to close it and return to the *Advanced Genetic Algorithm Optimizer Options* dialog box.
10. In the *Advanced Genetic Algorithm Optimizer Options* dialog box, specify the children as a Number of Individuals.
11. Set the **Pareto Front** value.

    This is the number of the very best individuals (identified relative to the cost function) to keep for future generations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Set the Next Generation parameters. The Next Generation is selected from the Parents, the children, and the Pareto front.
    - **Number of individuals** value field -- specify the number of individuals to survive to form the next generation for the optimizer to use.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.
13. Click **OK** to accept the settings for the Genetic Algorithm and to close the dialog box.

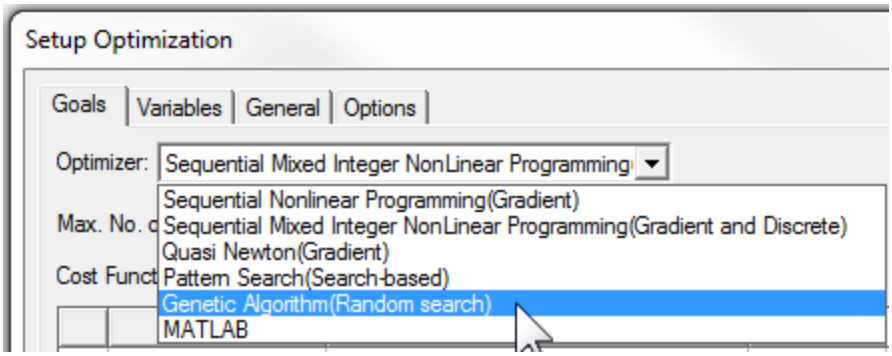
Optimization Setup for the Genetic Algorithm (Random search) Optimizer

Following is the procedure for setting up an optimization analysis using the Genetic Algorithm (Random search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Genetic Algorithm(Random search)** from the **Optimizer** pull-down list.



4. Click **Setup...** to modify the Advanced Genetic Algorithm Optimizer Options.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A norm is a function that assigns a positive value to the cost function.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MATLAB Optimizer

The MATLAB optimizer option lets you pass a script to MATLAB to perform the optimization. When the optimization is analyzed, MATLAB is launched and a script is passed in to MATLAB to perform the optimization. During the optimization, MATLAB will call back into our application to perform the solve and compute the cost. The cost will be reported back to MATLAB, and MATLAB's optimization will determine the next step in the optimization.

The optimization script is specified as part of the optimization setup. By modifying the optimization script, users can change the optimization parameters and optimization method as well as use the full power of MATLAB in their optimization.

Running the Optimization

The MATLAB optimization is launched just like any other optimization. The Message Window will display status messages when MATLAB is being launched, and status messages will be generated for each solve that is being performed.

In most cases, MATLAB will terminate when the optimization has been completed. Some reasons why MATLAB would not terminate are:

- The user has modified the MATLAB script to not terminate MATLAB after the optimization.
- A syntax error or some other has occurred.
- The user has added some other code which runs after the optimization has completed.

System Requirements

In order to use MATLAB to perform optimizations from your application:

- A version of MATLAB must be installed on your system.
- The computing platform (e.g., 32/64 bit or Linux) of MATLAB MUST match the platform of the Ansys application you are using it with.
- You must have the MATLAB Optimization Toolkit installed.
- The MATLAB installation must include the MATLAB Optimization Toolbox. In addition, the MATLAB license must support using the Optimization Toolbox.

To see if the optimization toolbox is installed, users can type the "ver" command at the command prompt of a running MATLAB instance. For example:

```
>> ver
-----------------------------------------------------------------------
--------------------------------
MATLAB Version: 8.1.0.604 (R2023a)
MATLAB License Number: 162684
Operating System: Microsoft Windows 10
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Java Version: Java 1.6.0_17-b04 with Sun Microsystems Inc. Java
HotSpot(TM) 64-Bit Server VM mixed mode
--------------------------------------------------------------------
---------------------------------
MATLAB Version 8.1 (R2023a)
Simulink Version 8.1 (R2023a)
Optimization Toolbox Version 6.3 (R2023a)
```

To see if the optimization toolbox is licensed, you can use the "license('test','optimization_ toolbox')" command at the MATLAB command prompt:

```
>> license('test','optimization_toolbox')
ans =
1
```

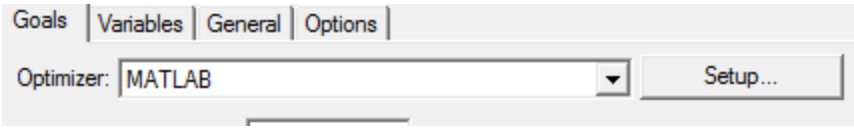
The answer will be 1 if the MATLAB Optimization Toolbox is licensed or 0 otherwise.

Specifying the MATLAB Location

The Tools> General Options: Miscellaneous group contains a setting for the MATLAB location. This setting must to point to the version of MATLAB to be used for performing the optimization. The platform (32/64 bit or Linux) of the specified version of MATLAB must match the platform of this application.

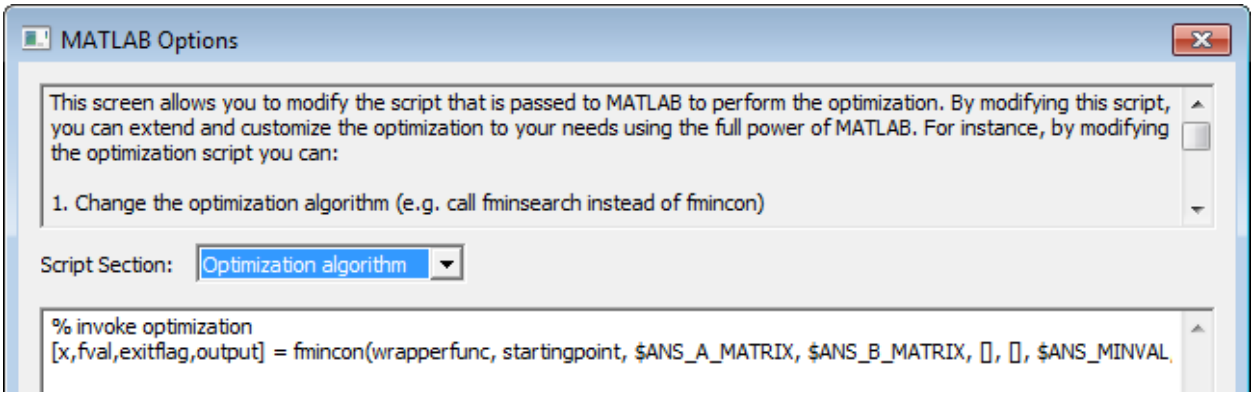
MATLAB Optimization Setup

MATLAB optimization starts by creating an optimization and selecting MATLAB from the optimizer drop-down menu. If you select MATLAB as the optimizer, the *Setup Optimization* dialog box displays a **Setup...** button.



Select **Setup...** to open the *MATLAB Options* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The upper text panel is informative. The Script section drop-down menu lets you select a lower panel display for Optimization algorithm, Options, or the Full script template.

This screen allows you to modify the script that is passed to MATLAB to perform the optimization. The complete script contains all the instructions necessary for MATLAB to connect to our application and perform the optimization, and a lot of that code is unimportant to users. We have addressed this issue by displaying a dropdown to let you view only the portion of code they are interested in without having to view the full script. The choices are:

- **Optimization algorithm**: displays only the line of code invoking the actual optimization function. By changing this line, users can use a different MATLAB function for optimization. By default we use fmincon() which is a derivative based constrained optimization. By modifying this line, users could replace the fmincon() call with fminsearch () to use an unconstrained pattern searching optimizer or another optimization function. See the MATLAB documentation for details about available optimization functions.
- **Options**: Each optimization function contains a multitude of options and parameters which are set in the MATLAB script prior to actually calling the optimization function. By modifying these options, the optimization can be customized as desired. For instance, options can be set for fmincon() to specify the algorithm that it uses internally. See the MATLAB documentation for details about options available for each optimization function.
- **Full script template**: This choice displays the full optimization script that is passed to MATLAB.

The initial Script Section display for the Optimization algorithm shows the following:

```
% invoke optimization
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, [],
[], [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon, options)
```

The initial Script Section Options display shows the following:

```
% customers can add their own options below
options = optimset(options, 'display', 'iter')
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
options = optimset(options, 'Algorithm', 'interior-point')
% options = optimset(options, 'PlotFcns', @optimplotfval)
```

You can modify the script to extend and customize the optimization to your needs. You must ensure that the script follows MATLAB syntax. For instance, by modifying the optimization script you can:

- Change the optimization algorithm (e.g., call fminsearch instead of fmincon).
- Change the parameters/options of the optimization algorithm (see the MATLAB documentation for details).
- Specify a plot function to provide graphical output during optimization.
- Specify a user defined output function to be called at completion or per iteration.

Symbols

When modifying the MATLAB code, users can use symbols to represent values from the optimization setup. The symbols and their definitions are listed below.

$ANS_VARIABLE_LIST:	list of variables we are optimizing
$ANS_STARTING_POINT:	vector of starting values of variables used in the optimization
$ANS_MAXITERATIONS:	maximum number of iterations specified in optimization setup
$ANS_MINVAL:	vector of minimum values from optimization setup
$ANS_MAXVAL:	vector of maximum values from optimization setup
$ANS_MINSTEP:	vector of minimum step sizes from optimization setup
$ANS_MAXSTEP:	vector of maximum step sizes from optimization setup
$ANS_A_MATRIX	matrix of linear constraint coefficients (left-hand side) generated from optimization setup
$ANS_B_MATRIX	matrix of linear constraint bounds (right-hand side) generated from optimization setup

Note:

The linear constraints as generated for MATLAB have the form [A][x] <= [B], where [A] is the coefficient matrix, [x] is the variable list matrix (column vector), and [B] is the bounds matrix (column vector).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> While modifying the script, please ensure that the script follows MATLAB syntax.

MATLAB Optimization Script Template

The script template shown in the Script Section is as follows:

```
% make sure platform matches
if strcmp(computer, '$ANS_EXPECTED_PLATFORM') ~= 1
   h = msgbox('32/64 platform does not match calling application,
  exiting')
   uiwait(h)
   exit
end
% add installation dir to search path so .mex file can be found
originalpath = addpath('$ANS_EXEDIR')
% connect back to opticomengine
callbackinterface = optimex('connect', '$ANS_CONNECTIONSTRING')
% set up optimization
% variables are: $ANS_VARIABLELIST
startingpoint = $ANS_STARTINGPOINT
options = optimset('MaxIter', $ANS_MAXITERATIONS)
iterationCallbackWrapper = @(x, optimValues, state) optimex
('notifyiterationcomplete', callbackinterface, x, optimValues.fval,
state)
options = optimset(options, 'OutputFcn', iterationCallbackWrapper)
% halt execution so debugger can be attached
% h = msgbox('attach debugger if desired')
% uiwait(h)
% attributes that user can pass to optimization algorithm
% variables are: $ANS_VARIABLELIST
% this is the objective function which returns cost
wrapperfunc = @(x)optimex('eval', callbackinterface, x)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
% this is our non linear constraint function, returns no constraints
returnempty = @(x)[];
nlcon = @(x) deal(returnempty(x), returnempty(x));
% DO NOT EDIT THIS LINE - START OPTIONS SECTION
% customers can add their own options below
options = optimset(options, 'display', 'iter')
options = optimset(options, 'Algorithm', 'interior-point')
% options = optimset(options, 'PlotFcns', @optimplotfval)
% DO NOT EDIT THIS LINE - END OPTIONS SECTION
% DO NOT EDIT THIS LINE - START OPTIMIZATION ALGO SECTION
% invoke optimization
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, $ANS_
A_MATRIX, $ANS_B_MATRIX, [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon,
options)
% DO NOT EDIT THIS LINE - END OPTIMIZATION ALGO SECTION
% write exit message to Ansoft message window (warning=0,error=1,info=2)
optimex('postansoftmessage', callbackinterface, 2, output.message)
% notify opticomengine that optimization is finished
optimex('optimizationfinished', callbackinterface, exitflag)
% restore original path
path = originalpath
% note: comment below line if you want MATLAB to remain
% running after optimization
exit
```

Optimization Setup for the MATLAB Optimizer

Following is the procedure for setting up an optimization analysis using the MATLAB Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **MATLAB** from the **Optimizer** drop-down menu. Selecting **MATLAB** enables the **Acceptable Cost** and **Noise** fields.
4. Click **Setup...** to modify the MATLAB Optimizer Options.
5. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the Max. No. of Iterations text box.
6. Under Cost Function, add a cost function by selecting the Setup Calculations button to open the Add/Edit Calculation dialog box.
7. Type the value of the cost function at which the optimization process should stop in the Acceptable Cost text box.
8. Type the cost function noise in the Noise text box.
9. If you want to select a Cost Function Norm Type:
   - Check the Show Advanced Option check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select L1, L2, or Maximum.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For more detail, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Mixed-Integer Sequential Quadratic Programming

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Non-Linear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to:

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$

$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

Where:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$x \in \mathbb{R}^{n_c}, y \in \mathbb{N}^{n_i}$$
$$x_l \leq x \leq x_u$$
$$y_l \leq y \leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second order corrections are retained. The Hessian of the Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP

Typically, the use of Nonlinear Programming by Quadratic Lagrangian (NLPQL) or Mixed-Integer Sequential Quadratic Programming (MISQP) optimizers is suggested for continuous problems when there is only one objective function. The problem might or might not be constrained and must be analytic. This means that the problem must be defined only by continuous input parameters and that the objective functions and constraints should not exhibit sudden "jumps" in their domain.

The main difference between these algorithms and Multi-Objective Genetic Algorithm (MOGA) is that MOGA is designed to work with multiple objectives and does not require full continuity of the output parameters. However, for continuous single objective problems, the use of NLPQL or MISQP gives greater accuracy of the solution as gradient information and line search methods are used in the optimization iterations. MOGA is a global optimizer designed to avoid local optima traps, while NLPQL and MISQP are local optimizers designed for accuracy.

For NLPQL and MISQP, the default convergence rate, which is specified by the Allowable Convergence (%) property, is set to 0.1% for a Direct Optimization system. The maximum value for this property is 100%. This is computed based on the (normalized) Karush-Kuhn-Tucker (KKT) condition. This implies that the fastest convergence rate of the gradients or the functions (objective function and constraint) determine the termination of the algorithm.

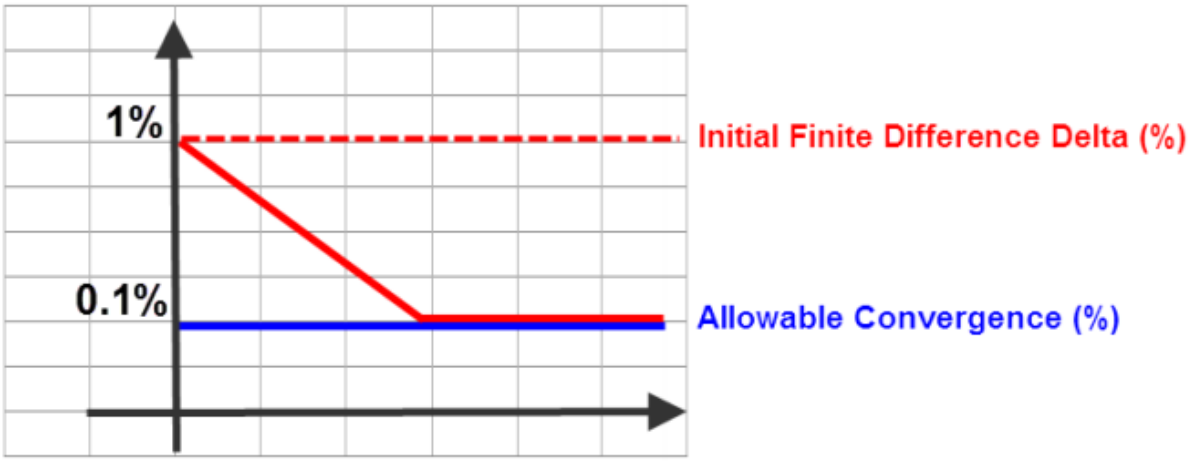
The default convergence rate is used in conjunction with the initial finite difference delta percentage value, which is specified by the Initial Finite Difference Delta (%) advanced property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This property defaults to 1% for a Direct Optimization system. You use this property to specify a percentage of the variation between design points to ensure that the Delta use in the calculation of finite differences is large enough to be seen over simulation noise. The specified percentage is defined as a relative gradient perturbation between design points.

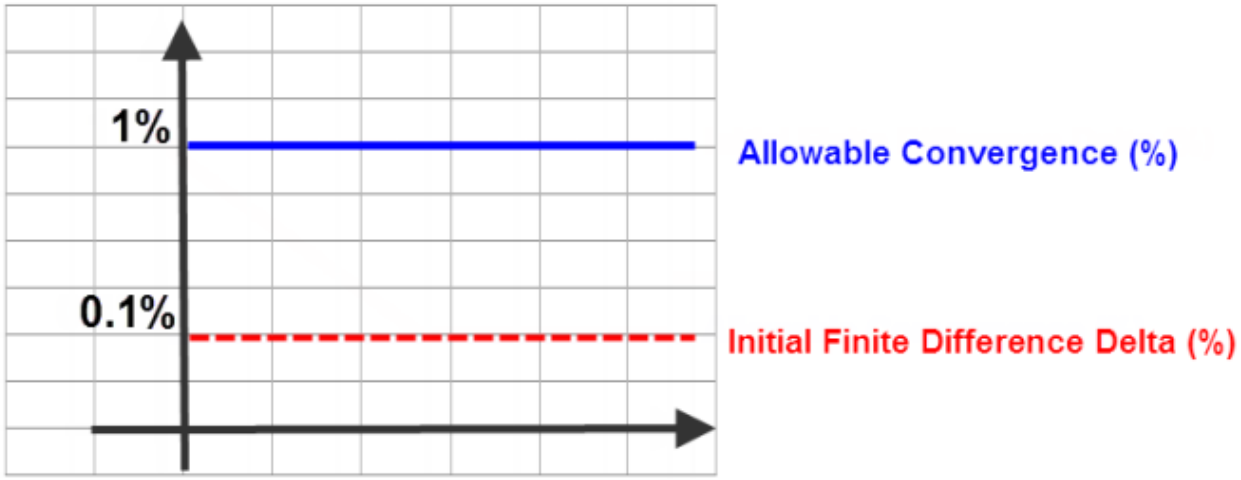
The advantage of this approach is that for large problems, it is possible to get a near-optimal feasible solution quickly without being trapped into a series of iterations involving small solution steps near the optima. To work most effectively with NLPQL and MISQP, keep the following guidelines in mind:

- If the Initial Finite Difference Delta (%) is greater than the Allowable Convergence (%), the relative gradient perturbation gets iteratively smaller, until it matches the allowable convergence rate. At this point, the relative gradient value stays the same through the rest of the analysis.



- If the Initial Finite Difference Delta (%) is less than or equal to the Allowable Convergence (%), the current relative gradient step remains constant through the rest of the analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Both the Initial Finite Difference Delta (%) and Allowable Convergence (%) should be higher than the magnitude of the noise in your simulation.

When setting the values for these properties, you have the usual trade-offs between speed and accuracy. Smaller values result in more convergence iterations and a more accurate (but slower) solution, while larger values result in fewer convergence iterations and a less accurate (but faster) solution. At the same time, however, you must be aware of the amount of noise in your model. For the input variable variations to be visible in the output variables, both values must be greater than the magnitude of the simulation's noise.

In general, default values for Initial Finite Difference Delta (%) and Allowable Convergence (%) cover the majority of optimization problems. For example, if you know that the noise magnitude in your direct optimization problem is 0.0001, then the default values (Allowable Convergence (%) = 0.001 and Initial Finite Difference Delta (%) = 0.01) are good.

When the defaults are not a good match for your problem, of course, you can adjust the values to better suit your model and your simulation needs. If you require a more numerically accurate solution, you can set the convergence rate to as low as 1.0E-10% and then set the Initial Finite Difference Delta (%) accordingly.

Setting Up Merit-based Sequential Quadratic Programming(Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Merit-based Sequential Quadratic Programming(Gradient) Optimizer or MBSQ Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Merit-based Sequential Quadratic Programming(Gradient)** from the **Optimizer** drop-down menu.
4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Use the **Options tab** if you want to enable use of a fast calculation-update algorithm to speed up Optimetrics and report updates during Optimetrics analyses, and to save the solution data for solved design variations in the analysis.

Setting Up Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Nonlinear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to

$$g_j(x,y)=0,\ \ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ \ j=m_e+1,\ldots,m$$

where

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second-order corrections are retained. The Hessian of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

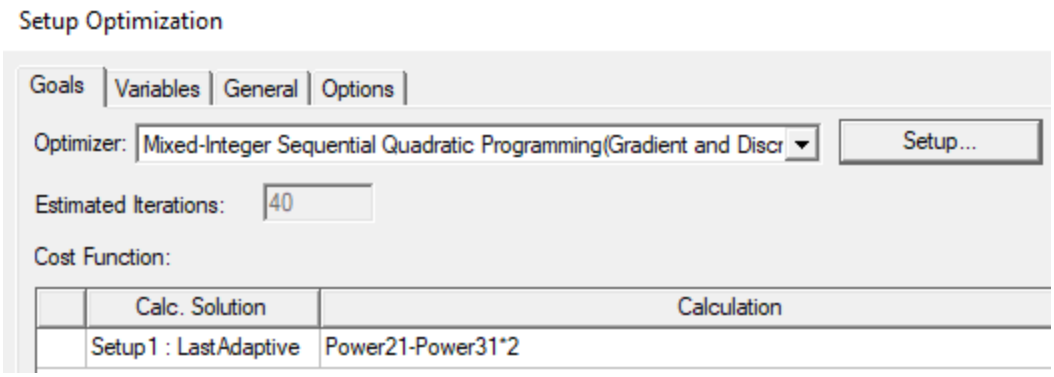
Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

Following is the procedure for setting up an optimization analysis using the Mixed-Integer Sequential Quadratic Programming optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Mixed-Integer Sequential Quadratic Programming (Nonlinear Programming by Quadratic Lagrangian) method (MISQP)can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. MISQP is available for both continuous and discrete input parameters, which is why mixed is in its name. MISQP can handle only one output parameter goal.. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

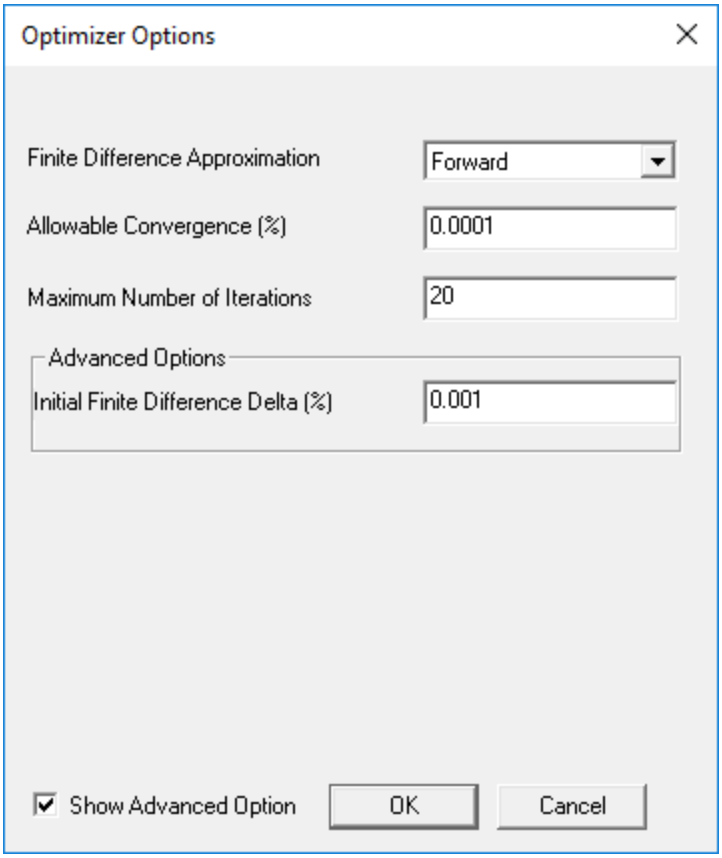
To generate samples and perform an NLPQL optimization:

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Finite Difference Approximation**: When analytical gradients are not available, MISQP approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For MISQP, the number of evaluations can be approximated according to the Finite Difference

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Approximation gradient calculation method, as follows:

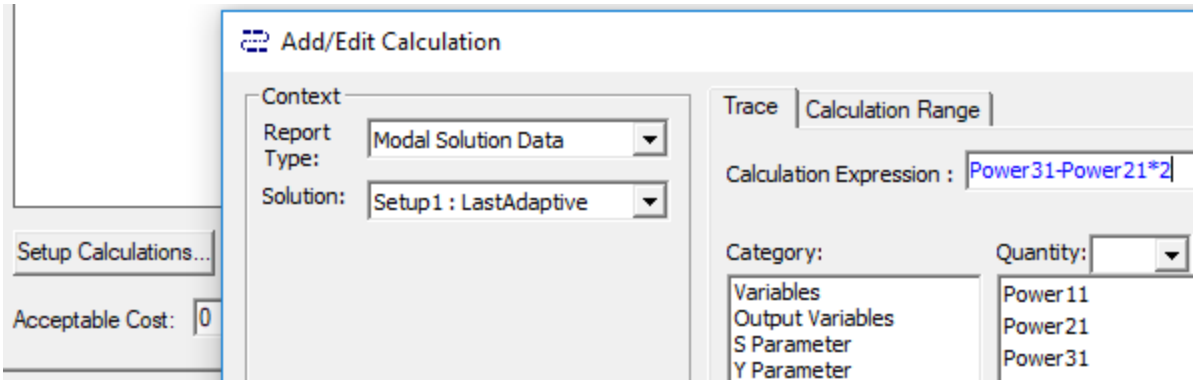
For **Central**: number of iterations * (2*number of inputs +1)

For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the MISQP process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.0001. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the MISQP solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in MISQP's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

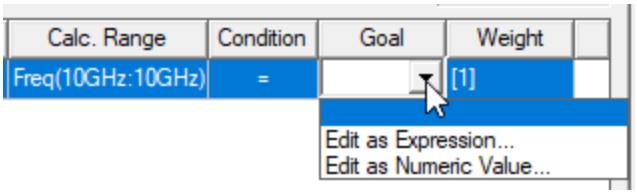
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

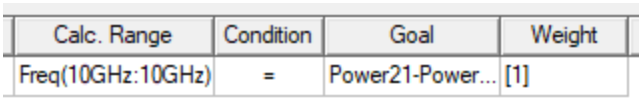
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu, for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
- Optionally, modify the values of fixed variables that are not being optimized.
- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting Up Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer

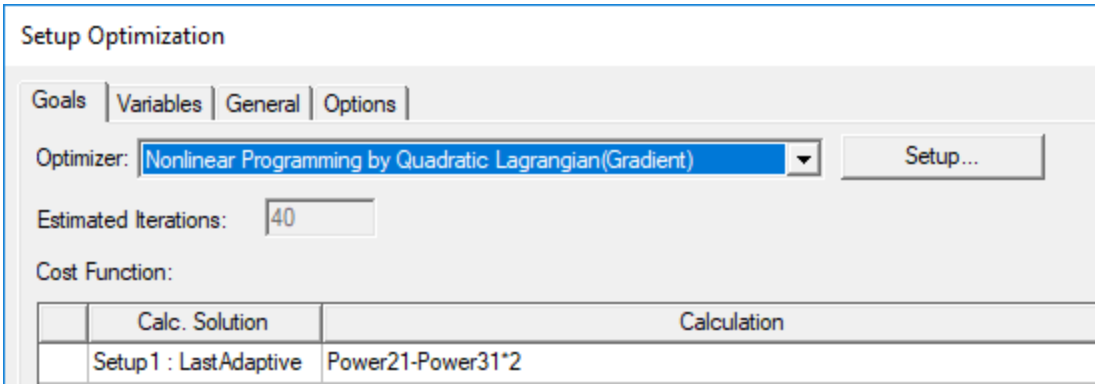
Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The NLPQL (Nonlinear Programming by Quadratic Lagrangian) method can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. Available for continuous input parameters only, NLPQL can handle only one output parameter goal. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

To generate samples and perform an NLPQL optimization:

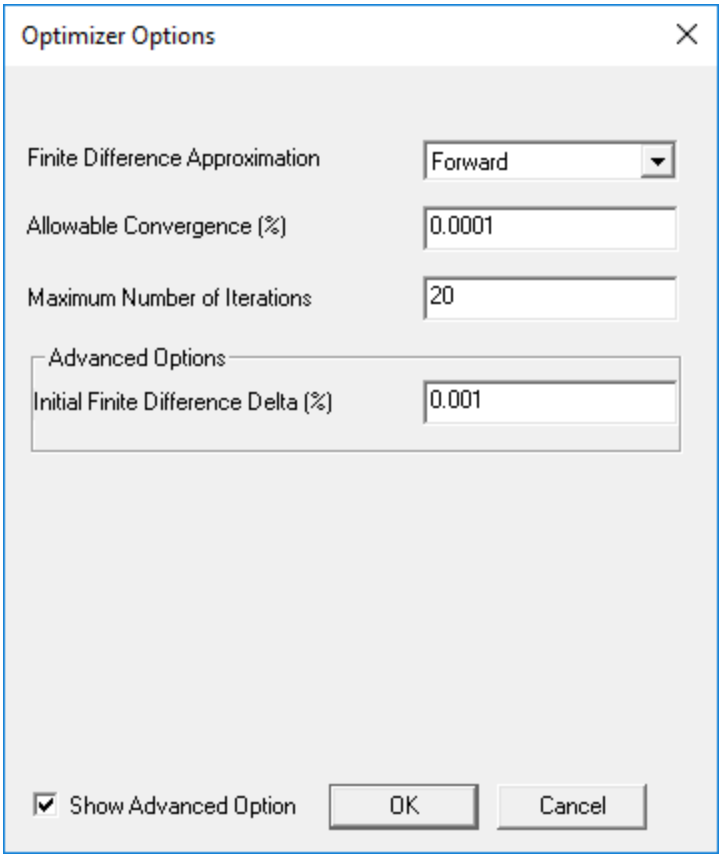
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Nonlinear Programming by Quadratic Lagrangian (Gradient)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the *Optimizer Options window*.



- **Finite Difference Approximation**: When analytical gradients are not available, NLPQL approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For NLPQL, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

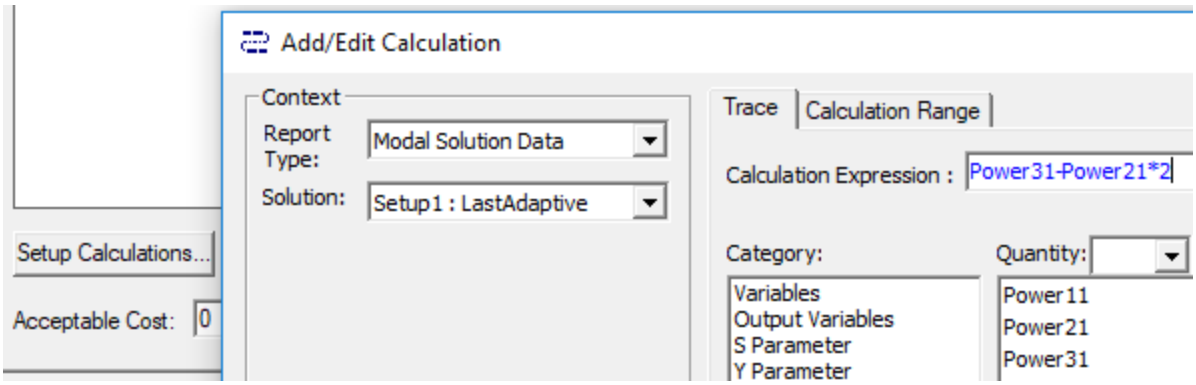
For **Central**: number of iterations * (2*number of inputs +1)

For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the NLPQL process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.1. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the NLPQL solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in NLPQL's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

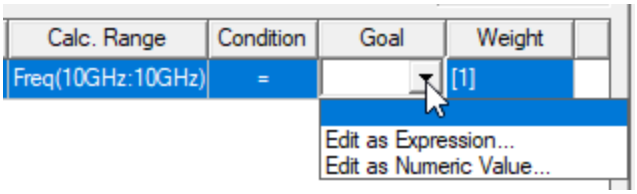
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

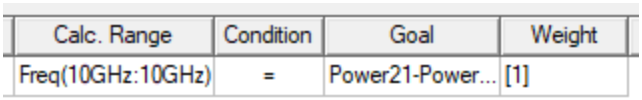
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value...



   This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Multi-Objective Genetic Algorithm (MOGA)

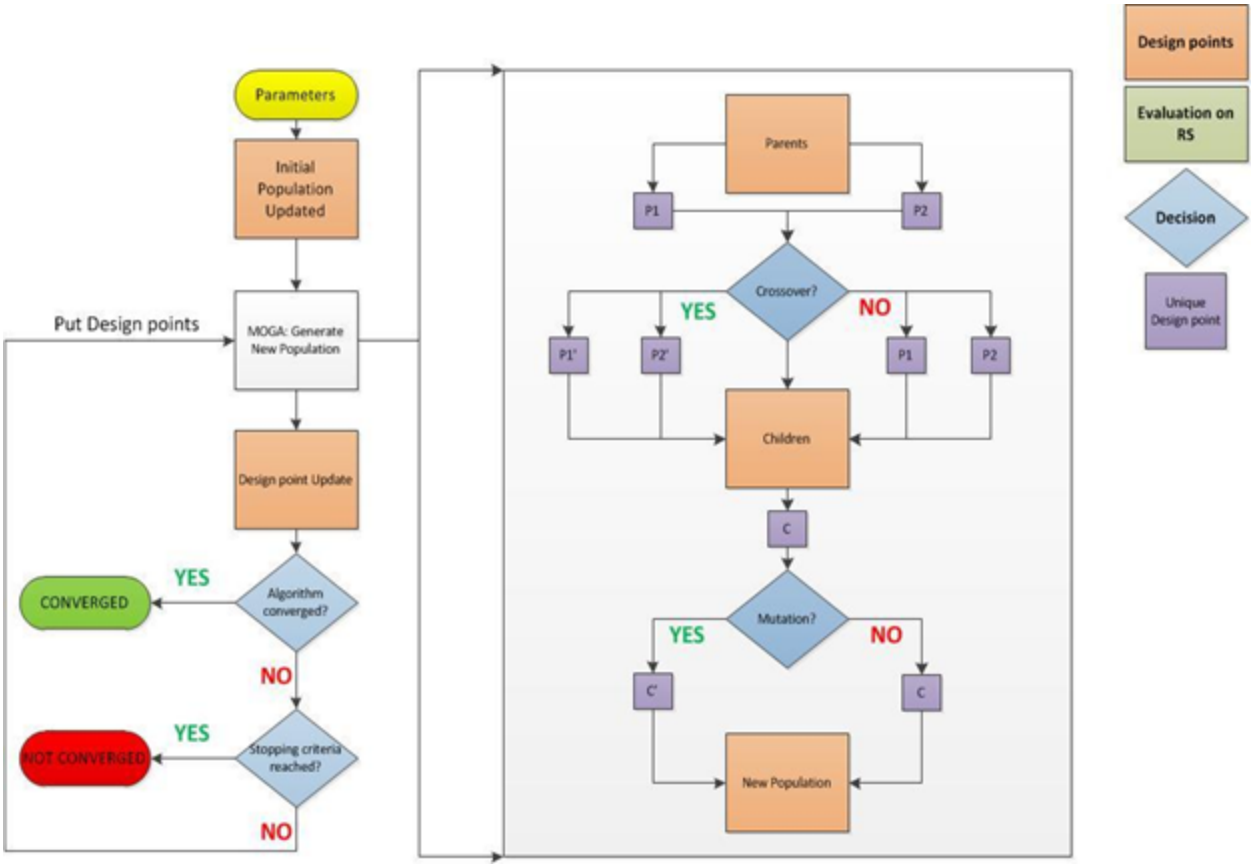
The Multi-Objective Genetic Algorithm (MOGA) used in GDO is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives. Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property. For more information about this and other MOGA properties, see Setup Multi-Objective Genetic Algorithm.

MOGA Workflow

The MOGO workflow follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



MOGA Steps

1. **First Population of MOGA**

   The initial population is used to run MOGA.

2. **MOGA Generates a New Population**

   MOGA is run and generates a new population via cross-over and mutation. After the first iteration, each population is run when it reaches the number of samples defined by the Number of Samples Per Iteration property. For details, see **MOGA Steps to Generate New Population** below.

3. **Design Point Update**

   The design points in the new population are updated.

4. **Convergence Validation**

   The optimization is validated for convergence.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Yes: Optimization Converged**

  MOGA converges when the Maximum Allowable Pareto Percentage or the Convergence Stability Percentage has been reached.

- **No: Optimization Not Converged**

  If the optimization is not converged, the process continues to the next step.

5. **Stopping Criteria Validation**

   If the optimization has not converged, it is validated for fulfillment of stopping criteria.

   - **Yes: Stopping Criteria Met**

     When the Maximum Number of Iterations criterion is met, the process is stopped without having reached convergence.

   - **No: Stopping Criteria Not Met**

     If the stopping criteria have not been met, MOGA is run again to generate a new population (return to Step 2).

6. **Conclusion**

   Steps 2 through 5 are repeated in sequence until the optimization has converged or the stopping criteria have been met. When either of these things occurs, the optimization concludes.

MOGA Steps to Generate a New Population

The process MOGA uses to generate a new population has two main steps: **Cross-over** and **Mutation**.

1. **Cross-over**

   Cross-over combines (mates) two chromosomes (parents) to produce a new chromosome (offspring). The idea behind cross-over is that the new chromosome can be better than both of the parents if it takes the best characteristics from each of the parents. Cross-over occurs during evolution according to a user-definable cross-over probability.

   - **Cross-over for Continuous Parameters**

     A cross-over operator that linearly combines two parent chromosome vectors to produce two new offspring according to the following equations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{Offspring1}=\mathbf{a}*\text{Parent1}+(1-\mathbf{a})*\text{Parent2}$$
$$\text{Offspring2}=(1-\mathbf{a})*\text{Parent1}+\mathbf{a}*\text{Parent2}$$

Consider the following two parents (each consisting of four floating genes), which have been selected for cross-over:

$$\text{Parent 1: } (0.3)(1.4)(0.2)(7.4)$$
$$\text{Parent 2: } (0.5)(4.5)(0.1)(5.6)$$

If a = 0.7, the following two offspring would be produced:

$$\text{Offspring1: } (0.36)(2.33)(0.17)(6.86)$$
$$\text{Offspring2: } (0.402)(2.981)(0.149)(6.842)$$

- **Cross-over for Discrete Parameters and Continuous Parameters with Manufacturable Values**

Each discrete parameter or continuous parameter with manufacturable values is represented by a binary chain corresponding to the number of levels. For example, a parameter with two values (levels) is encoded to one bit, a parameter with seven values is encoded to three bits, and an -bits chain represents a parameter with values.

The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

Three different kinds of cross-over are available:

- One-Point

  A one-point cross-over operator that randomly selects a cross-over point within a chromosome then interchanges the two parent chromosomes at this point to produce two new offspring.

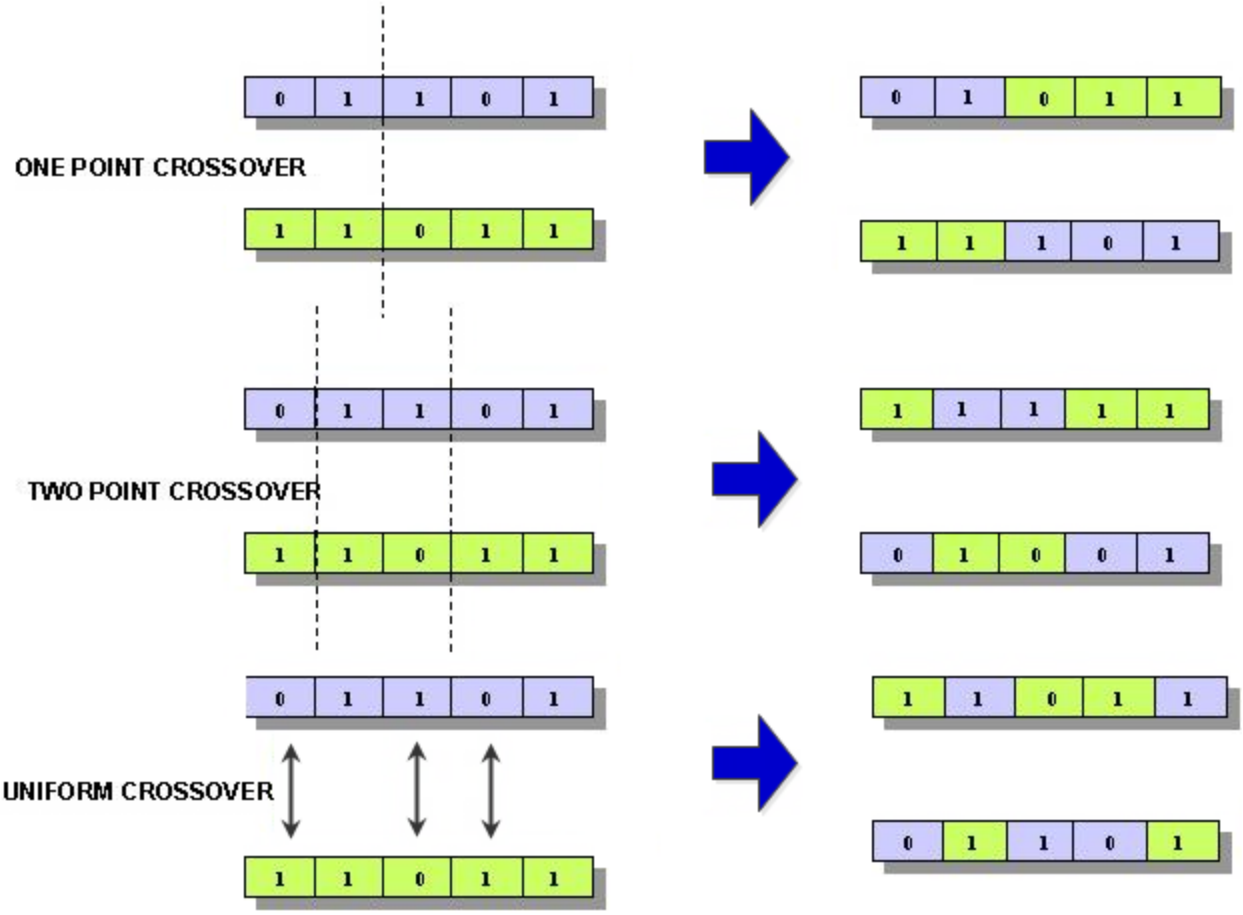
- Two-Point

  A two-point cross-over operator randomly selects two cross-over points within a chromosome then interchanges the two parent chromosomes between these points to produce two new offspring.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Uniform

  A uniform cross-over operator decides (with some probability, which is known as the "mixing ratio") which parent contributes each of the gene values in the offspring chromosomes. This allows the parent chromosomes to be mixed at the gene level rather than the segment level (as with one and two-point cross-over). For some problems, this additional flexibility outweighs the disadvantage of destroying building blocks.



2. **Mutation**

   Mutation alters one or more gene values in a chromosome from its initial state. This can result in entirely new gene values being added to the gene pool. With these new gene values, the genetic algorithm might be able to arrive at a better solution than was previously possible. Mutation is an important part of the genetic search, as it helps to prevent the population from stagnating at any local optima. Mutation occurs during evolution according to a user-defined mutation probability.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mutation for Continuous Parameters**

  For continuous parameters, a polynomial mutation operator is applied to implement mutation.

  $$C = P + (\text{UpperBound} - \text{LowerBound})\delta$$

  where C is the child, P is the parent, and δ is a small variation calculated from a polynomial distribution.

- **Mutation for Discrete Parameters and Continuous Parameters with Manufacturable Values**

  For discrete parameters or continuous parameters with manufacturable values, a mutation operator simply inverts the value of the chosen gene (0 goes to 1 and 1 goes to 0) with a probability of 0.5. This mutation operator can only be used for binary genes. The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

Convergence Criteria in MOGA-Based Multi-Objective Optimization

Convergence criteria are the conditions that indicate when the optimization has converged. In the Multi-Objective Genetic Algorithm (MOGA)-based multi-objective optimization methods, the following convergence criteria are available:

Maximum Allowable Pareto Percentage

The **Maximum Allowable Pareto Percentage** criterion looks for a percentage that represents a specified ratio of Pareto points per Number of Samples Per Iteration. When this percentage is reached, the optimization is converged.

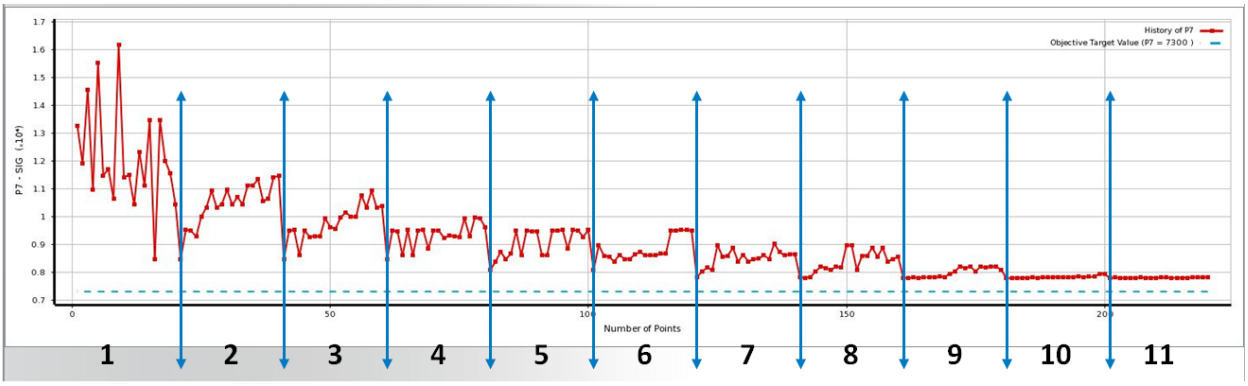
Convergence Stability Percentage

The **Convergence Stability Percentage** criterion looks for population stability, based on mean and standard deviation of the output parameters. When a population is stable with regards to the previous one, the optimization is converged. The criterion functions in the following sequence:

- **Population 1**: When the optimization is run, the first population is not taken into account. Because this population was not generated by the MOGA algorithm, it is not used as a range reference for the output range (for scaling values).
- **Population 2**: The second population is used to set the range reference. The minimum, maximum, range, mean, and standard deviation are calculated for this population.
- **Populations 3 – 11**: Starting from the third population, the minimum and maximum output values are used in the next steps to scale the values (on a scale of 0 to 100). The mean

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variations and standard deviation variations are checked. If both of these are smaller than the value for the Convergence Stability Percentage property, the algorithm is converged.



At each iteration and for each active output, convergence occurs if:

$$\frac{|Mean_i - Mean_{i-1}|}{Max - Min} < \frac{S}{100}$$

and

$$\frac{|StdDev_i - StdDev_{i-1}|}{Max - Min} < \frac{S}{100}$$

Where:

S=Stability Percentage

$Mean_i$= Mean of the *i*-th Population

$StdDev_i$ = Standard Deviation of the *i*-th Population

Max = Maximum Output Value calculated on the first generated population of MOGA

Min = Minimum Output Value calculated on the first generated population of MOGA

Setting Up Multi-Objective Genetic Algorithm(Random Search) Optimizer

Following is the procedure for setting up an optimization analysis using the Multi-Objective Genetic Algorithm (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

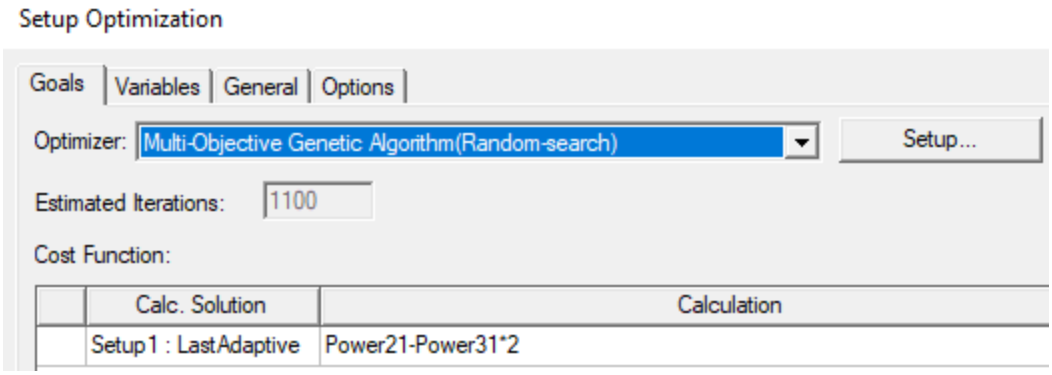
The Multi-Objective Genetic Algorithm (MOGA) is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives. Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

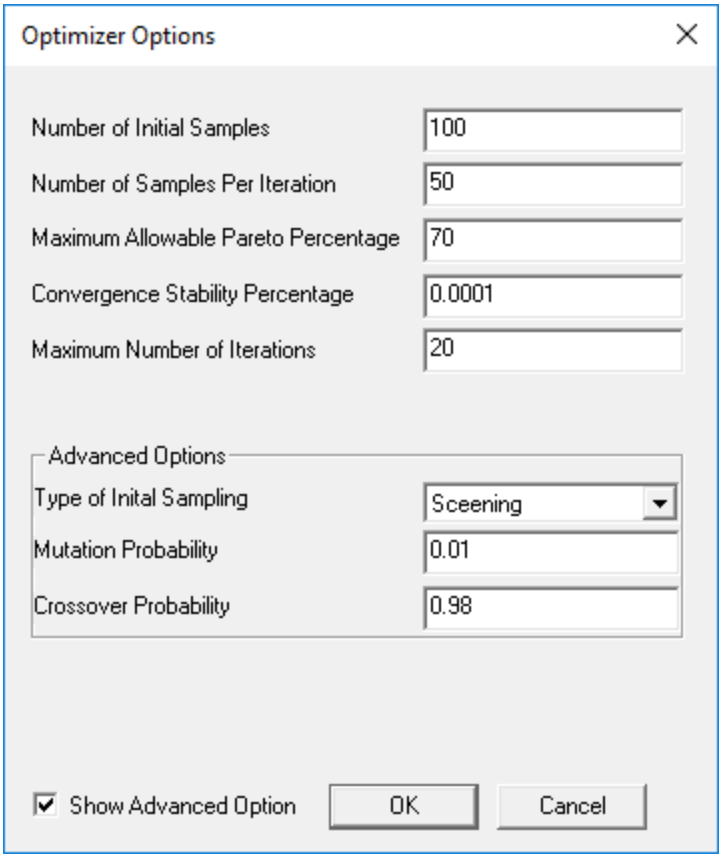
The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Multi-Objective Genetic Algorithm (Random Search)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

  The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

  If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can enter a minimum of 2 and a maximum of 10000.

- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

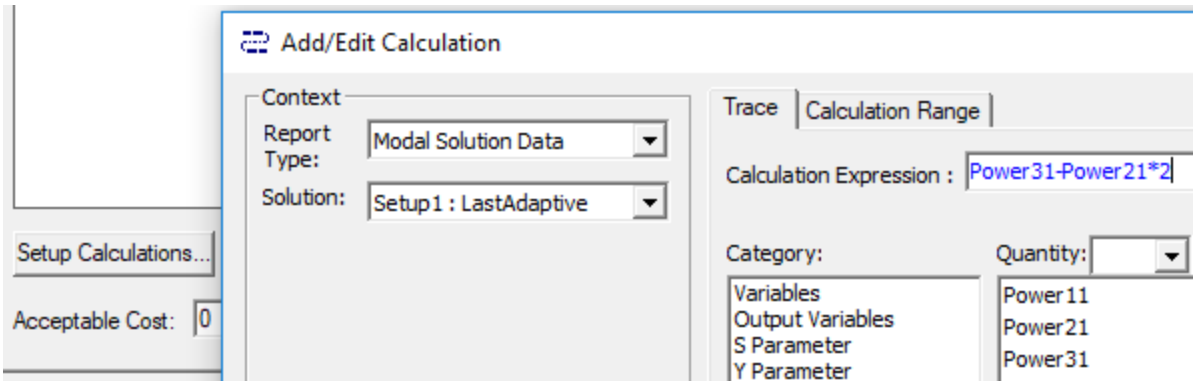
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

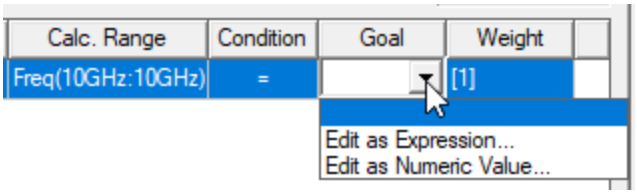
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog.

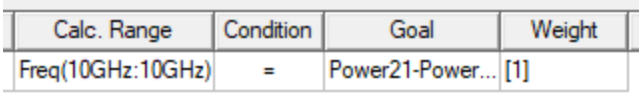
6. In the Optimization setup, in the dropdown for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box.

This enters the expression/value into the **Goal** column.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Pattern Search (Search-Based)

If the noise is significant in the nominal project, use the Pattern Search optimizer to obtain the results. It performs a grid-based simplex search, which makes use of simplices: triangles in 2D space or tetrahedra in 3D space. A simplex is a Euclidean geometric spatial element having the minimum number of boundary points, such as a line segment in one-dimensional space, a triangle in two-dimensional space, or a tetrahedron in three-dimensional space.

The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern continues. If no improvement occurs, the grid is refined. If improvement occurs, the step is accepted and the new simplex is generated to replace the original one. The figures below illustrate a triangular simplex mirrored several times to demonstrate the pattern search approach in two variables and the simplices superimposed on a 2D cost function to demonstrate the convergence toward a minimum in the cost function.

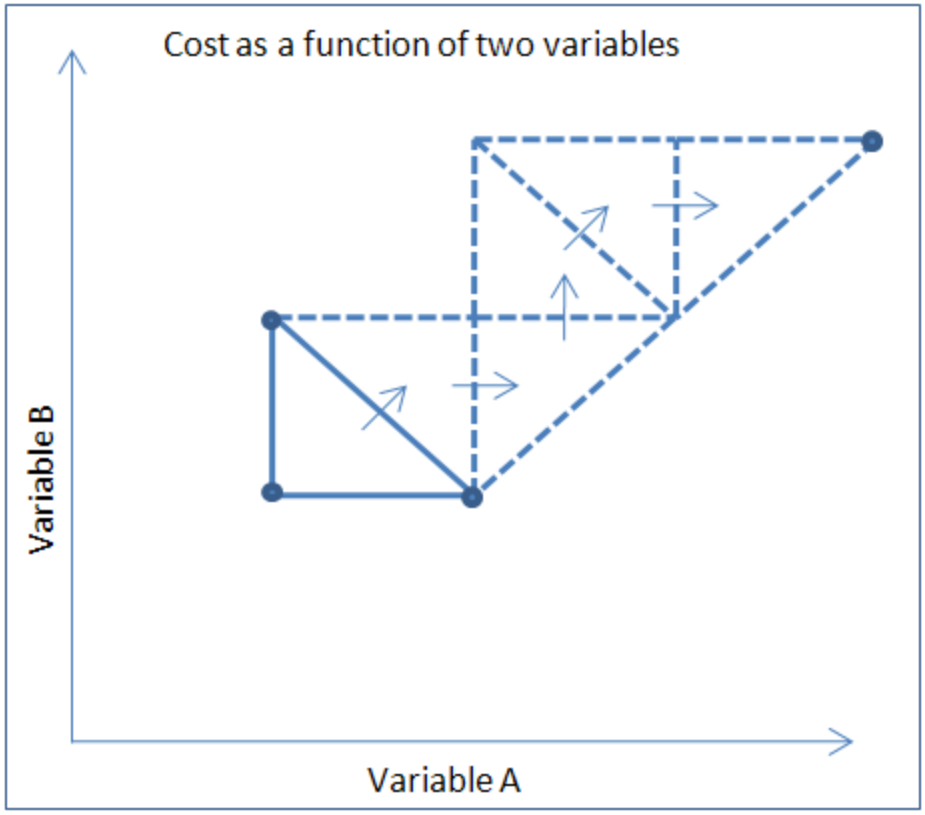
Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

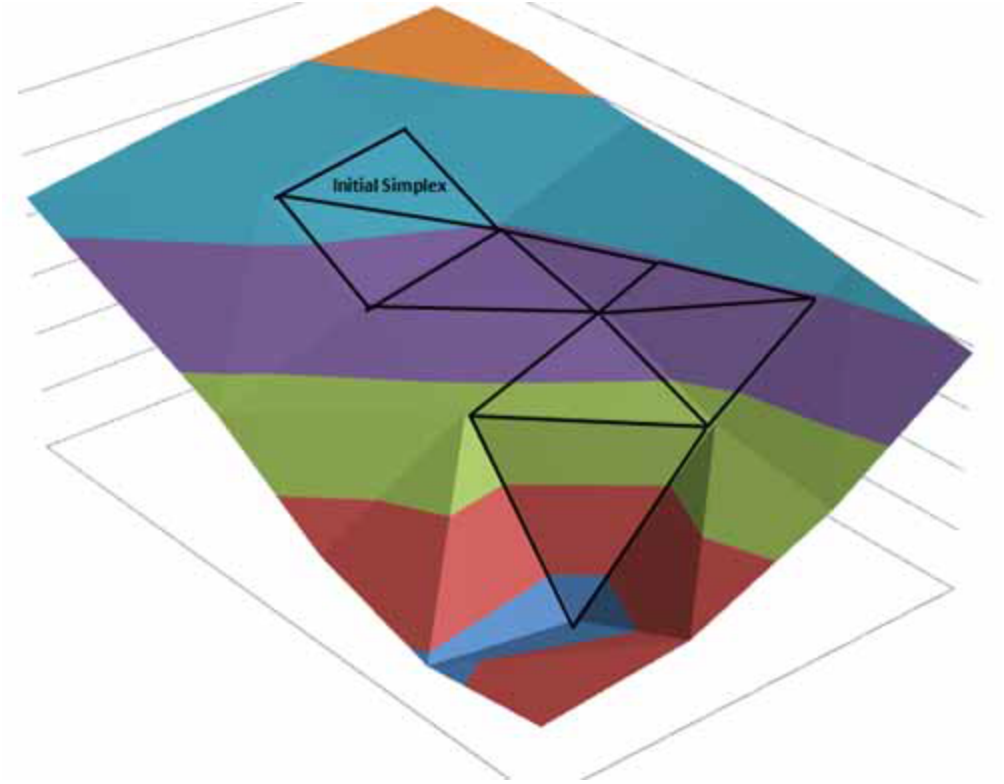
Note:

The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.



The Pattern Search algorithms are extensible to three variable optimization by using tetrahedral simplices, however, they are not easily represented in graphical form. Generally, Pattern Search algorithms are not used when more than three variables are used in the optimization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When there is no improvement in the cost function regardless of the direction the simplex is mirrored, then the simplex is subdivided into smaller simplices and the process restarted.

Pattern Search algorithms have several advantages over Quasi-Newton algorithms. First, they are less sensitive to noise because the cost function is evaluated at all node points on the simplex and the numerical noise averages out over the simplex. The second advantage is that the number of initial solutions is generally smaller. However, since the pattern search does not use gradient information to locate the minimum the process converges more slowly toward the true minimum, taking more steps to successively divide the simplices as the minimum is approached.

Optimization Setup for the Pattern Search (Search-based) Optimizer

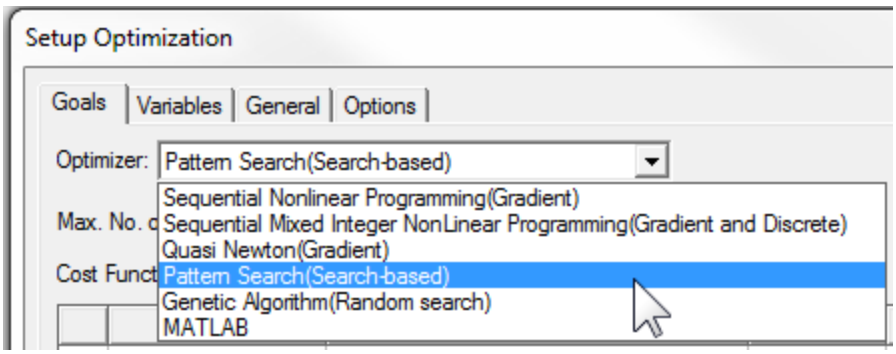
Following is the procedure for setting up an optimization analysis using the Pattern Search (Search-based) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Goals** tab, select the optimizer by selecting **Pattern Search (Search-based)** from the **Optimizer** drop-down menu.



   Selecting Pattern Search enables the **Acceptable Cost** and **Noise** fields.

4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box.
7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
- Optionally, modify the values of fixed variables that are not being optimized.
- Optionally, set Linear constraints.

Select the **View all columns** check box to see all columns, including hidden columns.

11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Quasi Newton (Gradient)

If the Sequential Non Linear Programming Optimizer has difficulty, and if the numerical noise is insignificant during the solution process, use the Quasi Newton optimizer to obtain the results. The Quasi Newton optimizer works on the basis of finding a minimum or maximum of a cost function which relates variables in the model or circuit to overall simulation goals. The user marks one or more variables in the project and defines a cost function in the optimization setup. The cost function relates the variable values to field quantities, design parameters like force or torque, power loss, etc. The optimizer can then maximize or minimize the value of the design parameter by varying the problem variables.

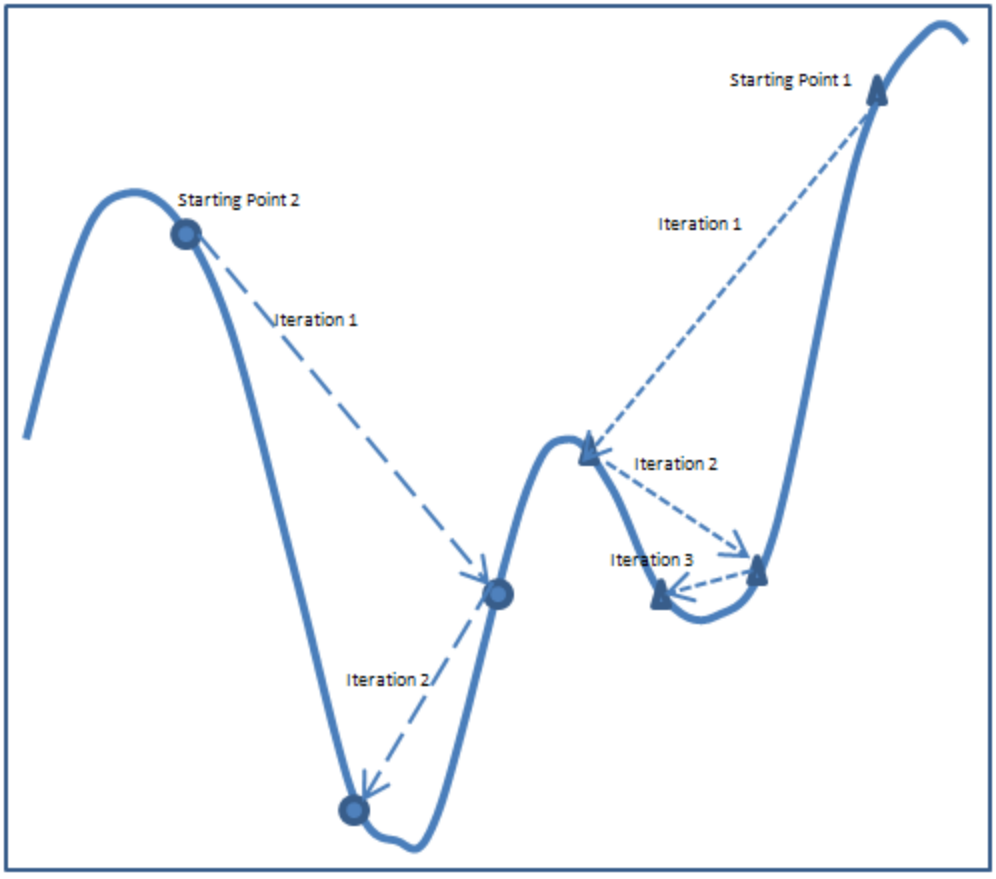
Sir Isaac Newton first showed that the maximum or minimum of any function can be determined by setting the derivative of a function with respect to a variable (x) to zero and solving for the variable. This approach leads to the exact solution for quadratic functions. However, for higher order functions or numerical analysis, an iterative approach is commonly taken. The function is approximated locally by a quadratic and the approximation is solved for the value of x. This value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is placed back into the original function and used to calculate a gradient which provides a step direction and size for determining the next best value of x in the iteration process.

In the Quasi-Newton optimization procedure, the gradients and Hessian are calculated numerically. Essentially, the change in x and the change in the gradient are used to estimate the Hessian for the next iteration. The ratio of the change in cost to the change in the values of x provides the gradient, whereas, the ratio of the change in the gradients to the change in the values of x provides the Hessian for the next step and is know as the quasi-Newton condition. In order to perform the Quasi-Newton optimization, at least three solutions are required for each parameter being varied. This can have a significant computational cost depending upon the type of analysis being performed.

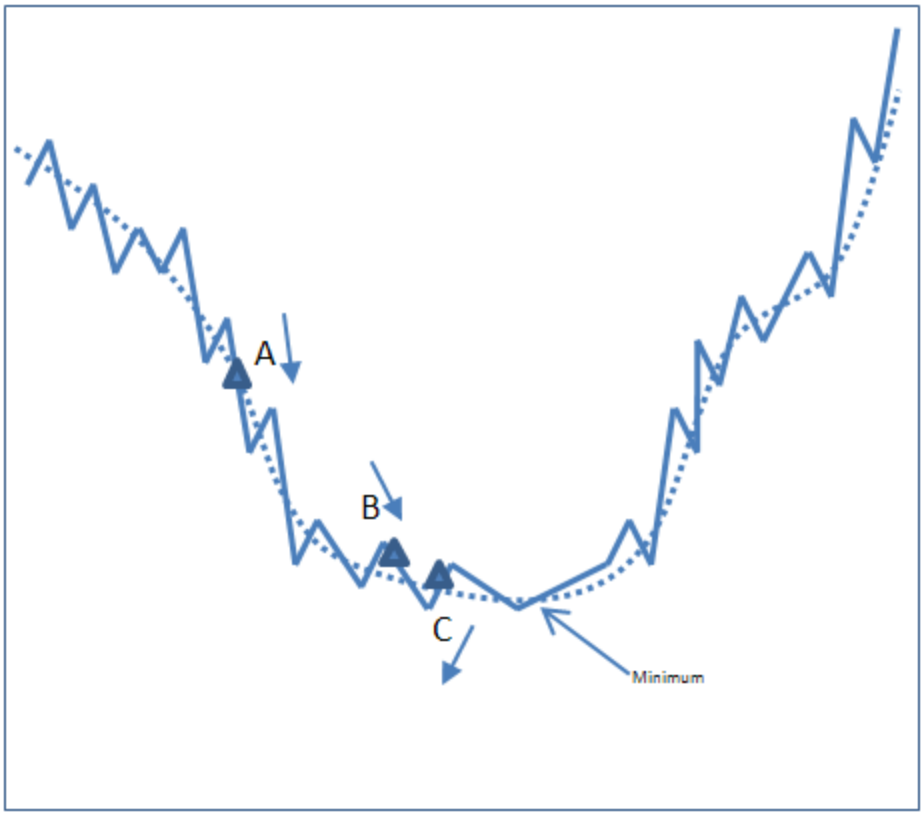
There are numerous methods described in the literature for solving for the Hessian and the details of the method used by Optimetrics are beyond the scope of this document. However, as the Quasi-Newton method is, at its heart, a gradient method, it suffers from two fundamental problems common to optimization. The first is the possible presence of local minima. The following figure illustrates the problem of local minima.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this scenario, you can see that in order to find the minimum of the function over the domain, a number of factors will determine the overall success including the initial starting point, the initial set of gradients calculated, the allowable step size, etc. Once the optimizer has located a minimum, the Quasi-Newton approach will locate the bottom and will not search further for other possible minima. In the example shown, when the optimizer begins at the point labeled "Starting Point 1" the minima it finds is a local minima and not a good global solution to the problem.

The second basic issue with Quasi-Newton optimization is numerical noise. In gradient optimization, the derivatives are assumed to be smooth, well behaved functions. However, when the gradients are calculated numerically, the calculation involves taking the differences of numbers that get progressively smaller. At some point, the numerical imprecision in the parameter calculations becomes greater than the differences calculated in the gradients and the solution will oscillate and may never reach convergence. To illustrate this, consider the figure shown below.



In this scenario, the optimizer is looking for the point labeled "minimum". Three possible solutions are labeled A, B and C, with each arrow indicating the direction of the derivative of the function at that point. If points A and B represent the last two solution points for the parameter, then it is easy to see that the changes in the magnitude and the consistent direction of the derivatives will serve to push the solution closer to the desired minimum. If, however, points A

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and C are the last two solution points respectively, the magnitude indicates the proper direction of movement, but the derivatives are opposite, possibly causing the solution to move away from the minimum, back in the direction of point A.

In order to use the Quasi-Newton optimizer effectively, the cost function should be based on parameters that exhibit a smooth characteristic (little numerical noise) and a starting point of the optimization should be chosen somewhat close to the expected minimum based on an understanding of the physical problem being optimized. This becomes increasingly difficult, however, when multiple parameters are being varied or when multiple parameters are to be optimized. In addition, the computational burden of multivariate optimization with Quasi-Newton increases geometrically with the number of variables being optimized. As a result, this method should only be attempted when 1 or 2 variables are being optimized as a time.

For more information regarding Quasi-Newton optimization methods, see the following reference:

Schoenberg, Ronald. *Optimization with the Quasi-Newton Method*. Aptech Systems, Inc. 2001.

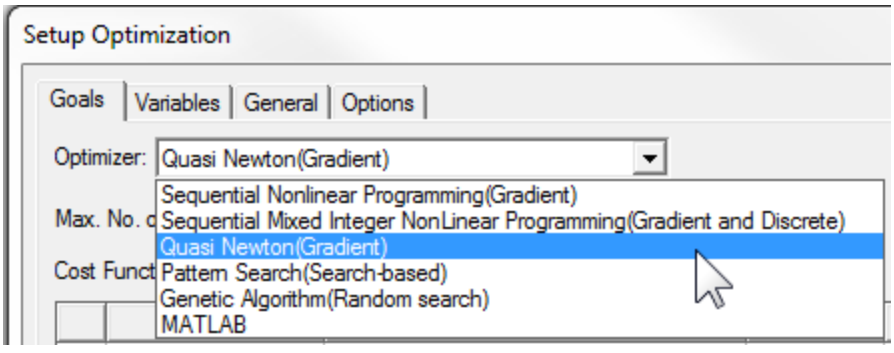
Optimization Setup for the Quasi Newton (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Quasi Newton (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Quasi Newton (Gradient)** from the **Optimizer** drop-down menu. Selecting Quasi Newton (Gradient) enables the **Acceptable Cost** and **Noise** fields.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box. Note that for Quasi Newton, if you specify 0 as the acceptable cost, the simulation stops after the first analysis.
7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)

The Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) optimizer is equivalent to the SNLP (Gradient) optimizer with only one difference. Many problems require variables take only discrete values. One example might be to optimize on the number of turns in a coil. To be able to optimize on number of turns or quarter turns, the optimizer must handle discrete optimization variables. The SMINLP optimizer can mix continuous variables among the integers, or can have only integers, and works if all variables are continuous. The setup resembles the setup for SNLP, except that you must flag the integer variables.supporting integer variables. You can set up internal variables based on the integer optimization variable.

For example, consider N to be an integer optimization variable. By definition it can only assume integer values. You can establish another variable, which further depends on this one: K = 2.345 * N, or K = sin(30 * N). This way K has a discrete value, but is not necessarily integer. Or, one can use N directly as a design parameter.

Optimization Setup for the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer

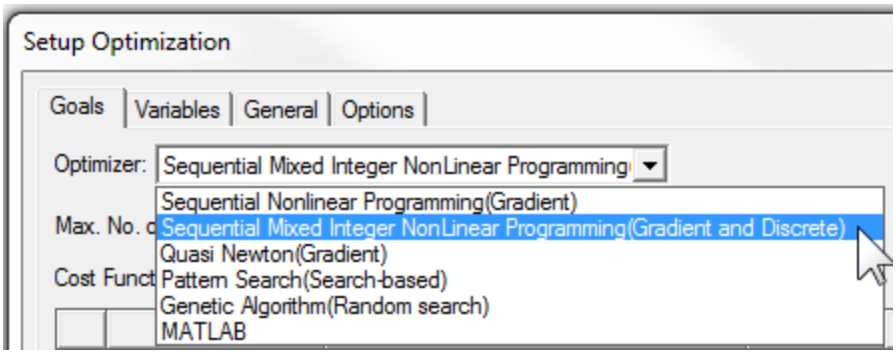
Following is the procedure for setting up an optimization analysis using the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - Check the Integer box for integer variables.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Sequential Nonlinear Programming (Gradient)

The main advantage of SNLP (Gradient) over Quasi Newton (Gradient) is that it handles the optimization problem in more depth. This optimizer assumes that the optimization variables span a continuous space. As a result, there is no Minimum Step Size specified in this optimizer and the variables may take any value within the allowable constraints and within the numerical precision limits of the simulator. Like Quasi Newton, the SNLP optimizer assumes that the noise is not significant. It does reduce the effect of the noise, but the noise filtering is not strong.

The SNLP optimizer approximates the FEA characterization with Response Surfaces (RS). With the FEA-approximation and with light evaluation of the cost function, SNLP has a good approximation of the cost function in terms of the optimization variables. This approximation allows the SNLP optimizer to estimate the location of improving points. The overall cost approximations are more accurate. This allows the SNLP optimizer a faster practical convergence speed than that of quasi Newton.

The SNLP Optimizer creates the response surface using a polynomial approximation from the FEA simulation results available from past solutions. The response surface is most accurate in the local vicinity. The response surface is used in the optimization loop to determine the gradients and calculate the next step direction and distance. The response surface acts as a surrogate for the FEA simulation, reducing the number of FEA simulations required and greatly speeding the problem. Convergence improves as more FEA solutions are created and the response surface approximation improves.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The SNLP method is similar to the Sequential Quadratic Programming (SQP) method in two ways: Both are sequential, updating the optimizer state to the current optimal values and iterating. Sequential optimization can be thought of as walking a path, step by step, toward an optimal goal. SNLP and SQP optimizers are also similar in that both use local and inexpensive surrogates. However, in the SNLP case, the surrogate can be of a higher order and is more generally constrained. The goal is to achieve a surrogate model that is accurate enough on a wider scale, so that the search procedures are well lead by the surrogate, even for relatively large steps. All functions calculated by the supporting finite element product (for example, Maxwell 3D or HFSS) is assumed to be expensive, while the rest of the cost calculation (for example, an extra user-defined expression) — which is implemented in Optimetrics — is assumed to be inexpensive. For this reason, it makes sense to remove inexpensive evaluations from the finite element problem and, instead, implement them in Optimetrics. This optimizer holds several advantages over the Quasi Newton and Pattern Search optimizers.

Most importantly, due to the separation of expensive and inexpensive evaluations in the cost calculation, the SNLP optimizer is more tightly integrated with the supporting FEA tools. This tight integration provides more insight into the optimization problem, resulting in a significantly faster optimization process. A second advantage is that the SNLP optimizer does not require cost-derivatives to be approximated, protecting against uncertainties (noise) in cost evaluations. In addition to derivative-free state of the RS-based SNLP, the RS technique also proves to have noise suppression properties.

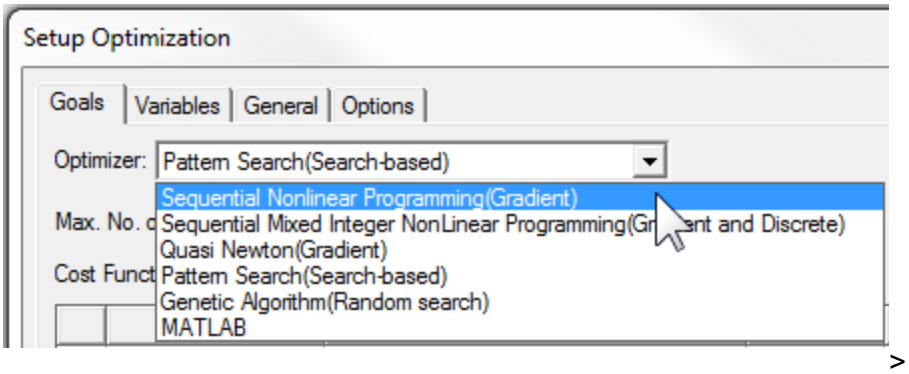
Optimization Setup for the Sequential Nonlinear Programming (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Sequential Nonlinear Programming (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Sequential Nonlinear Programming (Gradient)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Optimization Variables and the Design Space

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via FEA and assigned a cost value;

therefore, the cost function is defined over the design space (cost(*x*): $R^n \rightarrow R$, where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the Rn space. This subset is called the feasible domain and is defined via linear constraints.

Setting Up an Optimization Analysis

Optimization allows you to vary predefined variables in the nominal design to search for the solution that best satisfies a set of user defined goals or cost functions. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- You can define more than one optimization analysis setup per design.
- You can create an Optimization setup before defining variables but all variables must be defined before you start the Optimization analysis.
- Once you have created an optimization analysis setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To provide a broad range of capability, Optimetrics incorporates the following types of numerical optimizers:

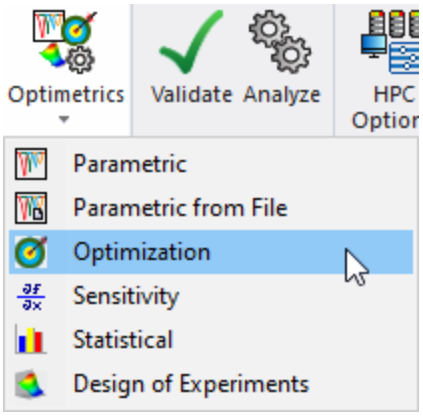
- Screening (Search based)
- Multi-Objective Genetic Algorithm
- Nonlinear Programming by Quadratic Lagrangian (Gradient)
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)
- Adaptive Multiple Objective (Gradient)
- Adaptive Single Objective
- MATLAB

Legacy Optimizers include

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random search)

Click on the links above to view the setup procedure for each optimizer. Options for the analysis are listed in the table. Besides setting up an Optimization analysis from the Optimization menu, you can also the **Simulation** tab of the ribbon, and select from the menu under the Optimetrics icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following *optional* optimization solution setup options can also be used:

- Modify the starting variable value.
- Edit the Calc. Range text field or use the Edit Calculation Range dialog box.
- Modify the minimum and maximum values of variables that will be optimized.
- Exclude variables from optimization.
- Modify the values of fixed variables that are not being optimized.
- Set the minimum and maximum step size between solved design variations (for the Quasi Newton (Gradient) and Pattern Search (Search based optimizers), **Variables** tab).
- Set the minimum and maximum focus size (for the SNLP Gradient and SMINLP Gradient and Discrete optimizers, **Variables** tab).
- Set Linear constraints.
- Request that Optimetrics solve a parametric sweep before an optimization analysis.
- Request that Optimetrics solve a parametric sweep during an optimization analysis.
- Automatically update optimized variables to the optimal values during an optimization or after an optimization analysis is completed.
- Change the norm used for the cost function calculation (Advanced Option)
- Open the HPC and Analysis Options window.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Setting the Maximum Iterations for an Optimization Analysis

The **Max. No. of Iterations** value is the maximum number of design variations that you want Optimetrics to solve during an optimization when using the **Sequential Nonlinear**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Programming (Gradient), **Sequential Mixed Integer NonLinear Programming**, **Quasi Newton (Gradient)**, or **Pattern Search (Search-based) Optimizer**. This value is a stopping criterion; if the maximum number of iterations has been completed, the optimization analysis stops. If the maximum number of iterations has not been completed, the optimization continues by performing another iteration, that is, by solving another design variation.

If the maximum number of iterations has not been reached, the optimizer performs iterations until the acceptable cost function is reached or until the optimizer cannot proceed as a result of other optimization setup constraints, such as when it searches for a variable value with a step size smaller than the minimum step size.

> **Note:**
>
> The **Genetic Algorithm** optimizer does not use the **Max. No. of Iterations** criteria.

To set the maximum number of iterations for an optimization analysis:

- Under the **Goals** tab of the *Setup Optimization* dialog box, type a value in the **Max. No. of Iterations** text box.

Modifying the Starting Variable Value for Optimization

A variable's starting value is the first value to be solved during the optimization analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each optimization setup.

> **Note:**
>
> If you choose to solve a parametric setup before an optimization analysis, a variable's starting value is ignored if a more appropriate starting value is calculated for it during the parametric analysis.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for the optimization analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this optimization analysis, and the current value set for the nominal model is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Optionally, click a new unit system in one of the **Units** text boxes.

> **Note:**
>
> To revert to the default starting value, clear the **Override** check box.

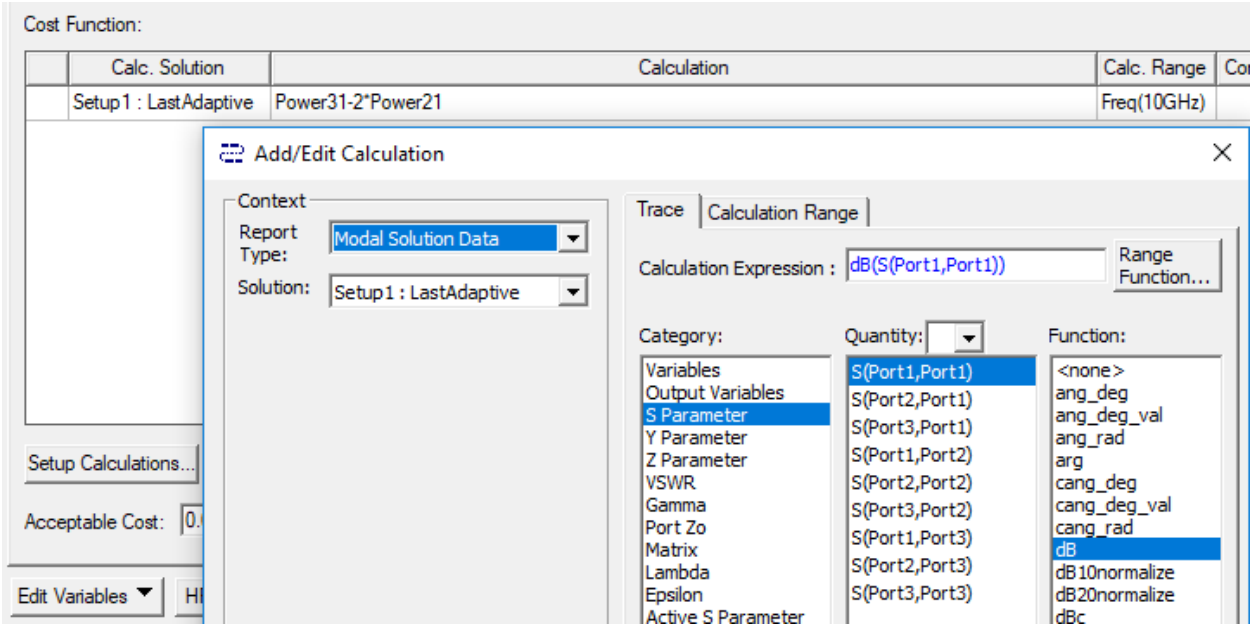
Cost Function

Optimetrics manipulates the model's design variable values to find the minimum location of the cost function; therefore, you should define the cost function so that a minimum location is also the optimum location. For example, if you vary a design to find the maximum transmission from *Wave Port 1* to *Wave Port 2* (S_{21}=>1), define the cost function to be -mag(S (WavePort2,WavePort1)).

When using the Quasi Newton optimizer, which is appropriate for designs that are not sensitive to noise, the best cost function is a smooth, second-order function that can be approximated well by quadratics in the vicinity of the minimum; the slope of the cost function should decrease as Optimetrics approaches the optimum value. The preferred cost function takes values between 0 and 1. In practice, most functions that are smooth around the minimum are acceptable as cost functions. Most importantly, the cost function should not have a sharp dip or pole at the minimum. A well designed cost function can significantly reduce the optimization process time.

The cost function is defined in the **Setup Optimization** dialog box or the **Design of Experiments** setup when you set up an optimization analysis. If you know the exact syntax of the solution quantity on which you want to base the cost function, you can type it directly in the **Calculation** text box. You can also use **Setup Calculations** to add a solution quantity via the *Add/Edit Calculation* dialog box, or to create an output variable that represents the solution quantity in the Output Variables dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Acceptable Cost

The acceptable cost is the value of the cost function at which the optimization process should stop; otherwise known as the *stopping criterion*.The cost function value must be equal to or below the acceptable cost value for the optimization analysis to stop. The acceptable cost may be a negative value.

Cost Function Noise

The numerical calculation of the electromagnetic field introduces various sources of noise to the cost function, particularly because of changes in the finite element mesh. You must provide the optimizer with an estimate of the noise. The noise indicates whether a change during the solution process is significant enough to support achievement of the cost function.

For example, if the cost function, *c*, is

$$c = 10000 \cdot |S_{11}|^2$$

where $|S_{11}|$ is the magnitude of the reflection coefficient, at the minimum, $|S_{11}|$ is expected to be very small, $|S_{11}| \approx 0$.

From the solution setup, the error in $|S_{11}|$ is expected to be $E_{S11} \approx 0.01$. The perturbed cost function is therefore

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$c_{perturbed} = 10000 \cdot (|S_{11}|_{min} + E_{S11})^2$$

Near the minimum, the error in the cost function E_c is given by

$$E_c = c_{perturbed} - c_{min} = 10000 \cdot (0.0 + 0.01)^2 - (10000 \cdot 0.0) = 1.0$$

Therefore, the cost function noise would be 1.0.

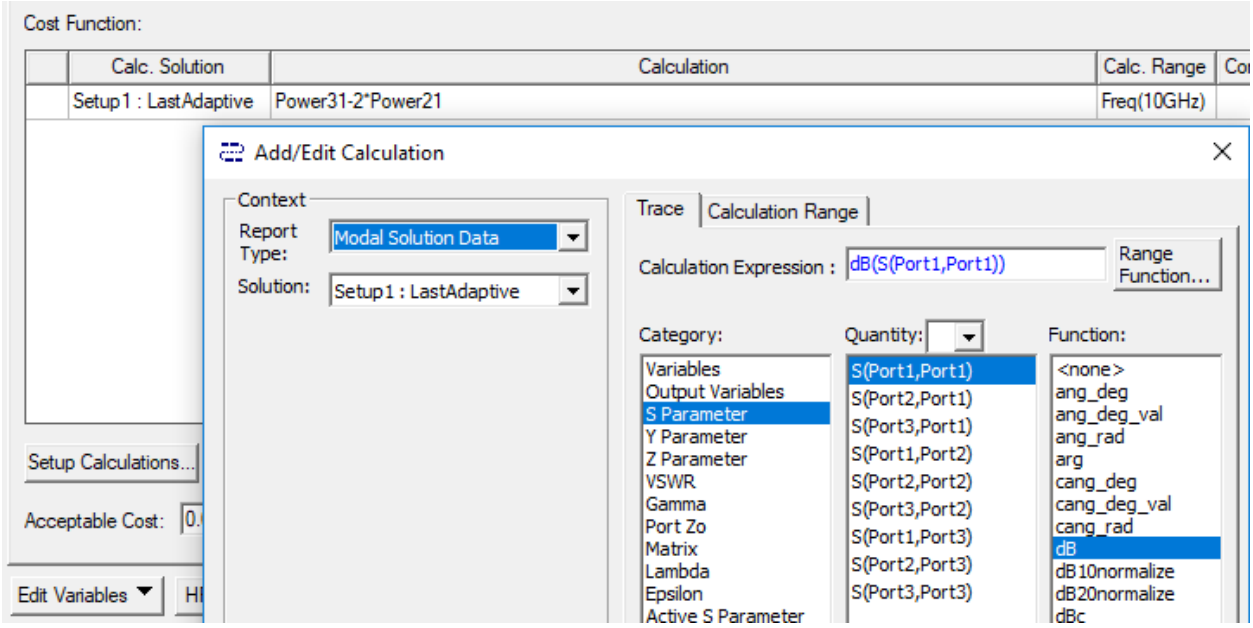
Adding a Cost Function

A cost function can include one or more goals for an optimization analysis. Optimetrics manipulates the model's design variable values to fulfill the cost function. The optimization will stop when the solution quantity meets the acceptable cost criterion.

Following is the general procedure for adding a cost function with a single goal:

1. Under the **Goals** tab of the *Setup Optimization* or *Design of Experiments* dialog box, click **Setup Calculations...**

   The Add/Edit Calculation dialog box is displayed.



2. In the *Add/Edit Calculation* dialog box, follow these general steps to set up a cost function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Set the **Context** for the calculation.
b. Choose the **Category** of available data type depending upon the Solution type of the design being optimized.
c. Select the **Quantity** to add to the **Calculated Expression** field. Available quantities depend upon the **Category** selection.
d. You may optionally make a selection from the function list to apply to the calculated expression.

> **Note:**
>
> Because Optimetrics works in SI values, you should use ang_deg_val or cang_deg_val functions which return unitless numbers in degrees rather than ang_deg or cang_deg, which return angular values in degree units but evaluate in radians in Optimetrics expressions. For example, you could Optimize successfully by changing the expression from abs(**cang_deg**(S(Port2,Port1))) to abs(**cang_deg_val**(S(Port2,Port1))). See the table of available functions and definitions.

e. When the **Calculation Expression** has the desired equation, click **Add Calculation** to add the expression to the cost function table.
f. Repeat to add additional calculations to the cost function or click **Done** to exit the **Add/Edit Calculation** dialog box and return to **Setup Optimization**.

3. To modify the **Solution** on which the calculation is based, click in the **Solution** column and select the solution from which the cost function is to be extracted from the drop-down menu.
4. To edit the calculation on which to base the cost function goal, select **Edit** from the drop-down menu.
5. In the **Condition** text box, click one of the following conditions from the drop-down menu:

<=	Less than or equal to
=	Equal to
>=	Greater than or equal to
Minimize	Reduce the cost function to a minimum value
Maximize	Identify a maximized condition

6. In the **Goal** text box, type the value of the solution quantity that you want to be achieved during the optimization analysis. If the solution quantity is a complex calculation, the goal value must be complex; two goal values must be specified. The **Minimize** and **Maximize** options do not require you to specify a **Goal** value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When Minimize is used as an optimization condition, the value of calculation is used as the cost (there is no target value to compare to). For maximize, the negative of calculation value is used as cost.

7. Optionally, if you have multiple goals and want to assign higher or lower priority to a goal, type a different value for the goal's weight in the **Weight** text box. The goal with the greater weight is given more importance. If the goal is a complex value, the weight value must be complex; two weight values must be specified. The weight value cannot be variable dependent.

   > **Note:**
   >
   > Click the **Edit Goal/Weight** button to open the *Edit Goal Value/Weight* dialog box, where you can modify weights for all goals simultaneously and set the **Goal Values** to expressions.

8. Specify other options (such as acceptable cost, noise, and number of passes), and then click **OK**.

The optimization stops when the solution quantity meets the acceptable cost criterion.

Specifying a Solution Quantity for a Cost Function Goal

When setting up a cost function, you must identify the solution quantity on which to base each goal. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters, and output variables.

1. Add a row (a goal) to the cost function table:
   a. Under the **Goals** tab of the *Setup Optimization* dialog box, click **Add**.

      A new row is added to the *Cost Function* table.

   b. In the **Solution** column, click the solution from which the cost function is to be extracted.
2. In the **Solution** text box, click the solution from which the solution quantity is to be extracted.
3. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

        The *Output Variables* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Add the expression you want to evaluate, click **Done**.
c. Click **Done** to close the *Output Variables* dialog box.

   In the *Setup Optimization* dialog box, the most recently created output variable appears in the *Calculation* text box.

d. To specify a different defined output variable, click the **Calculation** text box. It becomes a drop-down menu that displays all of the defined output variables. Click an output variable from the drop-down menu.

Setting the Calculation Range of a Cost Function Goal

The calculation range is the range within which you want a cost function goal to be calculated. It can be a single value or a range of values, depending on the solution or solution quantity selected for the goal.

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Cal. Range**.
2. In the **Variable** drop-down menu, click a variable.

   If you chose to solve a parametric setup during the optimization analysis, the variables swept in that parametric setup are available in the **Variable** drop-down menu. If you sweep a variable in the parametric setup that is also being optimized, that variable is excluded from the optimization.

   Other examples of available variables include frequency, if the solution quantity is an S-parameter quantity, and phi or theta, if the solution quantity is a radiated field quantity.

3. After you select a variable from the **Variable** drop-down menu, you can select a range of values for the calculation range as follows:
   a. Select **Range**.
   b. In the **Start** text box, type the starting value of the range.
   c. In the **Stop** text box, type the final value of the range.
4. To select a single value for the calculation range:
   a. Select **Single Value**.
   b. In the **Value** text box, type the value of the variable at which the cost function goal is to be extracted.
5. Click **Update** and then click **OK**.

Setting a Goal Value

A goal is the value you want a solution quantity to reach during an optimization analysis. It can be a real value or a complex value. If the solution quantity is a complex calculation, the goal value must be complex. You can type the goal value in the **Goal** text box. Alternatively, you can

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

use the *Edit Goal/Value Weight* dialog box to specify the goal value as a single value, a mathematical expression, or a value dependent on a variable such as frequency.

Specifying a Single Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Simple Numeric Value** from the **Type** list.
3. If the goal value is complex, click **real/imag** in the drop-down menu to the right if you want to specify the real and imaginary parts of the goal value.

   Alternatively, click **mag/ang** if you want to specify the magnitude and angle of the goal value.

4. Type the goal value in the **Goal Value** table.

   If the goal value is complex, type both parts of the goal value in the text box below the **Goal Value** heading. For example, type **1, 1** to specify the real part of the goal value as 1 and the imaginary part as 1.

   If the goal value is real, type a real goal value in the text box below the **Goal Value** heading.

5. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying an Expression as a Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Expression** from the **Type** list.
3. If you know the syntax of the mathematical expression or the existing output variable's name, type it in the text box below the **Goal Value** heading.

   Alternatively, if you want to create an output variable that represents the goal value, do the following:

   a. Click **Edit Expression**.

      The *Output Variables* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Add the expression you want to be the goal value, and then click **Done**.

The most recently created output variable is entered in the text box below the **Goal Value** heading.

4. Click **OK**.

The goal value you specified appears in the **Goal** text box.

Specifying a Variable-Dependent Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Variable Dependent** from the **Type** list.
3. Click a variable from the pull-down list to the left of the table.
4. Type the value of that variable in the first column of the table.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

5. Type a corresponding goal value for that variable value in the text box below the **Goal Value** heading.
6. Click **Add** to add another row to the reference curve.
7. Repeat steps 4, 5, and 6 until you have specified the reference curve.
8. Click **OK**.

The goal value is listed as being variable dependent in the **Goal** text box.

Goal Weight

If an optimization setup has a cost function made up of multiple goals, you can assign a different weight to each goal. The goal with the greater weight is given more importance during the cost calculation.

The error function value is a weighted sum of the sub-goal errors. Each sub-goal, at each frequency at which it is evaluated, gives rise to a (positive) error value that represents the discrepancy between the simulated response and the goal value limit. If the response satisfies the goal value limit, then the error value is 0. Otherwise, the error value depends on the differences between the simulated response and the respective goal limit. The error function may be defined as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\sum_{j}^{G} \frac{W_j}{N_j} \cdot \sum_{i}^{N_j} e_i$$

where

- G is the number of sub-goals.
- W_j is the weight factor associated with the j^{th} sub-goal.
- N_j is the number of frequencies for the j^{th} sub-goal.
- e_i is the error contribution from the j^{th} sub-goal at the i^{th} frequency.

The value of e_i is determined by the band characteristics, target value, and the simulated response value. The choices for band characteristics are <=, =, and >=.

Band Characteristics (Condition)	**e_i evaluation where s_i is the simulated response and g_i is the desired limit.**
<=	$e_i = \begin{cases} 0 & s_i \le g_i \\ s_i - g_i & s_i > g_i \end{cases}$
=	$e_i = \lvert s_i - g_i \rvert$
>=	$e_i = \begin{cases} 0 & s_i \ge g_i \\ g_i - s_i & s_i < g_i \end{cases}$

If the total error value is within the acceptable cost, the optimization stops.

Changing the Cost Function Norm

You can select the norm to be used in the calculation of the cost goal.

1. In the *Setup Optimization* dialog box, click the **Goals** tab.
2. Select **Show Advanced Options**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select a norm from the drop-down menu in the **Cost Function Norm Type** field. The options are **L1**, **L2**, and **Maximum**. **L2** is the default.

Explanation of L1, L2, and Max Norms in Optimization

When you set multiple goals for an optimization, the question arises as to what is actually going to drive the optimizer which is not a multi-objective one. The cost function will have a lot to do with it. The following discussion explains how the cost function is put together when there are multiple goals.

The general goal setting structure in Optimetrics is a logical sentence with the format:

$Calculation_{(i)}\ Condition_{(i)}\ Goal_{(i)}\ Weight_{(i)}$

The cost function that the optimizer uses is built based on the norm setting as long as there are multiple goals and none of those use the "minimize" or "maximize" conditions. Thus, in this case the error associated with each individual goal (weighted) is combined in a way that is specific for each norm type chosen.

For **L1** norm, the actual cost function uses the sum of absolute weighted values of the individual goal errors:

$$Cost = \sum_{1}^{N} \left| w_i \varepsilon_i \right|$$

For **L2** norm, the cost function computes the weighted sum of the squares of individual goal errors.

$$Cost = \sum_{1}^{N} w_i \varepsilon_i^2$$

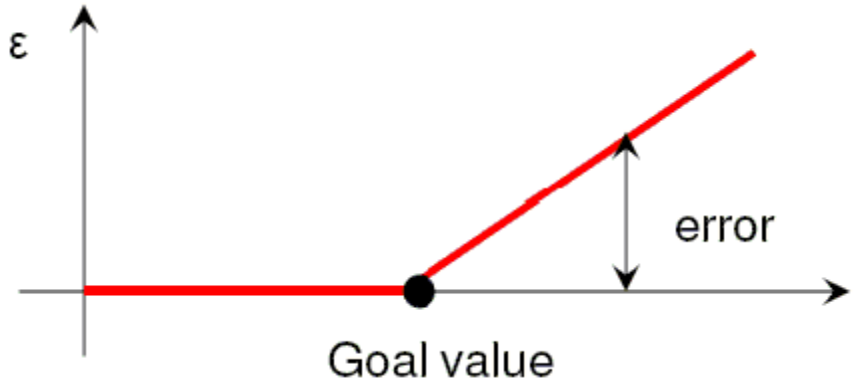
For the **Maximum** norm, the cost function uses the maximum among all the weighted goal errors, which means that cost is always less than zero:

$$Cost = \max_{1}^{N} W_i \cdot \varepsilon_i$$

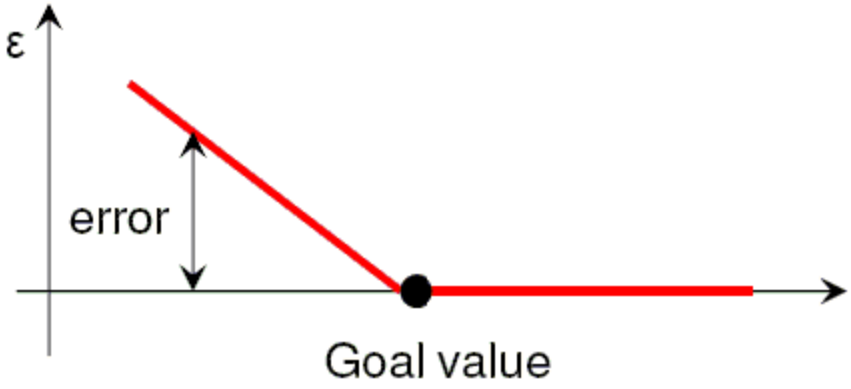
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all the above situations, N is the number of individual goals $w_i\varepsilon_i$ are individual weighting factors and residual error respectively. A minimization of the cost function is performed during optimization since it makes sense to minimize the error in the sense of the chosen norm type.

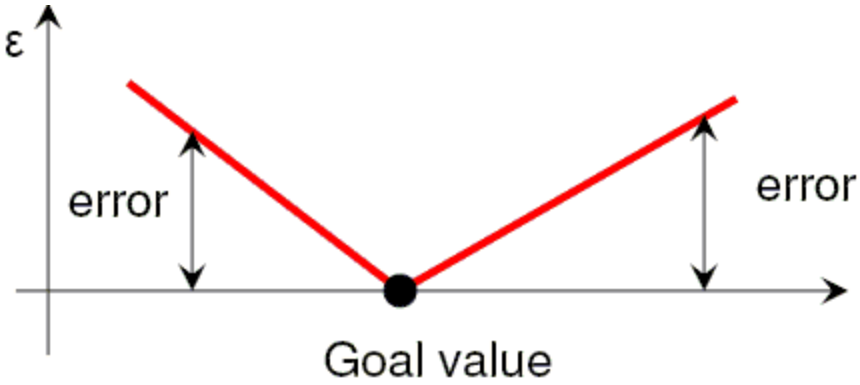
The graphical representation of the error is possible and depends upon the actual condition being used. If a "<" condition is used, the error can be represented as below:



If a ">" condition is used, the error can be represented as below:



If a "=" condition is used, the error is double-sided and can be represented as below:



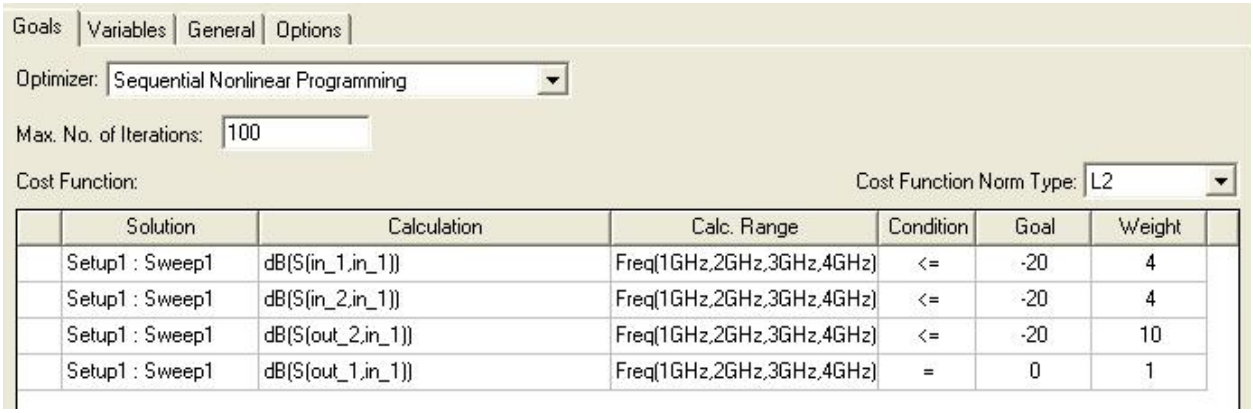
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios. Note that when using "minimize" or "maximize" settings for the condition there should be a single goal setting which in this case coincides with the cost function.

Example of a More Complex Cost Function

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

As an example of a more sophisticated cost function, consider the figure. It belongs to a connector simulated in HFSS with more than four ports.



The cost function given here concentrates only on a signal sent into port in_1. Suppose the specifications to be met are: reflection, backward cross talk and forward cross talk all smaller than or equal to -20 dB, of which the forward cross talk is the most important.

The first three entries in the cost function enforce those specifications, with the weight for the forward cross talk being a larger number than the other weights. The actual values for the weights are somewhat arbitrary and serve as examples only. For this cost function, as long as specifications are not met, the optimizer puts the most effort in getting the forward cross talk close to its specification. Once the three specifications have been satisfied, their contributions to the cost function become zero, and only the fourth entry remains. Remember that the connector has more than four ports, so satisfying the given specs does not guarantee maximum transmission.

The fourth line tries to maximize the transmission by asking for S(out_1, in_1) to be 0 dB. That will never be reached, but its presence forces the optimizer to improve the connector a bit beyond the specifications.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The cost function norm type specifies how the four lines are combined into one cost function with one value. With L1 and L2, all four contribute simultaneously, rather than only the largest of the four at any one time.

Setting the Min. and Max. Variable Values for Optimization

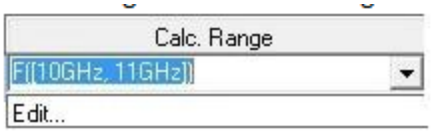
For every optimization setup, Optimetrics automatically sets the minimum and maximum values it will consider for a variable being optimized. Optimetrics sets a variable's minimum value equal to approximately 50% of its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 150% of the starting value. During the optimization analysis, variable values that lie outside of this range are not considered.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any Optimetrics analysis.

Edit Calculation Range

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

In the *Setup Optimization* dialog box, you can enter the Calc. Range Sweep Min/Max by directly editing the Calc. Range field or by accessing an *Edit Calculation Range* dialog box.



The edit field accepts the following forms of text:

- sweep that allows your to select different discrete values:

  discrete values, for example, F(10GHz, 11GHz)

  min/max range, for example, F([10GHz, 11GHz])

- editable sweep, which allows you to customize values (that is a sweep that has an enabled "edited" radio button in sweep selection dialog):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The min/max is used on top of selected values. For example, if you use the sweep dialog and choose "0 deg, 60 deg, 180 deg, 240 deg", then [60deg, 240deg] will select values "60 deg, 180 deg, 240 deg".

- sweep that uses a full range:

  all values, for example, Time(All)

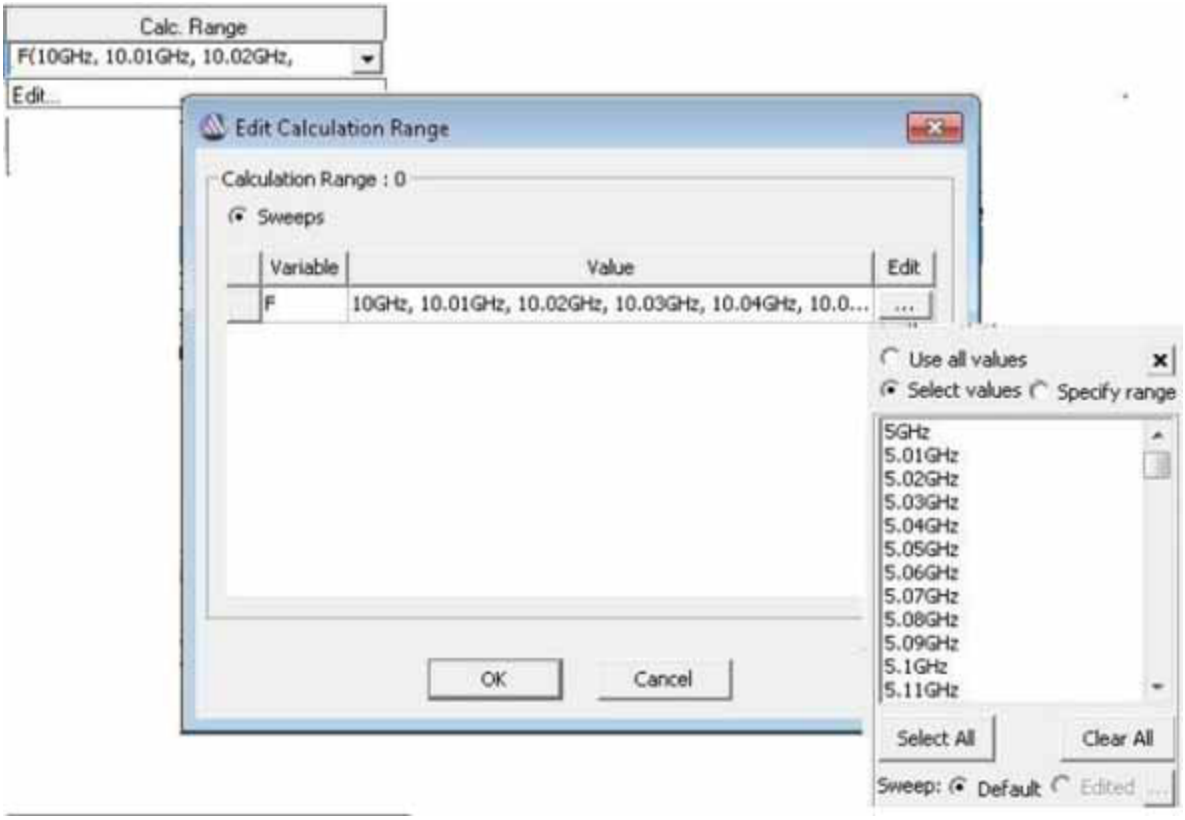
  min/max range, for example, Time([1ms, 2ms]) HFSS/MAXWELL/TWINBUILDER

- You solve 1 to 20 GHz step .1 and specify F[10.381GHz, 11.381GHz], it is equivalent to selecting values between 10.4GHz and 11.3GHz.
- You can specify multiple sweep values by separating those with comma ","

  For example, F(1GHz), cap(1pf, 1.2pf)

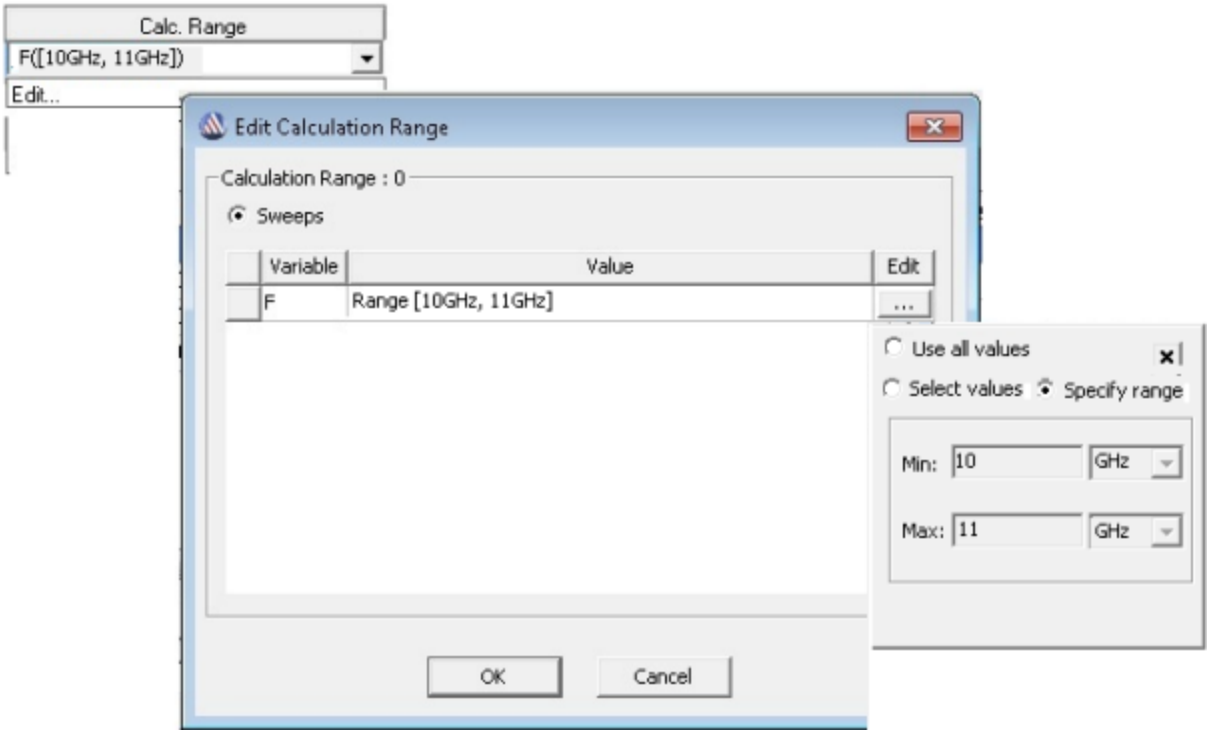
  For example, Distance(All), Freq([1ghz,2ghz]), Phase(0 deg)

If you click **Edit..** on the menu, you see the **Edit Calculation Range** dialog box. Click on the ellipsis [...] button to select radio buttons for Use all values, Select Values or Specify range.



This example shows that when you specify a range, how the range appears in the Calc. Range field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You could also enter the range directly in the Cal. Range field.

Overriding the Min. and Max. Variable Values for a Single Optimization Setup

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for optimization analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The *Override* option is now selected, indicating that the value you entered is used for this optimization analysis; the variable's current **Min** or **Max** value in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default minimum and maximum values, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Min. and Max. Variable Values for Every Optimization Setup

1. Make sure that the variable's minimum and maximum values are not being overridden in any single optimization setup.
2. If the variable is a design variable, do the following: Click **Mechanical**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project**>**Project Variables**.

   The *Properties* dialog box appears.
3. Select **Optimization**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.
5. Click **OK**.

   When Optimetrics solves an optimization setup, it does not consider variable values that lie outside of this range.

Step Size

To make the search for the minimum cost value reasonable, the search algorithm is limited in two ways. First, you do not want the optimizer to continue the search if the step size becomes irrelevant or small. This limitation impacts the accuracy of the final optimum. Second, in some cases you do not want the optimizer to take large steps either. In case the cost function is suspected to possess large variations in a relatively small vicinity of the design space, large steps may result in too many trial steps, which do not improve the cost value. In these cases, it is safer to proceed with limited size steps and have more frequent improvements.

For these two limitations, the optimizer uses two independent distance measures. Both are based on user-defined quantities: the minimum and maximum step limits for individual optimization variables. Since the particular step is in a general direction, these measures are combined together in order to derive the limitation for that particular direction.

The step vector between the i^{th} and $(i+1)^{th}$ iterate is as follows:

$$s_i = x_{i+1} - x_i$$

The natural distance measure is

$$\|s_i\| = \sqrt{s_i^T s_i}$$

which is the Euclidean norm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A more general distance measure incorporates some "stretching" of the design space:

$$\|s_i\|_D = \sqrt{s_i^T D^T D s_i}$$

where the matrix *D* incorporates the linear operation of the stretching of design space. The simplest case is when the *D* matrix is diagonal, meaning that the design space is stretched along the orthogonal direction of the base vectors.

The optimizer stops the search if

$$\|s_i\|_{D_{min}} < 1$$

where D_{min} consists of diagonal elements equal to the inverse of the **Min. Step** value assigned to the corresponding optimization variable. Similarly, the optimizer truncates steps for which

$$\|s_i\|_{D_{max}} > 1$$

where D_{max} has diagonal elements equal to the inverse of **Max. Step** values of the corresponding optimization variables.

Setting the Min. and Max. Step Sizes

For the Quasi Newton and Pattern Search optimizers, the step size is the difference in a variable's value between one solved design variation and the next. The step size is determined when Optimetrics locates the next design variation that should be solved in an effort to meet the cost function.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min Step** and **Max Step** columns, with default values for each variable to be optimized.
3. In the **Min Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** text box.
4. In the **Max Step** text box, type the maximum step size value. Optionally, modify the unit system in the **Units** text box.

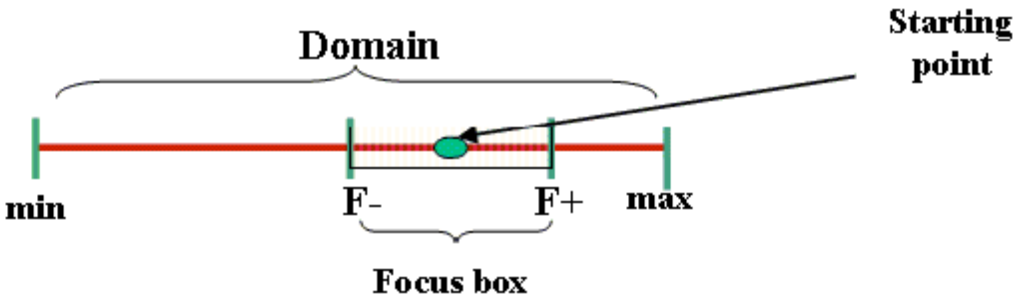
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

> **Tip:**
>
> A value of zero is recommended for the minimum step size.

Setting the Min and Max Focus

For the SNLP, SMINLP and Genetic Algorithm optimizers, the min focus and max focus criteria allow you to specify a sub-range of parameter values where the optimizer should look when performing the optimization. This focus box is where you suspect the optimal solution will be, so it is a hint for the optimizer.



- The domain limits the search. The domain = physical limits.
- The focus box does not limit the search. Rather, the Focus box = an initial guess of optimum search domain. The starting point is the center of the focus box, but the search does extend beyond the box.
- This focus must be inside the domain limits. Consequently, it has to be equal or smaller size. An error message is generated if you specify a focus outside the domain.
- The focus box must be at least one hundredth of the domain size. Otherwise, an error message is sent.

Equalizing the Influence of Different Optimization Variables

The optimizer seeks optimal values for the optimization variables. These variables are usually quantities with specified units. The change in one variable could be measured in [mm] and the change in other variable could be measured in [mA]. Instead of those units, the optimizer uses internal abstract units, so that a change in one variable changes the design behavior about as much as the same change in another variable, where changes are measured in the respective internal abstract units. When you define the focus box, the unit of the abstract internal unit is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined as the difference of the upper and lower focus limits. This way you can use the focus box to equalize the influence of different optimization variables on the design behavior.

Setting the Min and Max Focus Values

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min. Focus** and **Max. Focus** columns, with default values for each variable to be optimized.

   If you do not have an initial guess based on your knowledge of the problem, make the focus box equal to the domain; that is, the physical limits. This tells SNLP to search the entire decision space.

   - In the **Min. Focus** text box, type the minimum value of the focus range. Optionally, modify the unit system in the **Units** text box.
   - In the **Max. Focus** text box, type the maximum value of the focus range. Optionally, modify the unit system in the **Units** text box.
3. Click **OK**.

Solving a Parametric Setup Before an Optimization

Solving a parametric setup before an optimization setup is useful for guiding Optimetrics during an optimization.

To solve a parametric setup before an optimization setup:

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve before optimization.

   > **Note:**
   >
   > The parametric setup must include sweep definitions for the variables you are optimizing.

3. Select **Solve the parametric sweep before optimization**.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the cost value evaluated at each parametric design variation to determine the next step in the optimization analysis. This enables you to guide the direction in which the optimizer searches for the optimal design variation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Parametric Setup During an Optimization

Solving a parametric setup during an optimization analysis is useful when you want Optimetrics to solve every design variation specified in the parametric setup at each optimization iteration. A cost function goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during an optimization analysis:

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve during an optimization.
3. Select **Solve the parametric sweep during optimization**.
4. Optionally, you can adjust the sweep values to be used during the optimization.
    a. Click on the **Goal** tab, click **Setup Calculations** to specify a calculation.

       The *Add/Edit Calculation* dialog box is displayed.

    b. Click the **Calculation Range** tab.
    c. Click **Edit** for the sweep to be modified.
    d. In the pop-up dialog box, select the sweep values to use.
    e. Close the pup-up dialog box. Click **Done** to close the *Add/Edit Calculation*dialog box.

Automatically Updating a Variable's Value after Optimization

When Optimetrics finds an optimal variable value by solving an optimization setup, it can automatically update that variable's current value set for the nominal model to the optimal value.

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. Select **Update design parameters' values after optimization**.

   When optimization is complete, the current variable value for each optimized variable is changed to the optimal value.

Sensitivity Analysis Overview

During a sensitivity analysis, Optimetrics explores the vicinity of the design point to determine the sensitivity of the design to small changes in variables. The variables and their attributes define the design point, the problem around which the sensitivity analysis is performed.

When Optimetrics performs a sensitivity analysis, its goal is to calculate the second-order regression polynomials for all of the design's output parameters. The algorithm first determines an appropriate interval for each variable. The intervals are further sub-divided according to the available number of iterations and variables. If the primary output is not used, the specified initial displacement values define those intervals.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When all of the design calculations are complete, the second-order polynomials are fitted for all the output parameters. Optimetrics then reports the following quantities:

- Regression value at the current variable value
- First derivative of the regression
- Second derivative of the regression

> **Note:** Sensitivity Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Selecting a Primary Output

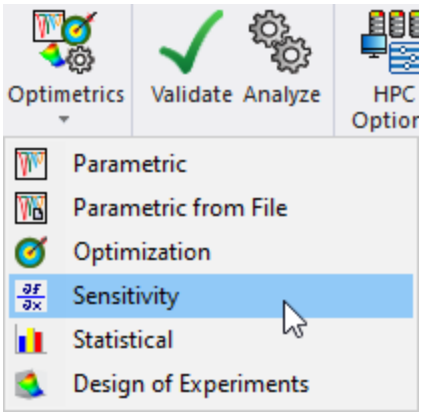
During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. Therefore, if you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

Setting Up a Sensitivity Analysis

Following is the general procedure for setting up a sensitivity analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes. You can create a sensitivity setup before defining variables but all variables must be defined before you start the sensitivity analysis.

1. Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Design Properties** dialog box.
2. Click **Mechanical** > **Optimetrics Analysis** > **Add Sensitivity** . You can also select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down menu under the Optimetrics icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Sensitivity Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations per variable value that you want the software to perform in the **Max. No. of Iterations/Sensitivity Variable** text box.
4. Set up an output parameter calculation, and select a Primary Output.
5. Specify the value of the design point at which the sensitivity analysis should stop in the **Approximate Error in Primary Output** text box.
6. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Initial Displacement (Initial Disp.)** for the analysis.

   You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.

7. In the **General** tab, specify whether Optimetrics should use the results of a previous parametric analysis or perform one as part of the optimization process.

   Checking the Optional **Worst Case Analysis** option does an extreme value analysis that focuses on the upper and lower boundaries of all the analyzed parameters. Some setup is required before performing Worst Case Analysis.

8. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

The following ***optional*** sensitivity analysis setup options can also be used:

- Modify the starting variable value.
- Modify the minimum and maximum values of variables that will be solved.
- Exclude variables from the sensitivity analysis.
- Set the initial displacement.
- Modify the values of fixed variables that are not being modified during the sensitivity analysis.
- Set linear constraints.
- Request that Optimetrics solve a parametric sweep **before** a sensitivity analysis.
- You can also request that Optimetrics solve a parametric sweep **during** a sensitivity analysis.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Setting the Maximum Iterations Per Variable

The **Max. No. of Iterations/Sensitivity Variable** value is the maximum number of design variations that Optimetrics solves per variable during a sensitivity analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the sensitivity analysis stops. If the maximum number of iterations has not been completed, the sensitivity analysis continues by performing another iteration, that is, by solving another design variation. It performs iterations until the approximate error in primary output value is reached or until Optimetrics cannot proceed as a result of other sensitivity setup constraints, such as when it searches for a variable value that is larger than the maximum value.

To set the maximum number of iterations for a sensitivity analysis:

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Max. No. of Iterations/Sensitivity Variable** text box.

Setting Up an Output Parameter

Following is the general procedure for adding an output parameter to a sensitivity setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click **Setup Calculations** to open the *Add/Edit Calculations* dialog box.
2. In the *Add/Edit Calculations* dialog box, set up output parameter calculations to be evaluated for sensitivity.
3. To modify the solution from which the output parameter is to be extracted, click in the **Solution** column and select from the options in the pop-up list.
4. You can modify the Calculation specified by clicking on the output parameter in the table and selecting **Edit**.
5. For output parameters based on swept variable, you must choose a single value in the **Calculation Range** at which to evaluate the output parameter.
6. If the output parameter is based on a swept variable, in the **Calculation Range** column, set the value of the variable at which the output parameter is to be computed.
7. If you have more than one output parameter, select **Primary Output** if you want Optimetrics to use the output parameter to base its selection of solved design variations.

> **Note:**
>
> During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. If you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

Specifying a Solution Quantity for an Output Parameter

When setting up an output parameter, you must identify the solution quantity on which to base the output parameter. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters; and output variables.

The *Add/Edit Calculation* dialog box allows you to define the mathematical equation for one or multiple output parameters. To set up an output parameter:

1. In the **Context** section of the dialog box:
   - Select the **Report Type** with a pull-down selection list containing the available types for this design.
   - Select the **Solution** from the drop down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the **Geometry** from the drop down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.

2. The **Output Variables** button opens the Output Variables dialog box allowing you to create special output variables to be used in the output parameter.
3. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the output parameter. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
   - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
   - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
   - Select a **Function** to apply to the value in the calculated expression.
   - For swept variables, the **Range Function** button opens the **Set Range Function** dialog to apply functions to the expression that apply over the sweep range.
4. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
5. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Sensitivity Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculation Expression** and using the **Add Calculation** button.
6. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
7. Click **Done** to return to the *Setup Sensitivity Analysis* dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Setting the Calculation Range of an Output Parameter

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is to be extracted. For a sensitivity setup, the calculation range must be a single value. If you specified that the solution quantity be extracted from a frequency sweep solution, by default, Optimetrics uses the starting frequency in the sweep. If you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup.

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Edit Calculation Range* dialog box appears.

2. In the **table**, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the sensitivity analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a variable in the parametric setup that is also a sensitivity variable, that variable is excluded from the sensitivity analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.
4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable, and return to the *Setup Sensitivity Analysis* dialog box.

Modifying the Starting Variable Value for Sensitivity Analysis

The design point of the sensitivity analysis is the starting value of the sensitivity variable and is usually the first variation to be solved. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify the design point for each sensitivity setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for the sensitivity analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Min. and Max. Variable Values

For every sensitivity setup, Optimetrics automatically sets the minimum and maximum values that it will consider for a sensitivity variable. Optimetrics sets a variable's minimum value equal to approximately one-half its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 1.5 times the starting value. During sensitivity analysis, variable values outside this range are not considered.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for sensitivity analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the variable's current **Min** or **Max** value set in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

   To revert to the default minimum and maximum values, clear the **Override** option.

Changing the Min. and Max. Variable Values for Every Sensitivity Setup

1. Make sure the variable's minimum and maximum values are not being overridden in any sensitivity setup.
2. If the variable is a design variable, do the following: Click **Mechanical**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.

3. Select **Sensitivity**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   When Optimetrics solves a sensitivity setup, it does not consider variable values that lie outside of this range.

Setting the Initial Displacement

The initial displacement is the difference in a variable's starting value and the next solved design variation. During the sensitivity analysis, Optimetrics does not consider an initial variable value that is greater than this step size away from the starting variable value.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.
2. Optimetrics displays the **Initial Disp.** column, with default values for each sensitivity variable.
3. In the **Initial Disp.** text box, type the initial displacement value. Optionally, modify the unit system in the **Units** text box.

Solving a Parametric Setup before a Sensitivity Analysis

Solving a parametric setup before a sensitivity setup is useful for guiding Optimetrics in a sensitivity analysis.

To solve a parametric setup before a sensitivity setup:

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve before the sensitivity setup from the **Parametric Analysis** drop-down menu.

   > **Note:**
   >
   > The parametric setup must include sweep definitions for the sensitivity variables.

3. Select **Solve the parametric sweep before analysis**.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the results (of the solution calculation you requested under the **Goals** tab of the *Setup Sensitivity* dialog box) to determine the next design variation to solve for the sensitivity analysis.

Solving a Parametric Setup during a Sensitivity Analysis

Solving a parametric setup during a sensitivity analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each sensitivity analysis iteration. An output parameter goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during a sensitivity analysis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the sensitivity analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Performing Worst Case Analysis

Two popular worst case analysis techniques can be implemented: extreme value analysis and Monte Carlo analysis.

Some setup is required before performing worst case analysis. First, identify uncertainties in design and create a local or project variable for each of them. Second, determine the variation range of each variable (Min and Max) – its statistical distribution is optional. Third, determine a measurement of performance, especially for extreme value analysis.

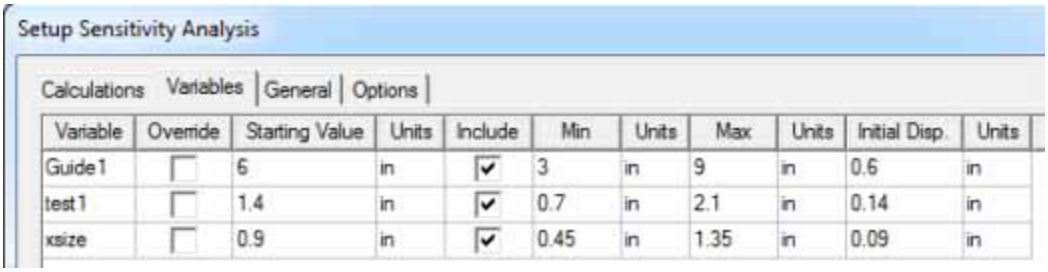
1. **Extreme value analysis:**

   This is one of the most popular methods to estimate worst-case performance. To start, a sensitivity analysis is performed. The results (sensitivities/first derivative) allow us to pick an extreme value (upper or lower bound) for each variable. The corresponding simulation result is used to predict upper and lower bound of performance. The assumption is that extreme performance is reached at boundary value (note that in certain cases, making such an assumption is not valid).

   To perform an extreme value analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics** > **Sensitivity**.

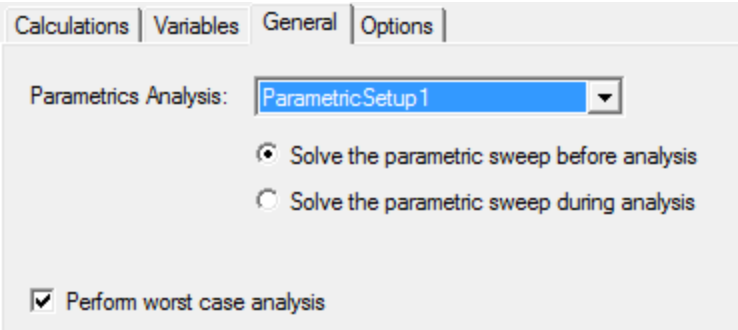
   - Setup:

     First, include variables in a sensitivity analysis. To do so, go to your list of variables, under Sensitivity, specify Min and Max values of each variable, and check 'Include'. The following figure shows an example with three variables included.



     Second, follow **Setting Up a Sensitivity Analysis** in the help. During this procedure, set your performance measurement in **Calculations** tab; check all variables in **Variables** tab; check **Perform worst case analysis** in **General** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Checking **Perform worst case analysis** calculates 1st derivatives for each variable. If we have three variables and for Var1 first derivative is negative, Var2 1st derivative is positive, Var3 1st derivative is positive, then for Worst Case Analysis, we request two more variations:

Var1@minimum, Var2@maximum, Var3@maximun

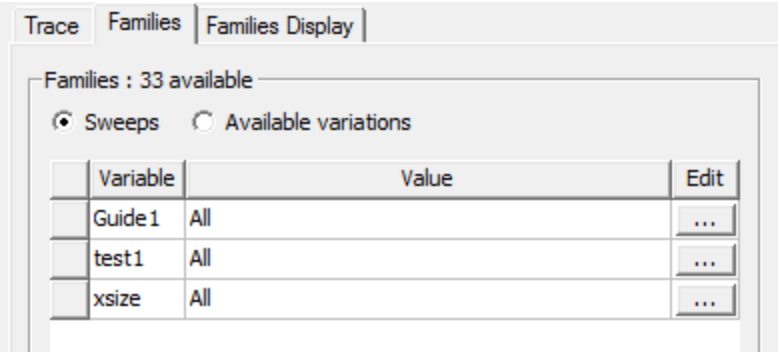
and Var1@maximum, Var2@minimum, Var3@minimum

- After setup, analyze your Sensitivity Setup under Optimetrics.
- To view worst-case result (upper bound only):

  Create a Data Table report.

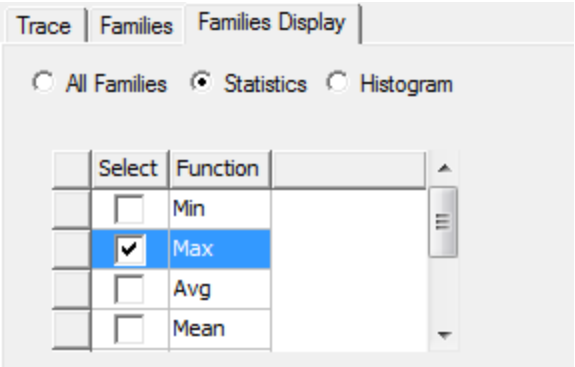
  Note:

  Under the Context pane in the **Report** dialog box, select matching Solution and Optimetrics setup. Under the **Families** tab, change the setting to 'All' under 'Value' of each variables.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under the **Families Display** tab, select **Statistics**and then check **Max**.



The associated data table shows the Max values.

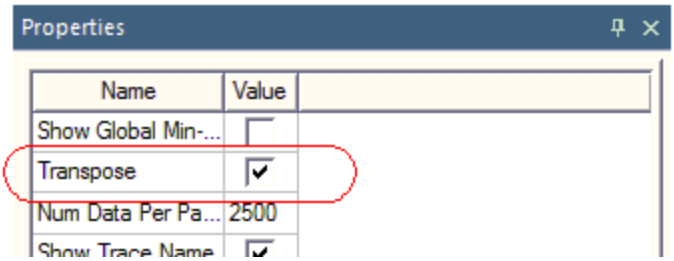
	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Max
1	3.000000	-0.000000
2	4.000000	-0.000000
3	5.000000	-0.105323
4	6.000000	-0.248322
5	7.000000	-0.267220
6	8.000000	-0.202039
7	9.000000	-0.298840

To see the corresponding variable values, select 'All Families' under the **Families Display** tab, and locate the max value to see value of variables.

	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Guide1='3in' test1='2.1in' xsize='0.45in'
1	3.000000	-0.545922
2	4.000000	-2.300003
3	5.000000	-10.634331
4	6.000000	-4.586505
5	7.000000	-5.950253
6	8.000000	-9.019574
7	9.000000	-3.210097

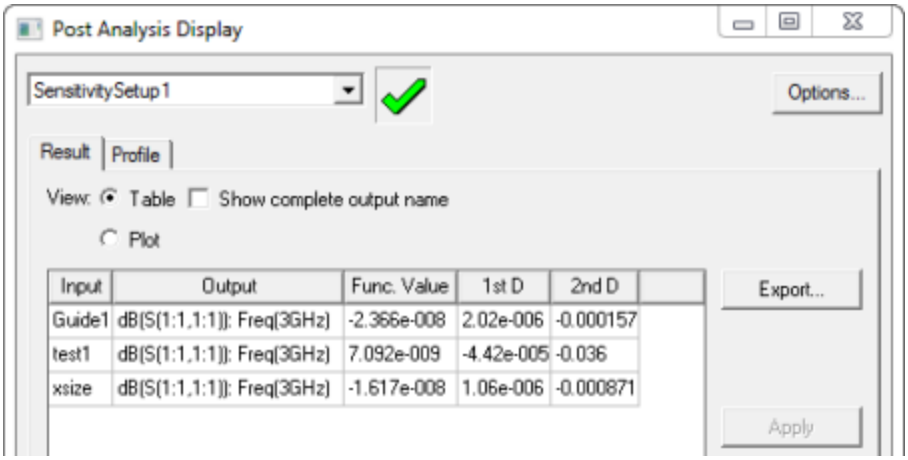
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> Transpose the table for an alternate view (double-click data table to view the **Properties** window, and on the **Data Table** tab check **Transpose**).



To estimate lower bound of performance:

First, see the sensitivity of performance to the respect of each variable. (Follow the documentation for Viewing Output Parameter Results for a Sensitivity Analysis). Identify the variables that have major influence over performance.



Second, in your project, manually change these variables to the corresponding bounds: choose Min for positive 1st derivative and Max for negative 1st derivative.

2. **Monte Carlo analysis:**

   This method does not assume a circuit is linear - better accuracy is achieved with more iterations. The cost is computing time and resources.

   To perform Monte Carlo Analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics> Statistical**. Follow the documentation on Statistical Analysis to set

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this up. The upper and lower bound of performance can be found on the edge of performance distribution.

> **Tip:**
>
> If distribution of performance is not of interest, set all variables as uniformly distributed.

Statistical Analysis Overview

Statistical analysis allows you to explore the effects of random combinations of values of selected variables on selected global or local available analysis results. Therefore, before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis. For each variable you must specify the type of distribution (Uniform, Gaussian, Lognormal or User Defined) and the corresponding parameters of the selected distribution.

> **Note:** A statistical analysis is limited to using a maximum of 30 variables.

In addition to specifying the variables to be used in the statistical analysis and the parameters of the chosen distribution, the output quantities of interest also need to be specified. These quantities can be global ones such as previously defined parameters (Force/torque, inductance / capacitance, etc), other named quantities, quantities defined in the field calculator as global (such a domain integral of a certain field quantity) or local (such as field value at a certain location). The calculations to be performed during the statistical analysis are specified during setup, in a manner similar to other types of analysis in Optimetrics.

Following the analysis the statistical distribution of the output quantities can be visualized in histogram format. To access available reports, after the statistical analysis is complete, right click the respective Statistical analysis setup and select **View Analysis Result**.

> **Note:**
>
> Statistical Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

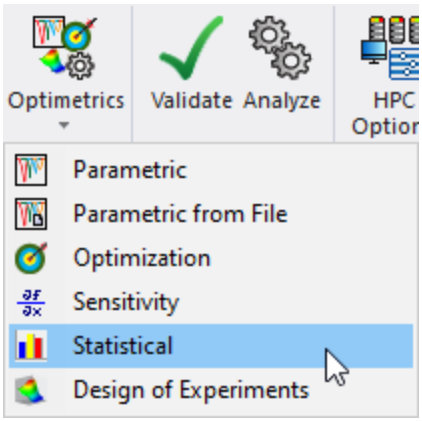
Setting Up a Statistical Analysis

Following is the general procedure for setting up a statistical analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

whole process for minor changes. You can create a statistical setup before defining variables but all variables must be defined before you start the statistical analysis.

1. Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** dialog box.
2. Click **Mechanical**> **Optimetrics Analysis** > **Add Statistical** , or select the **Simulation** tab of the ribbon and, under the Optimetrics icon, select **Statistical** from the drop-down menu:



The **Setup Statistical Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations you want Mechanical to perform in the **Maximum Iterations** text box.
4. If you want to specify an initial seed value, select the **Specify initial seed value** check box and enter a positive value in the text box. Each different seed value creates a new statistical sequence.
5. Specify a solution quantity to evaluate.
6. In the **Calculation** text box, set the value at which the solution quantity is to be computed.
7. Optionally, modify the distribution criteria to be used.
8. The following *optional* statistical analysis setup options can also be used:
   - Modify the starting variable value.
   - Exclude variables from the statistical analysis.
   - Modify the values of fixed variables that are not being modified during the statistical analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Request that Optimetrics solve a parametric sweep during a statistical analysis.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

9. If you want to save the field solution data for the design variations solved during analysis, select **Save Fields**.

Setting the Maximum Iterations for a Statistical Analysis

The **Maximum Iterations** value is the maximum number of design variations Optimetrics solves during a statistical analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the analysis stops. If the maximum number of iterations has not been completed, Optimetrics continues by performing another iteration, that is, by solving another design variation.

To set the maximum number of iterations for a statistical analysis:

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Maximum Iterations** text box.

Specifying the Solution Quantity to Evaluate for Statistical Analysis

When you add a statistical setup, you can identify one or more solution quantities to evaluate. The solution quantities are specified by mathematical expressions that are composed of basic quantities. You can see the distribution of the solution quantities in the results.

1. In the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click **Setup Calculations**.

   The **Add/Edit Calculations** dialog box is displayed, allowing you to define one or more mathematical expressions for statistical evaluation.

2. In the **Context** section of the dialog box:
   - Select the **Report Type** with a drop-down selection list containing the available types for this design.
   - Select the **Solution** from the drop-down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the **Geometry** from the drop-down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
   - When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The **Output Variables** button opens the *Output Variables* dialog box allowing you to create special output variables to be used in the output parameter.
4. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the solution quantities. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
    - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
    - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
    - Select a **Function** to apply to the value in the calculated expression.
    - For swept variables, the Range Function button opens the **Set RangebFunction** dialog to apply functions to the expression that apply overbthe sweep range.
5. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
6. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Statistical Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculated Expression** and using the **Add Calculation** button.
7. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
8. Click **Done** to return to the Setup Statistical Analysis dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Setting the Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is extracted. For a statistical setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you specified that the solution quantity be extracted from a frequency sweep solution, Optimetrics will use the starting frequency in the sweep by default. The calculation range should be set during the setup of the solution quantity for statistical evaluation. In order to modify the calculation range, do the following:

1. Under the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Edit Calculation Range* dialog box appears.

2. In the table, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the statistical analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a variable in the parametric setup that is also a statistics variable, that variable is excluded from the statistics analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.
4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable and return to the *Setup Statistical Analysis* dialog box.

Setting the Distribution Criteria

For every statistical setup, Optimetrics automatically sets the distribution criteria to be uniform within a 10% tolerance of the variable's starting value. You can modify the distribution type and criteria for a single statistical setup or for every statistical setup.

Overriding the Distribution Criteria for a Single Statistical Setup

To override the default distribution criteria for a single statistical setup:

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for statistical analysis are listed.

2. Check or clear the **Include** check box for each variable to define the specific variables to be varied in the statistical analysis setup.
3. For each included variable, select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined** in the **Distribution** column for the variable you want to override.

   If you changed the distribution type, the **Override** option is now selected. This indicates that the distribution type you selected is to be used for this optimization analysis; the current distribution type selected for the variable in the nominal design is ignored in this statistical analysis.

   - Alternatively, you can select the **Override** option first, and then select a different distribution type in the **Distribution**text box.

4. Optionally, if you want to change the distribution criteria, click in **Distribution Criteria** column for the variable you want to override.

   The *Edit Distribution* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If the distribution type is **Gaussian**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
    b. Type the mean value of the distribution in the **Mean** text box.
    c. Type the standard deviation of the distribution in the **Std Dev** text box.

    The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.
6. If the distribution type is **Uniform**, Eeter a tolerance value in the text box.

    The design variations are solved within the tolerance range of the starting value, using an even distribution.
7. If the distribution type is **Lognormal**, do the following:
    a. Enter the cutoff probability in the **Cutoff Probability** text box.
    b. Enter the sigma value of the distribution in the **Sigma** text box and select a unit from the pull-down.
    c. Enter the m value of the distribution in the **M** text box.
    d. Enter the theta value in the Theta text box and select a unit from the pull-down.
8. If the distribution type is User Defined, do the following:
    a. Enter the cutoff probability in the **Cutoff Probability** text box.
    b. Click **Edit XY Data** to open the *Edit Datasets* dialog box in which you can select an existing dataset, or create a new one.
9. By default, all variables are set to sample using **Latin Hypercube** sampling. This sampling method provides for greater variability than random sampling by keeping track of chosen samples and guaranteeing that samples cannot be repeated. You may revert to random sampling by clearing the check box in the **Latin Hypercube** column for any desired variable.
10. Click **OK**.

To revert to the default distribution settings, clear the **Override** option.

Changing the Distribution Criteria for Every Statistical Setup

To change the default distribution criteria for every statistical setup:

1. Make sure that the variable's distribution criteria are not being overridden in any statistical setup.
2. If the variable is a design variable, do the following: On **Mechanical** > **Design Properties**, select **Statistics**.

    If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Properties* dialog box appears.

3. Select **Statistics**.
4. Click in the **Distribution** column for the variable you want to change, and then select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined**.
5. Optionally, if you want to change the distribution criteria, click in the **Distribution Criteria** column for the variable you want to change.

   If the distribution type is **Gaussian**, the *Gaussian Distribution* dialog box appears. If the distribution type is **Uniform**, the *Uniform Distribution* dialog box appears.

6. If the distribution type is **Gaussian**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Type the standard deviation of the distribution in the **Std Dev** text box.

   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

7. If the distribution type is **Uniform**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type mean and tolerance values in the corresponding text boxes.

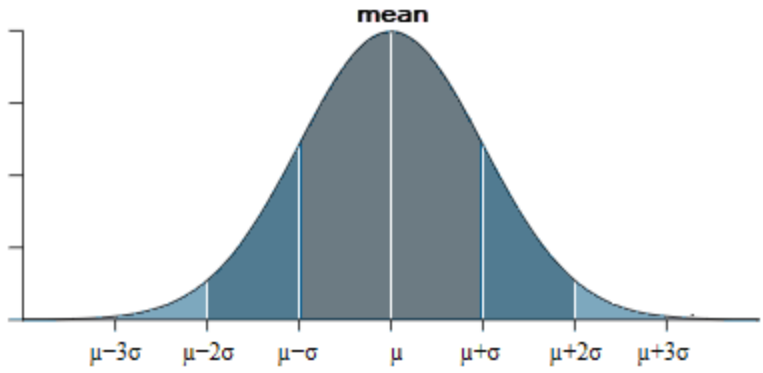
   The design variations are solved within the tolerance range of the starting value, using an even distribution.

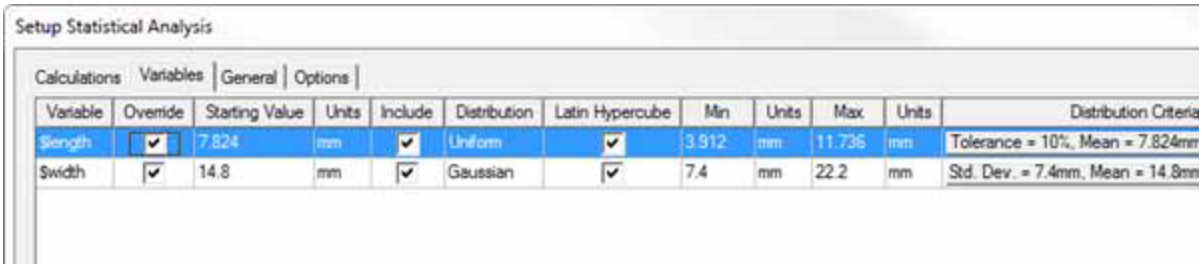
8. If the distribution type is **Lognormal**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type values for Sigma, M, and Theta in the corresponding text boxes.
9. If the distribution type is **User Defined**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Click **Edit XY Data** to open the *Edit Dataset* dialog box.
   c. Either type or import the X and Y data values for the distribution in the *Edit Dataset* dialog box.
10. Click **OK**.

Statistical Cutoff Probability

The cutoff probability values affects the Gaussian distribution criteria. This is a value =>0 and < 0.1. The design variations are solved using a Gaussian distribution using a lower limit cutoff probability and specified mean and standard deviation values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Uniform distributions such as **length** variable above use only the **Tolerance** value and do not have a cutoff probability.

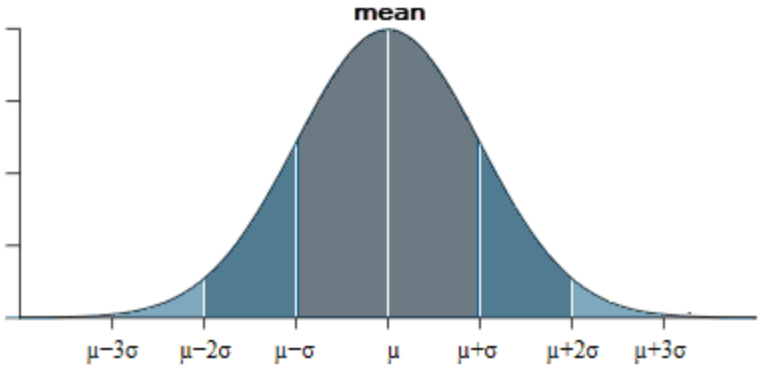
Edit Distribution

When setting the distribution type for a variable, you have the option of changing the distribution parameters from the default values.

1. If the distribution type is **Gaussian**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
    b. Type the mean value of the distribution in the **Mean** text box.
    c. Type the standard deviation of the distribution in the **Std Dev** text box.

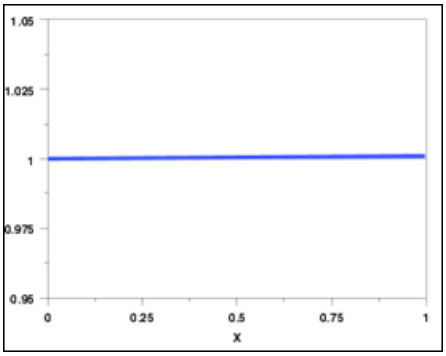
    The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If the distribution type is **Uniform**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Type the mean value of the distribution in the **Mean** text box.
    c. Enter the tolerance in the **Tolerance** text box.

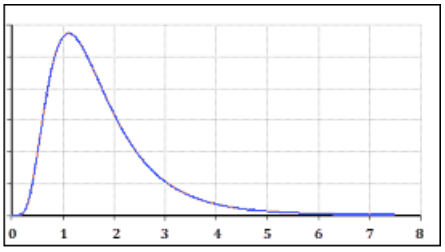
    The design variations are solved within the tolerance range of the starting value, using an even distribution.



3. If the distribution type is **Lognormal**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Enter the shape parameter of the distribution in the **Sigma** text box.
    c. Enter the scale parameter in the **M** text box. The scale parameter should be set to 1 for the standard lognormal distribution.
    d. Enter the location parameter value for **Theta** in the text box. The value for a standard lognormal distribution is 0.

    The design variations are solved with a logarithmic distribution using the shape, scale and location parameters provided.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. If the distribution type is **User Defined**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Select the **Edit XY Data** button to manually define the data distribution using datasets.

Modifying the Starting Variable Value for Statistical Analysis

A variable's starting value is the first value that is solved during the statistical analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each statistical setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables selected for the statistical analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this statistical analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Parametric Setup During a Statistical Analysis

Solving a parametric setup during a statistical analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each statistical analysis iteration.

To solve a parametric setup during a statistical analysis:

1. In the *Setup Statistical Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the statistical analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Using Design of Experiments

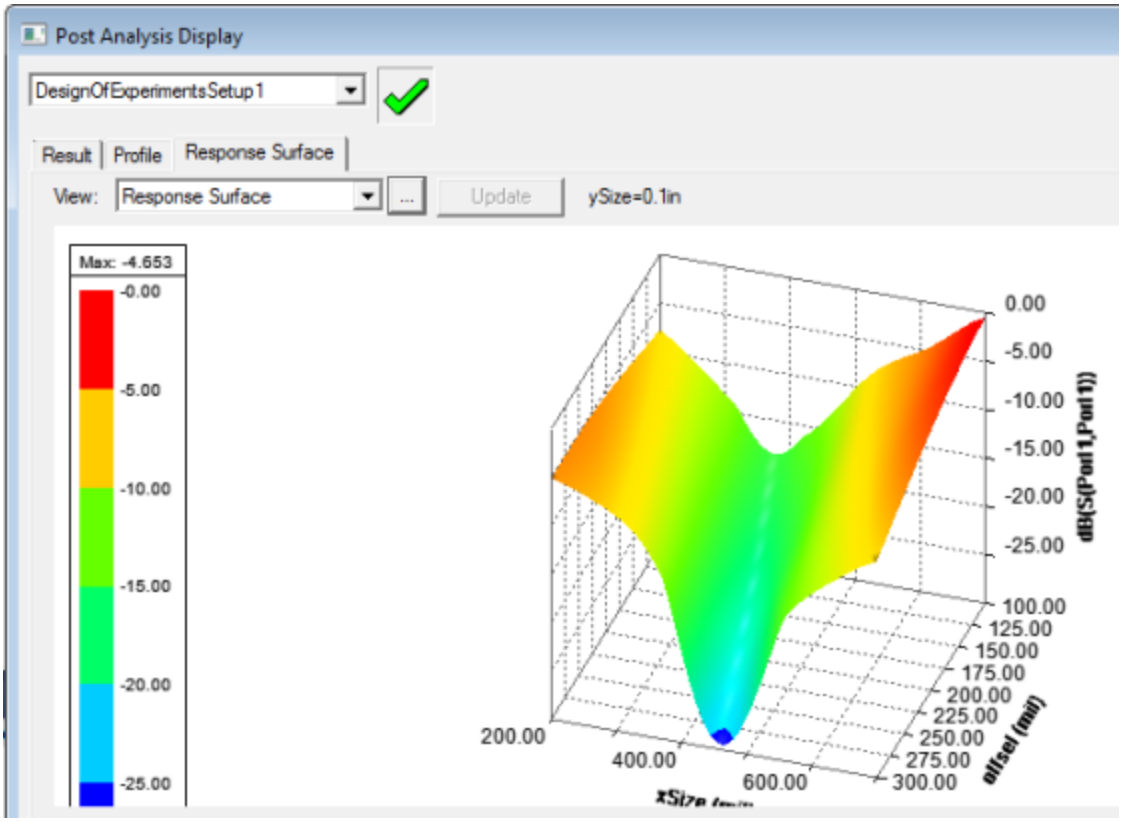
Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems. Design of Experiments plus a mathematical approximation of output parameters lets you

- reduce the number of simulations
- interactively explore the design space before running optimization

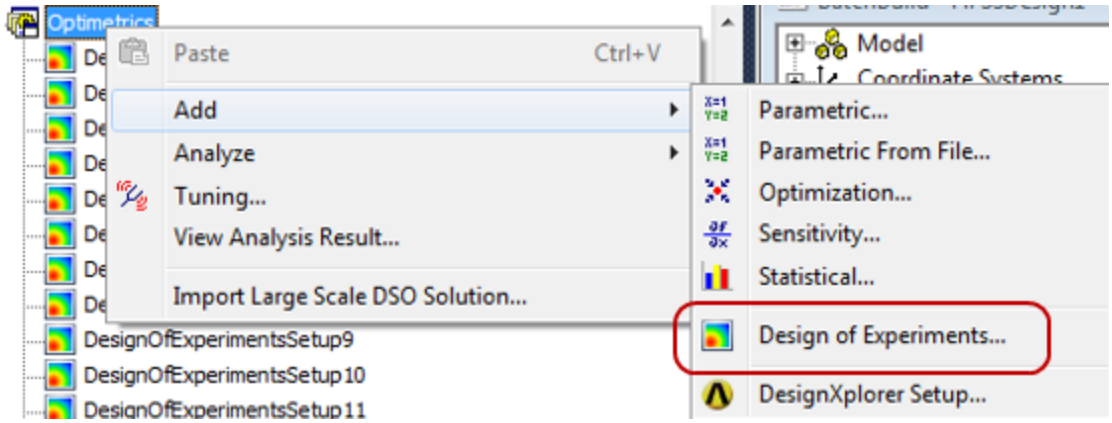
Design of Experiments describes the relationship between the design variables and the performance of the product by using Design of Experiments (DOE), combined with response surfaces. DOE and response surfaces provide all of the information required to achieve Simulation Driven Product Development. Once the variation of the performance with respect to the design variables is known, it becomes easy to understand and identify all changes required to meet the requirements for the product.

The goal is to create a response surface by interpolating through calculated points (a best curve fit). For each design, you can create a response surface for each output parameter. Once the response surfaces are created, you can share the information can in easily understandable terms: curves, surfaces, sensitivities, etc. They can be used at any time during the development of the product without requiring additional simulations to test a new configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

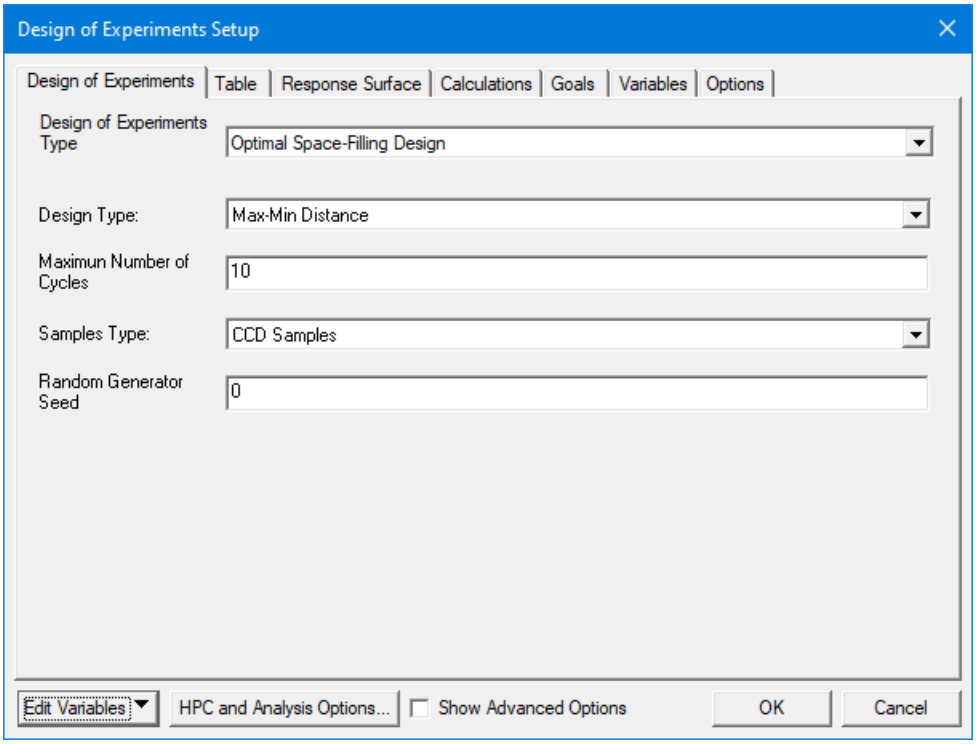


The Design of Experiments feature is integrated inside Electronics Desktop. Combined with Electronics Desktop‘s distributed solve feature, you can build the response surfaces from the DOE variation table much faster.



Selecting **Design of Experiments** under **Optimization** opens a window with several tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the Design of Experiments setup, you select the DOE type, select the Response Surface, specify goals, view and include variables.

There are a wide range of DOE algorithms or methods available in engineering literature. These techniques all have one common characteristic: they try to locate the sampling points such that the space of random input parameters is explored in the most efficient way, or obtain the required information with a minimum of sampling points. Sample points in efficient locations only reduce the required number of sampling points and increases the accuracy of the response surface generated. For more information on the available types of DOE, see Design of Experiments Types.

Once you have set up your input parameters, you can update the DOE, which submits the generated design points to the analysis system for solution. Design points are solved simultaneously if the analysis system is set up to do so; sequentially, if not. After the solution is complete, you can update the Response Surface cell, which generates response surfaces for each output parameter based on the data in the generated design points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Requirements and recommendations regarding the number of input parameters vary according to DOE type. For more information, see Number of Input Parameters for DOE Types.

If you change the Design of Experiments type after doing an initial analysis and preview the Design of Experiments Table, any design points generated for the new algorithm that are the same as design points solved for a previous algorithm will appear as up-to-date. Only the design points that are different from any previously submitted design points need to be solved.

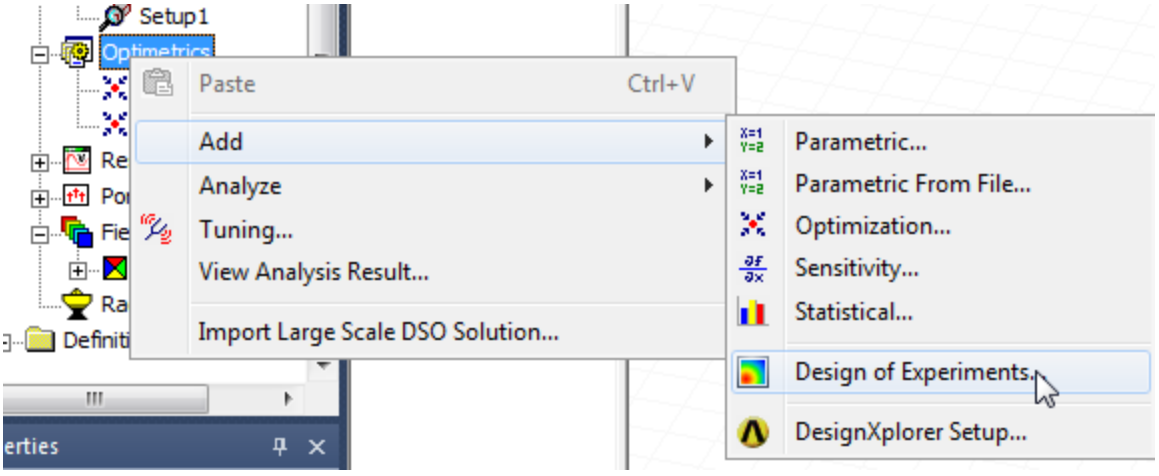
You should set up your DOE Properties before generating your DOE Design Point matrix. The following topics describe setting up and solving your Design of Experiments, and viewing the results.

Setting Up Design of Experiments in Ansys Electronics Desktop

The process for setting up a Design of Experiments analysis in Ansys Electronics Desktop follows:

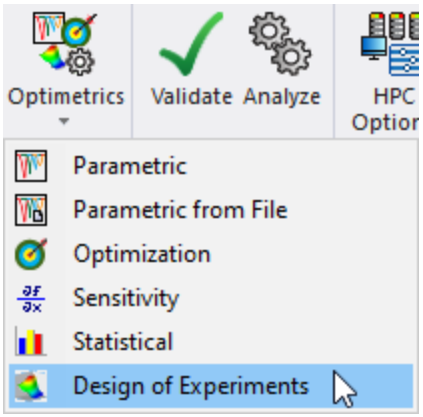
Add the Design of Experiments Setup

Right-click the Optimetrics icon in the Project tree and select **Add** > **Design of Experiments**. You can also use **Mechanical** > **Optimetrics Analysis** > **Add Design of Experiments**.

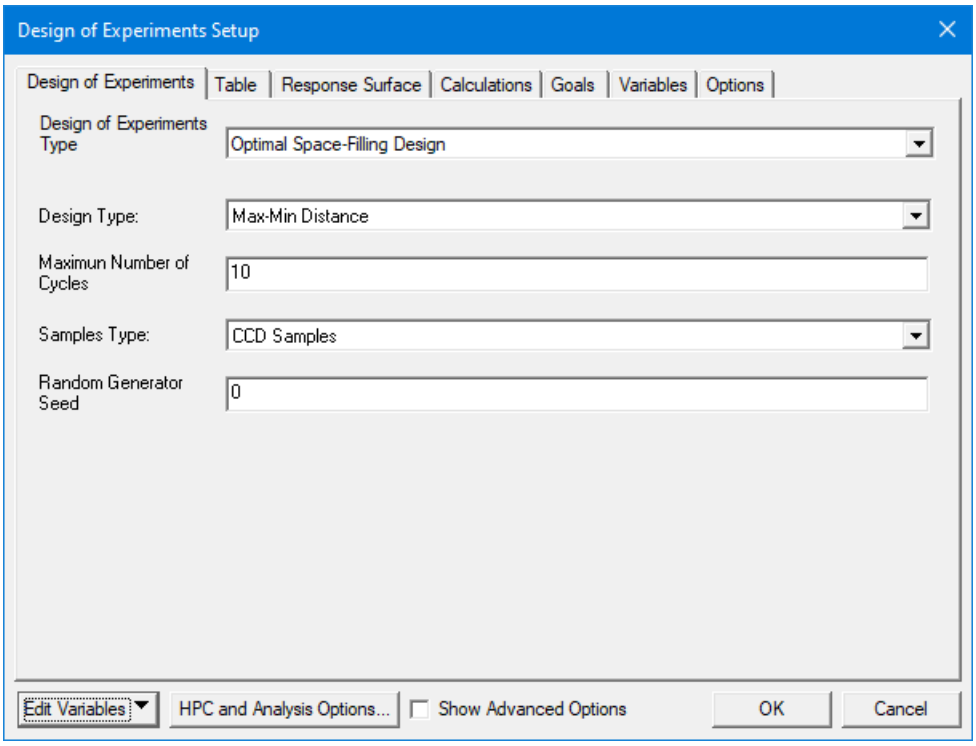


You can also select the **Simulation** tab and under the Optimetrics icon, select Design of Experiments from the drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Design of Experiments Setup** window appears.



It has tabs for:

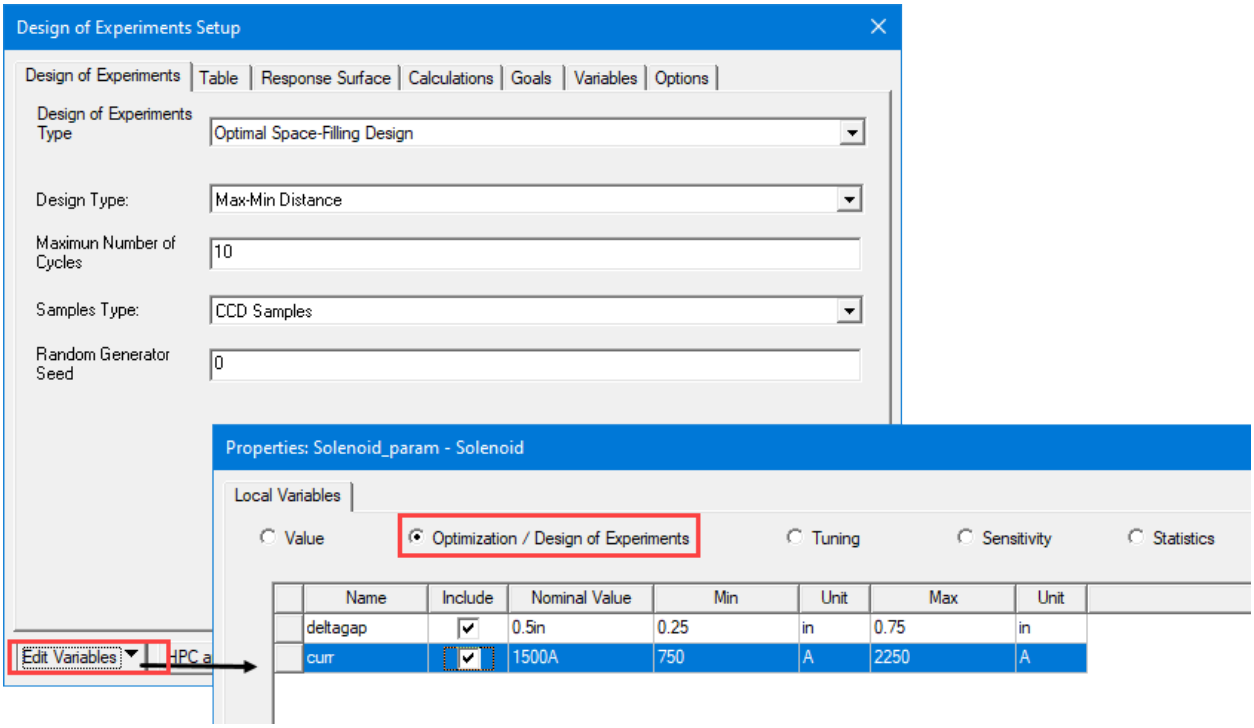
- Design of Experiments, where you specify the sampling points and associated parameters.
- Table, which shows the sampling points defined by the Design of Experiments settings you provide. If you select Custom and the Design of Experiments type, you can import

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

data files as well as add or edit rows. You can also Export files of DOE sampling points you have defined by any method.

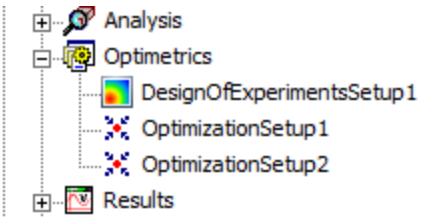
- Response Surface, which specifies the Response surface type, and Refinement parameters.
- Calculations, which accesses Optimetrics calculations.
- Goals, including cost functions calculations and norm type.
- Variables, previously defined for Optimization/Design of Experiments as Project or Design variables, and whether to include them, treat as discrete, whether to Use Manufacturable Variables, and the available Levels.
- Options, to Save Fields and Mesh, and whether to copy geometrically equivalent meshes.

You must define local or project variables as Optimization /Design of Experiments variables for the Design of Experiments setup to include the variable.

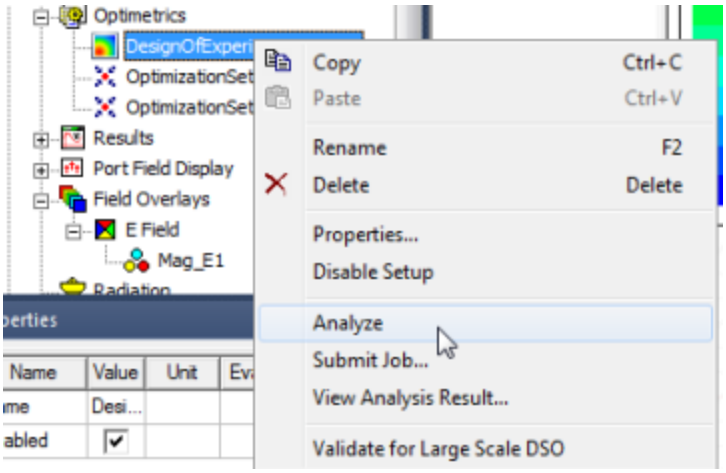


Once you have set parameters and click **OK**, the Design of Experiments setup appears under the Optimetrics icon in the Project tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From here you can right-click the DesignOfExperiments setup from the shortcut menu to run **Analyze**, **Submit Job...** or **Validate for Large Scale DSO**.

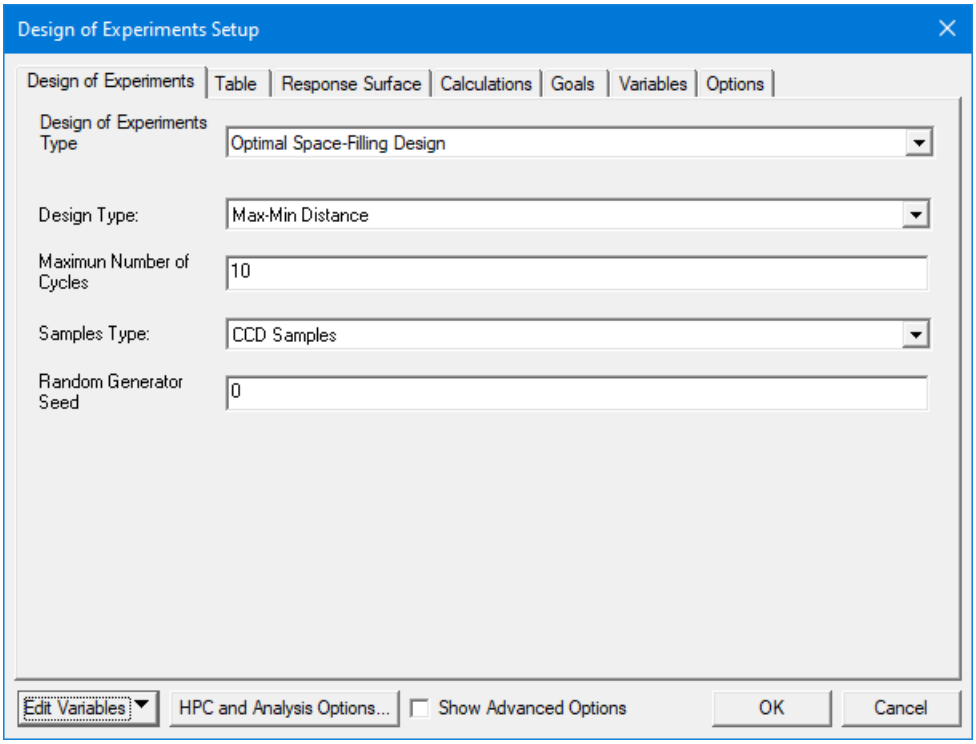


Stand-alone and distributed simulation will have the same behavior as the parametric setup.

Design of Experiments Tab

The Design of Experiments tab in the DOE Setup includes selections for defining the sampling points that define your experiment. Each selection for Design of Experiments type has a different set of associated parameters so the appearance of the dialog changes to show the parameters for your selection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined

The Design of Experiments Types available in the Desktop include the following:

Design of Experiments Types	Brief Description (see links for more details)
Optimal Space Filling (Default)	An optimized Latin Hypercube Sampling maximizing distance between experiments. Several design type criteria are available: • Max-Min Distance • Centered L2 • Maximum Entropy Sampling type determines the number of samples in the design: • CCD samples (Central Composite Designs are five level factorial designs that are suitable for calibrating the quadratic response model) • Linear model samples

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design of Experiments Types	Brief Description (see links for more details)
	• Pure quadratic model samples • Full quadratic model samples • User-Defined samples You also specify a Maximum Number of Cycles and a Random Generator Seed.
Central Composite Design	Several design types available: • Face-Centered • Rotatable • VIF-Optimality • G-Optimality • Auto-defined You can also choose a Standard or Enhanced Template.
Box-Behnken	Avoids critical configurations in the corner of the design space. Maximum number of input parameters is 12.
Custom	Lets you customize a DOE matrix, by editing values, adding or removing samples, and/or importing samples from a CSV file. Selecting Custom enables an Import button and the **Table** tab, as well as buttons to **Add** editable rows or **Delete** selected rows. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.
Latin Hypercube Sampling	Statistical design where no two experiments share input parameters of the same value. The samples Type can be • CCD Samples • Linear Model Samples • Pure Quadratic Model Samples • Full Quadratic Model Samples • User-Defined Samples, for which you also specify the Number of Samples. For each samples type you also specify a Random Generator Seed.

The Table tab provides a preview view of the design points defined by your selections.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimal Space Filling Design (OSF)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Optimal Space-Filling Design (OSF) creates optimal space filling Design of Experiments (DOE) plans according to some specified criteria. Essentially, OSF is a Latin Hypercube Sampling Design (LHS) that is extended with post-processing. It is initialized as an LHS and then optimized several times, remaining a valid LHS (without points sharing rows or columns) while achieving a more uniform space distribution of points (maximizing the distance between points).

To offset the noise associated with physical experimentation, classical DOE types such as CCD focus on parameter settings near the perimeter of the design region. Because computer simulation is not quite as subject to noise, though, the Optimal Space-Filling (OSF) design is able to distribute the design parameters equally throughout the design space with the objective of gaining the maximum insight into the design with the fewest number of points. This advantage makes it appropriate when a more complex meta-modeling technique such as Kriging, Non-Parametric Regression, or Neural Networks is used.

OSF shares some of the same disadvantages as LHS, though to a lesser degree. Possible disadvantages of an OSF design are:

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported.
- Extremes, such as the corners of the design space, are not necessarily covered.
- The selection of too few design points can result in a lower quality of response prediction.

The following properties are available for the OSF DOE type.

- **Design Type**: The following choices are available:
  - **Max-Min Distance** (default): Maximizes the minimum distance between any two points. This strategy ensures that no two points are too close to each other. For a small size of sampling (N), the Max-Min Distance design generally lies on the exterior of the design space and fill in the interior as N becomes larger. Generally, this is the faster algorithm.
  - **Centered L2**: Minimizes the centered L2-discrepancy measure. The discrepancy measure corresponds to the difference between the empirical distribution of the sampling points and the uniform distribution. This means that the centered L2 yields a uniform sampling. This design type is computationally faster than the **Maximum Entropy** type.
  - **Maximum Entropy**: Maximizes the determinant of the covariance matrix of the sampling points to minimize uncertainty in unobserved locations. This option often

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provides better results for highly correlated design spaces. However, its cost increases non-linearly with the number of input parameters and the number of samples to be generated. Thus, it is recommended only for small parametric problems.

- **Maximum Number of Cycles**: Determines the number of optimization loops the algorithm needs, which in turns determines the discrepancy of the DOE. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the DOE smaller. For practical purposes, 10 cycles is generally good for up to 20 variables. The value must be greater than 0. The default is 10.
- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:
  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.
  - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
  - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
  - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different samplings by changing the value or regenerate the same sampling by keeping the same value. The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

Central Composite Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Central Composite Design (CCD) provides a screening set to determine the overall trends of the metamodel to better guide the choice of options in Optimal Space-Filling Design. The CCD DOE type supports a maximum of 20 input parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following properties are available for the CCD DOE type.

- **Design Type**: By specifying the **Design Type** for CCD, you can help to improve the response surface fit for DOE studies. For each CCD type, the alpha value is defined as the location of the sampling point that accounts for all quadratic main effects. The following CCD design types are available:
  - **Face-Centered**: A three-level design with no rotatability. The alpha value equals 1.0. A Template Type setting automatically appears, with Standard and Enhanced options. Choose Enhanced for a possible better fit for the response surfaces.
  - **Rotatable**: A five-level design that includes rotatability. The alpha value is calculated based on the number of input variables and a fraction of the factorial part. A design with rotatability has the same variance of the fitted value regardless of the direction from the center point.
  - **VIF-Optimality**: A five-level design in which the alpha value is calculated by minimizing a measure of non-orthogonality known as the Variance Inflation Factor (VIF). The more highly correlated the input variable with one or more terms in a regression model, the higher the VIF.
  - **G-Optimality**: Minimizes a measure of the expected error in a prediction and minimizes the largest expected variance of prediction over the region of interest.
  - **Auto-Defined**: Design exploration automatically selects the Design Type based on the number of input variables. Use of this option is recommended for most cases as it automatically switches between the G-Optimality if the number of input variables is 5 or VIF-Optimality otherwise.
    However, you can use the Rotatable design if the default option does not provide good values for the Goodness of Fit from the response surface plots. Additionally, you can use the Enhanced template if the default Standard template does not fit the response surfaces well.
- **Template Type**: Enabled for the Rotatable and Face-Centered design types. The following options are available:
  - **Standard**
  - **Enhanced**: Choose this option for a possible better fit for the response surfaces

Box Behnken Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

A **Box-Behnken Design** is a three-level quadratic design that does not contain fractional factorial design. The sample combinations are treated in such a way that they are located at midpoints of edges formed by any two factors. The design is rotatable (or in cases, nearly rotatable).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One advantage of a Box-Behnken design is that it requires fewer design points than a full factorial CCD and generally requires fewer design points than a fractional factorial CCD. Additionally, a Box-Behnken Design avoids extremes, allowing you to work around extreme factor combinations. Consider using the Box-Behnken Design DOE type if your project has parametric extremes (for example, has extreme parameter values in corners that are difficult to build). Because the Box-Behnken DOE doesn't have corners and does not combine parametric extremes, it can reduce the risk of update failures.

Possible disadvantages of a Box-Behnken design are

- Prediction at the corners of the design space is poor and that there are only three levels per parameter.
- A maximum of twelve input parameters is supported.

No additional properties are available for the Box-Behnken Design DOE type.

Custom DOE Type

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

The **Custom DOE** type allows for definition of a custom DOE Table. You can manually add new design points, entering the input and (optionally) output parameter values directly into the table. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.

You can change the edition mode of the DOE table to edit the output parameter values. You can also copy and paste data and import data from a CSV file by right-clicking and selecting Import Design Points.

Latin Hypercube Sampling

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

In the **Latin Hypercube Sampling** Design DOE type, the DOE is generated by the LHS algorithm, an advanced form of the Monte Carlo sampling method that avoids clustering samples. In a Latin Hypercube Sampling, the points are randomly generated in a square grid across the design space, but no two points share the same value. This means that no point shares a row or a column of the grid with any other point.

Possible disadvantages of an LHS design are

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported. For more information, see Number of Input Parameters for DOE Types.
- Extremes, such as the corners of the design space, are not necessarily covered. Additionally, the selection of too few design points can result in a lower quality of response prediction.

> **Note:**
>
> The Optimal Space-Filling Design DOE type is an LHS design that is extended with post-processing.

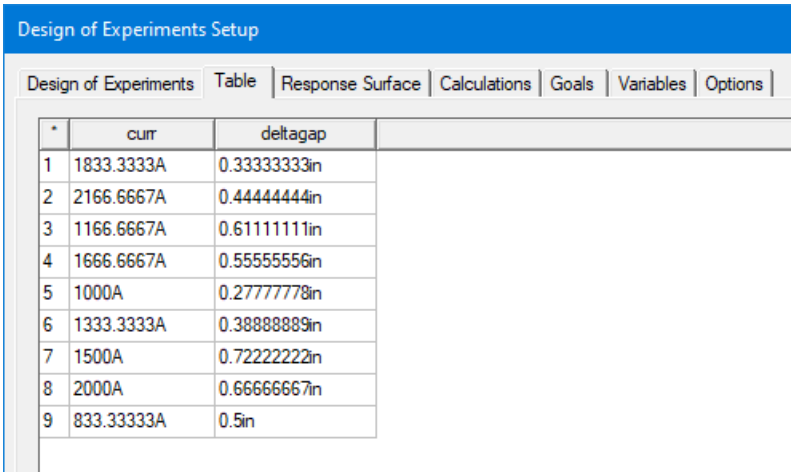
The following properties are available for the LHS DOE type:

- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:
  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.
  - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
  - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
  - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different LHS samplings (by changing the value) or to regenerate the same LHS sampling (by keeping the same value). The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

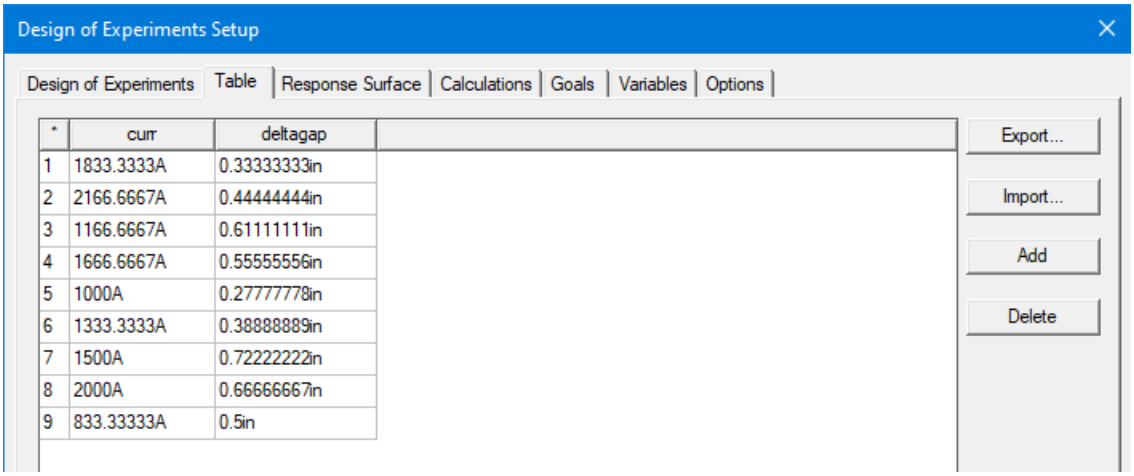
Table Tab for Design of Experiments

The **Table** tab for the **Design of Experiments Setup** window displays a preview of the design points designed by your selections on the **Design of Experiments** tab. There is one column for each defined variable. The **Export** button lets you create a file of the table in a format you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have specified Custom as the Design of Experiments Type, the table is editable, and the **Table** tab includes buttons for **Add** and **Delete** rows. All rows are editable. You can add new rows by entering values in the * row of the table. You enter values in the input parameter columns. Once you have entered a value in one column in the * row, the row is added to the table and the values for the remaining input parameters are set to the initial values of the parameters. You can then edit that row in the table and change any of the other input parameter values if needed. Output parameter values are then calculated when the design is solved updated.



Depending on the context, the tables are read-only and filled automatically, or they are partially or completely editable. The background color of a cell indicates if it is editable or not:

- A gray background indicates a read-only cell
- A white background indicates an editable cell

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output parameter values calculated from a simulation (a design point update) are displayed in black text.

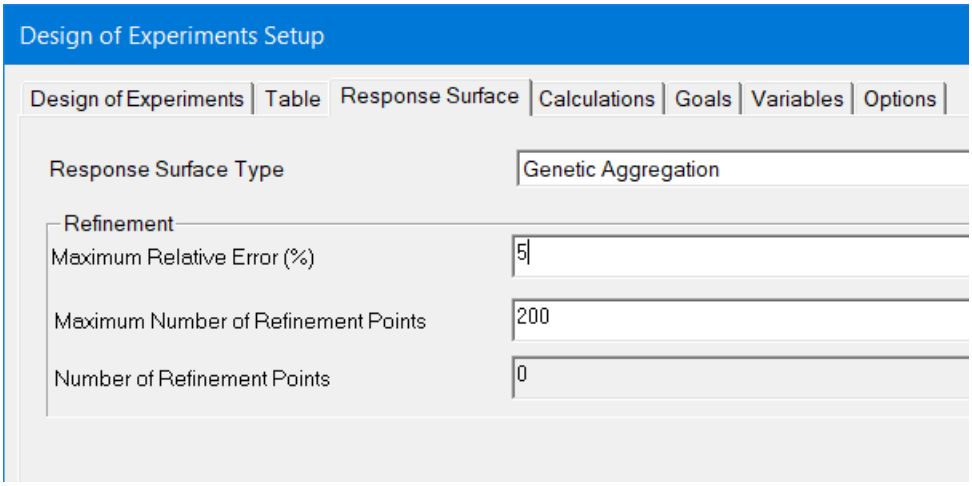
The Custom Table view also includes an Import button. Import and Export files can be

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files, (*.txt)

The Table updates automatically when you change your selections on the **Design of Experiments** tab.

Response Surface Tab for Design of Experiments

The Response Surface tab for the **Design of Experiments Setup** dialog lets you select the Response Surface type as Genetic Aggregation or Standard Response Surface.



The Response Surface type selection specifies the refinement applied to the initial Design of Experiments. The Genetic Aggregation Response Surface finds the best possible response surface for each output automatically by combining

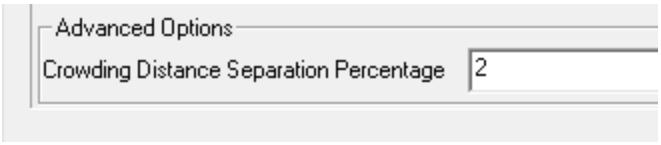
- Metamodels
- Settings
- Kernel Variation
- Polynomial Regression

For each output, a fitness factor works to minimize error, including cross-validation errors. The automatic refinement adds design points to the DOE until the response surface accuracy meets user requirements. You can specify requirements for:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

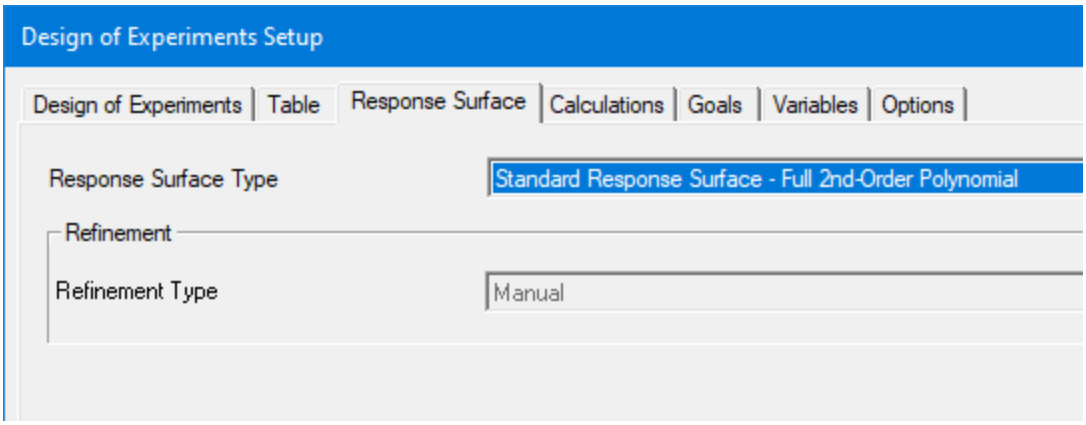
- **Maximum Relative Error %**: This apply to all output calculations. Empty this field to turn off Auto refinement. (Note: Workbench does not have this field.)
- **Maximum Number or Refinement Points**: Determines the maximum number of refinement points that can be generated for use with the Genetic Aggregation algorithm.
- **Number of Refinement Points**: Read-only property indicating the number of existing refinement points.

If you check Show Advanced Options at the bottom of the **Setup** window, you can also specify Crowding Distance Separation Percentage:



Crowding Distance Separation Percentage: determining the minimum allowable distance between new refinement points, implemented as a constraint in the search for refinement points.

You do have the option of selecting Standard Response Surface- Full 2nd Order Polynomial.

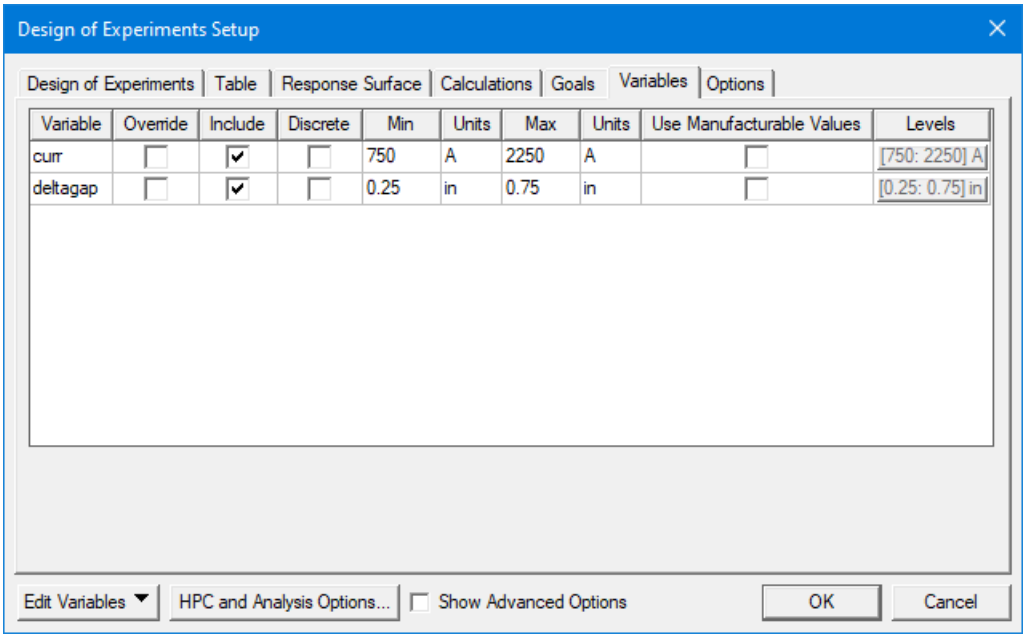


After you have completed an analysis you can view the generated plot.

Variables Tab for Design of Experiments

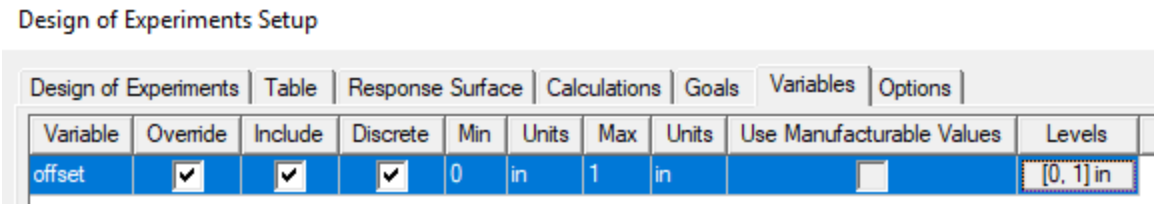
The Variables tab for the **Design of Experiments Setup** window displays list of the variables defined for the Design and Project as Optimetrics/Design of Experiments variables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The columns list the Variable names, the current Min and Max Values and Units, and provides options for the following:

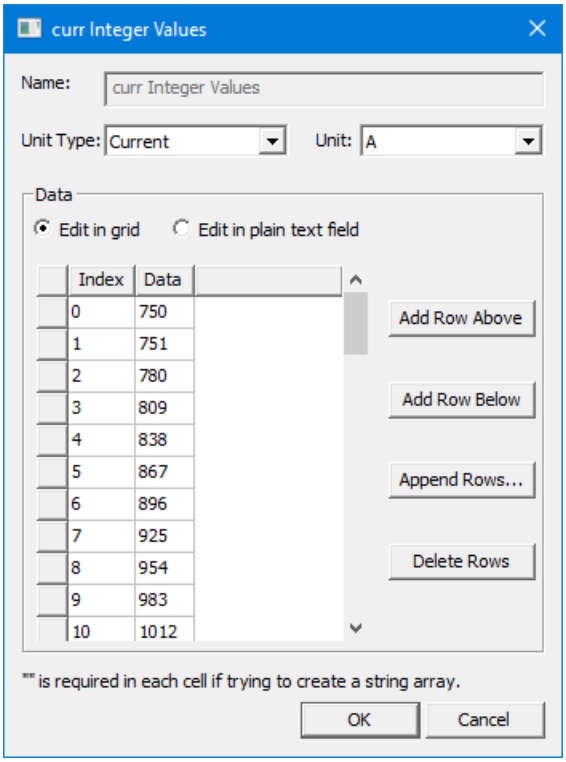
- **Override** – check this to override the current design value. Unchecking this causes a dialog to appear asking you to confirm the return to the design value.
- **Include** – whether to include the variable in an analysis.
- **Discrete** – **Discrete Variables** physically represent different configurations or states of the model. When check box in the Discrete column is checked, the button in the “Levels” column will be enabled and Use Manufacturable Values is disabled.



The Levels show the variable values. For continuous variable, Levels is determined by the min/max columns. For Integer variables Levels can be a subset of the min/max range. The discrete values can be bounded by a min/man range and/or manufacturable values.

Click the enabled **Levels** button for the row to edit the Discrete values. An edit dialog box for the variable appears:

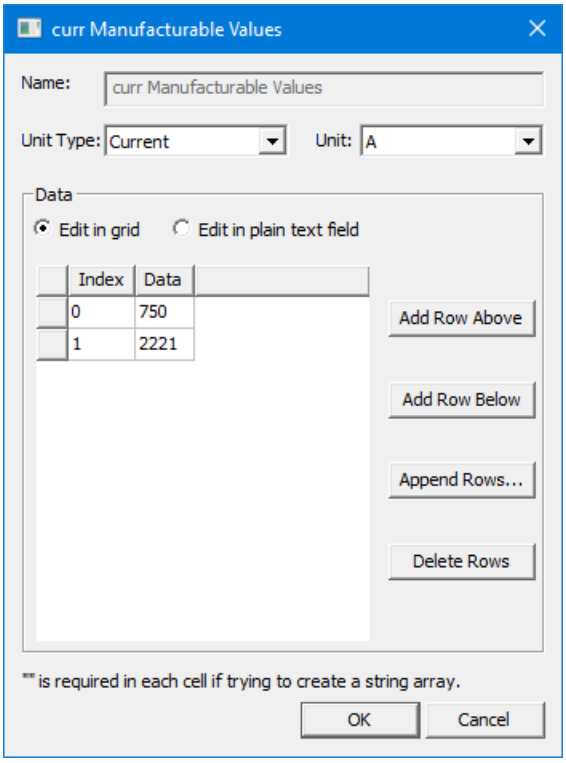
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For continuous variables (with **Discrete** unchecked), the values physically vary in a continuous manner between a lower and an upper bound (min/max) defined by the user. With continuous variables, you can also check **Use Manufacturable Values**.

- When you enable the check box in the "**Use Manufacturable Values**" column the button in the "**Levels**" column is enabled. Manufacturable Values represent real-world manufacturing or production constraints. The min max values for the variable will be upper and lower constraints on the available manufacturable values bounded by the min/max range. If you check **Use Manufacturable values**, you edit the Levels to define the Manufacturable values. Click the Levels button to edit the values. An edit dialog for the variable appears.

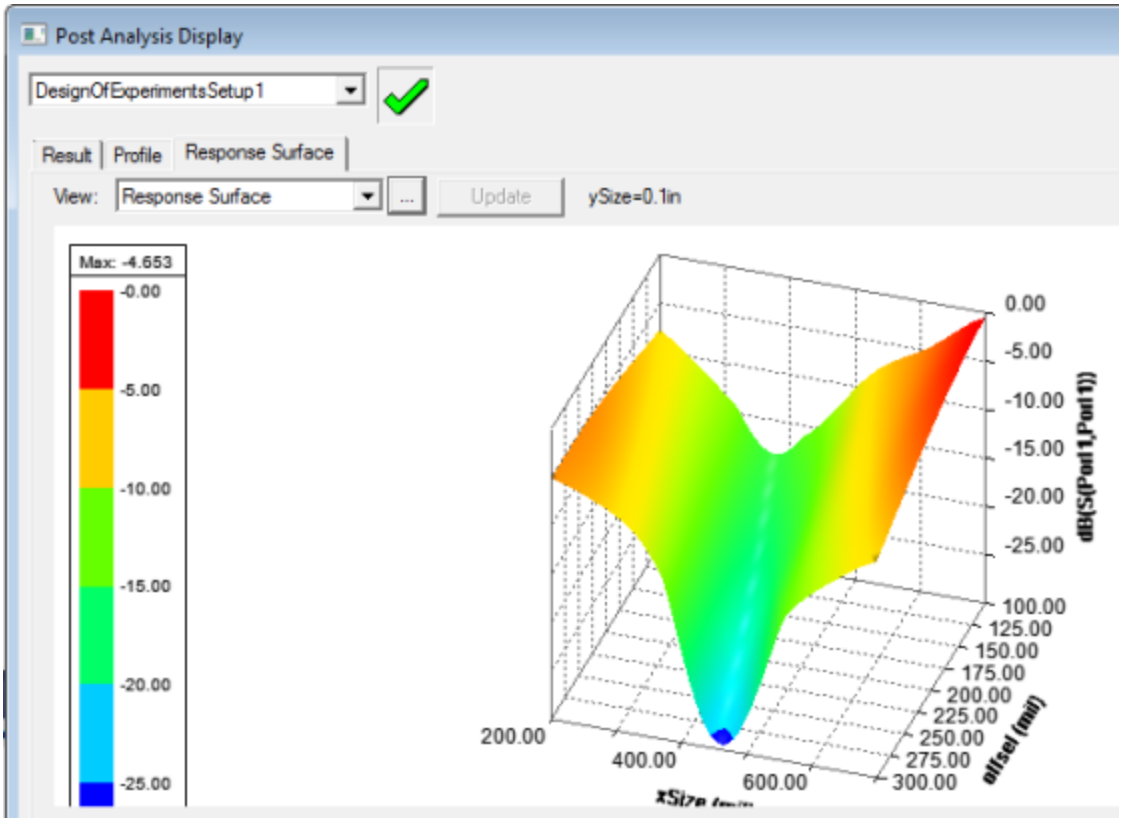
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Analysis Result for Design of Experiments

After the Analysis has competed, you can right-click the Design of Experiments setup in the **Project tree** and click **View Analysis Result** on the shortcut menu. This opens a dialog that includes a **Result** tab that lists the variations and variable values, a solution **Profile** tab with start, stop, time elapsed, and machines used, as well as a **Response Surface** tab. When you include more than one variable in the setup, response surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the ellipsis [...] button next to the view list box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



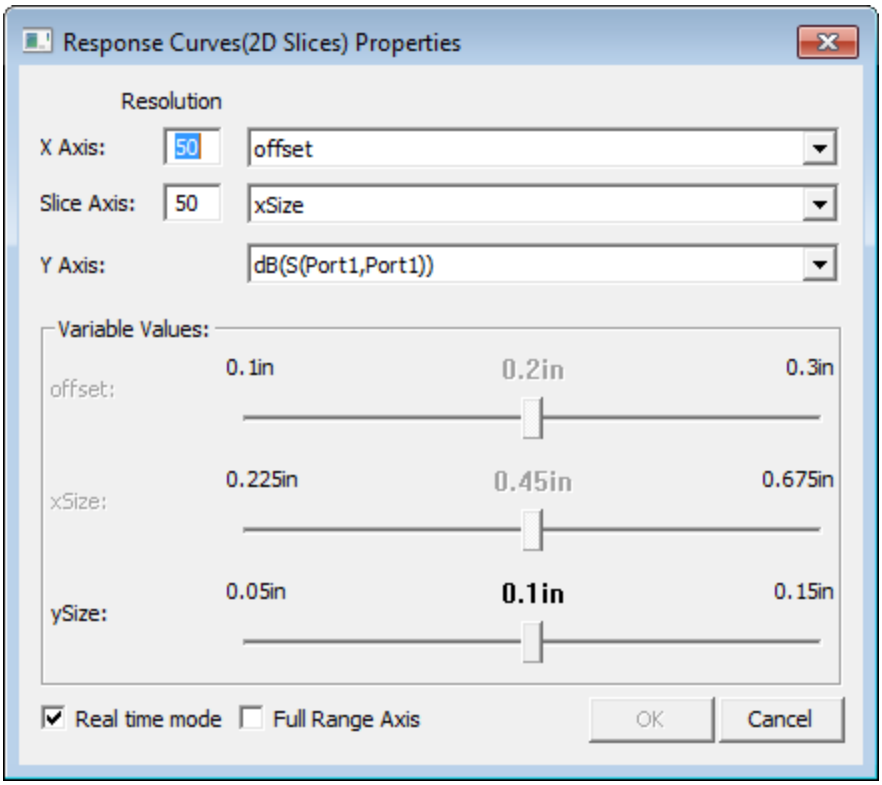
From the **Response Surface** tab, the “View” list box lets you select all available views of the selected response-surface-setup.

- Min Max Search
- Refinement points table
- Response points table
- Verification points table
- Goodness of Fit
- Response Curve
- Response Curve (2D Slices)
- Response Surface
- Local Sensitivity Charts
- Local Sensitivity Curves

The **Update** push button is disabled when the response surface is up to date. After the setup, if you modify a verification point or refinement points, it is enabled. Click this button to regenerate the response-surface with new settings. It may start new simulations if any of the design points in the DOE, refinement points, or verification points has not been solved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the ellipsis button [...] by the View for Response Surface opens a dialog box that lets you adjust the variables selected and the values applied.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

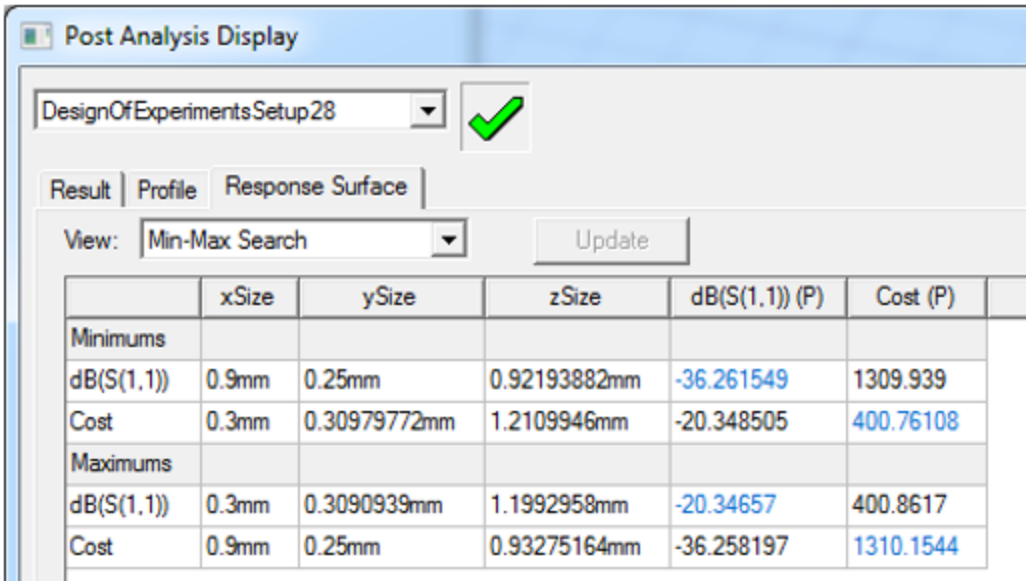
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

Min-Max Search View for Design of Experiments Result

From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the **View** list box lets you select all available views of the selected response-surface-setup.

The **Min-Max Search** view examines the entire output parameter space from a response surface to approximate the minimum and maximum values of each output parameter. When you select a Min/Max row **Apply** is enabled, and you can then apply the selected variation variable values to the variables' nominal values.



	xSize	ySize	zSize	dB(S(1,1)) (P)	Cost (P)
Minimums					
dB(S(1,1))	0.9mm	0.25mm	0.92193882mm	-36.261549	1309.939
Cost	0.3mm	0.30979772mm	1.2109946mm	-20.348505	400.76108
Maximums					
dB(S(1,1))	0.3mm	0.3090939mm	1.1992958mm	-20.34657	400.8617
Cost	0.9mm	0.25mm	0.93275164mm	-36.258197	1310.1544

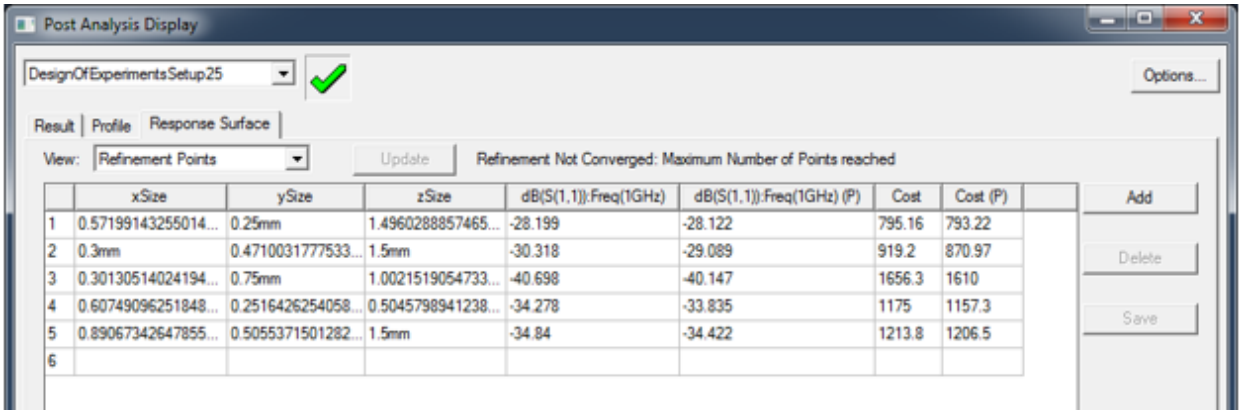
You can Export the Table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Refinement Points Table

From the Response Surface tab, the "View" list box lets you select all available views of the selected response-surface-setup. All refinement points are shown in the Refinement Points table:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Refinement points are points added to your model to enrich and improve your response surface. They can either be generated automatically with the response surface update or added manually, as described in **Performing a Manual Refinement**. As with design points, DesignXplorer must perform a design point update (a "real solve") in order to obtain the output parameters for the refinement points.

Upon update, the refinement points are used to build the response surface and are taken into account for the generation of verification points. Along with DOE points, refinement points are also used as "learning points" for Goodness of Fit calculations.

Performing a Manual Refinement

Manual refinement is a way to force the response surface to take into account points of your choice, in addition to the points already in the Design of Experiments. You can insert a refinement point in the Refinement Points table, and you do not need to solve for an initial response surface (without the refinement point) before updating it with your manual refinement. Manual refinement is available for all response surface types except for Sparse Grid.

You can add, delete, or modify refinement points by clicking the **Add** or **Delete** buttons, or by modifying point values directly in the grid.

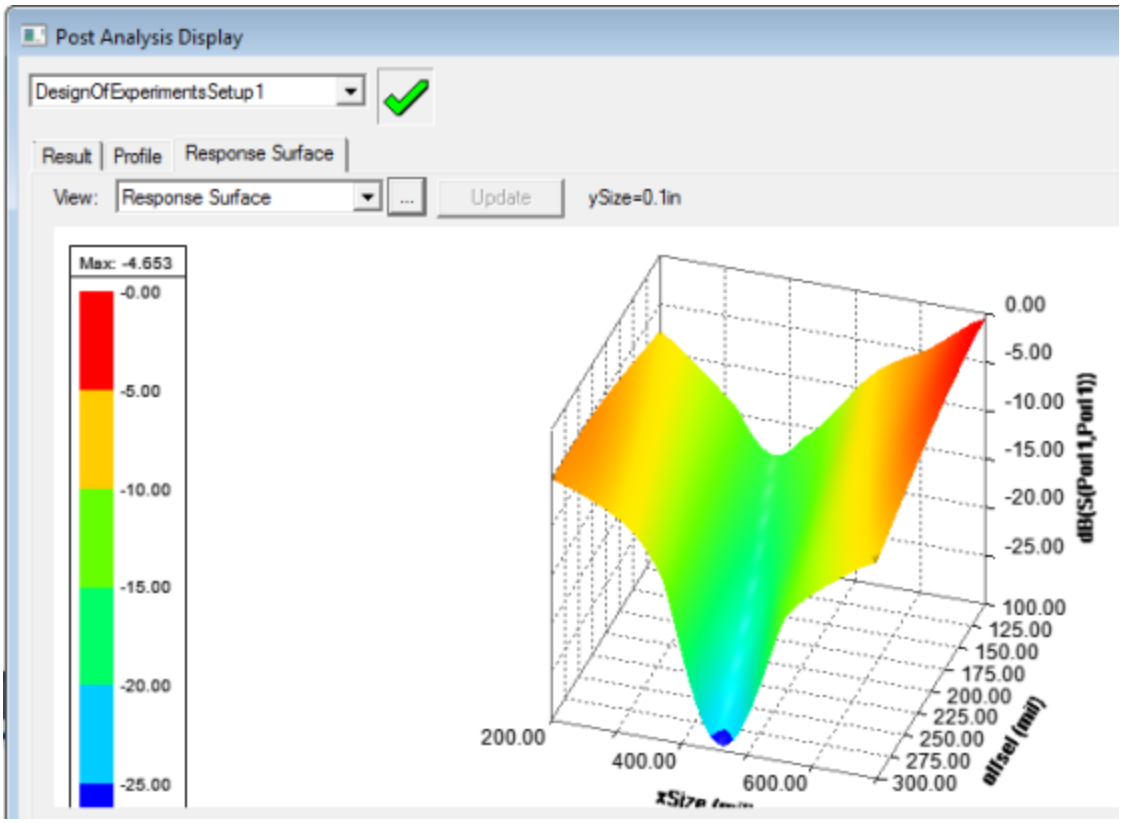
Manual Refinement Point can be inserted from the Response point table and Verification Points Table.

Response Surface Results

When more than one variable is included in the setup, the Response Surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the “...” button next to the view list box. From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the “View” list box lets you select all available views of the selected response-surface-setup.

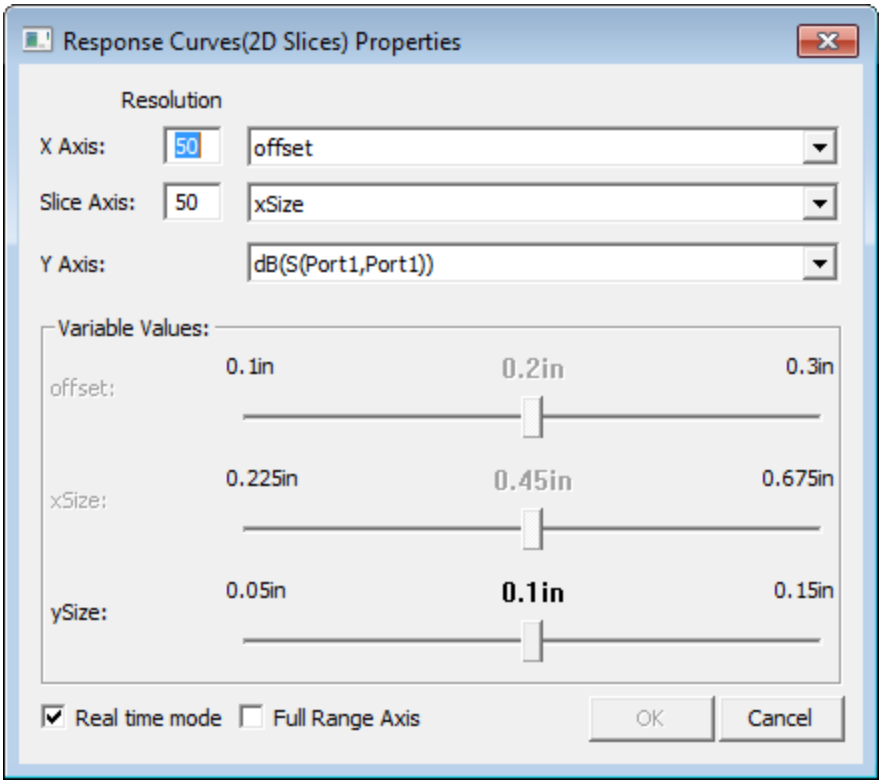
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Response surfaces are functions of varying natures in which the output parameters are described in terms of the input parameters. Built from the Design of Experiments (DOE), they quickly provide the approximated values of the output parameters throughout the design space without having to perform a complete solution. The accuracy of a response surface depends on several factors: the complexity of the variations of the solution, the number of points in the original DOE, and the response surface type. Once a response surface has been generated, you can create and manage response points and charts. These postprocessing tools help you to understand how each output parameter is driven by input parameters and how you can modify your design to improve its performance.



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Surface

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response surface plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked, all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Surface Data

You can Export the Response surface data as a table in the following formats:

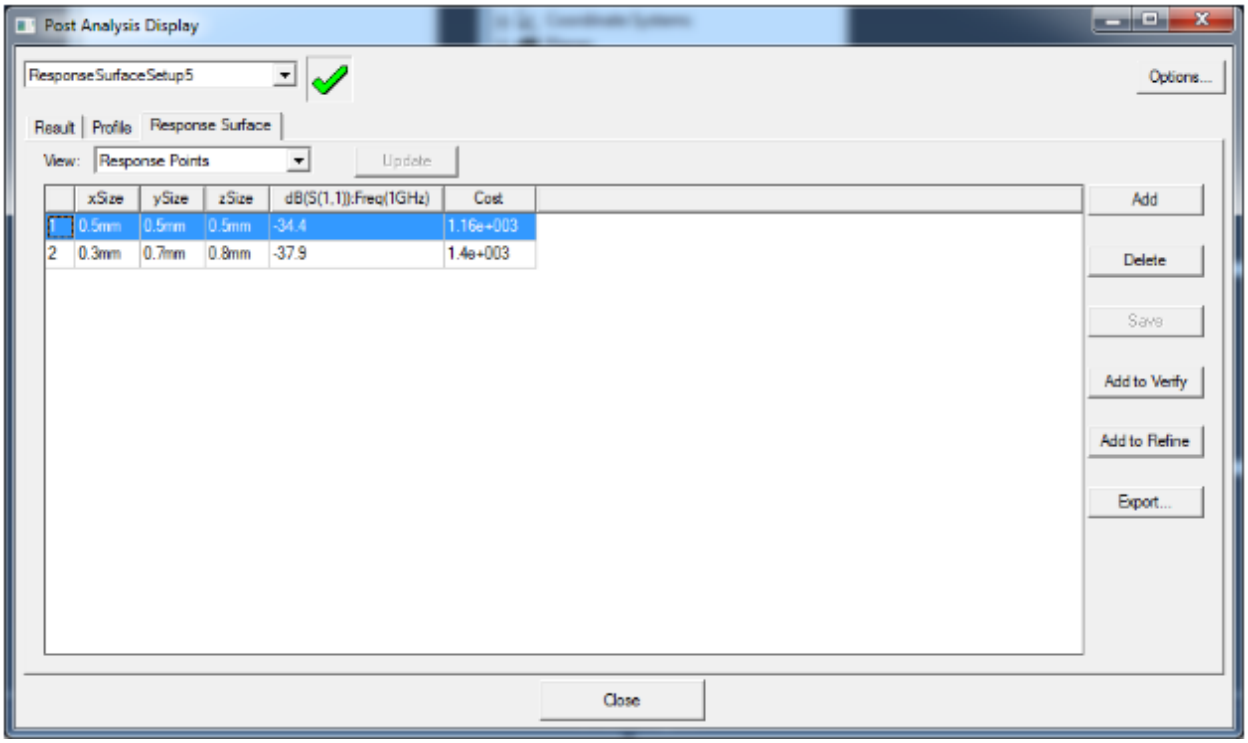
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Exported files can be imported into a report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Response Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Response points are shown in the Response Points table:



A response point is defined by a snapshot of variable values where output calculation values were calculated in Ansys DesignXplorer from a response surface. As such, the output calculation(or cost) values are approximate and calculated from response surfaces.

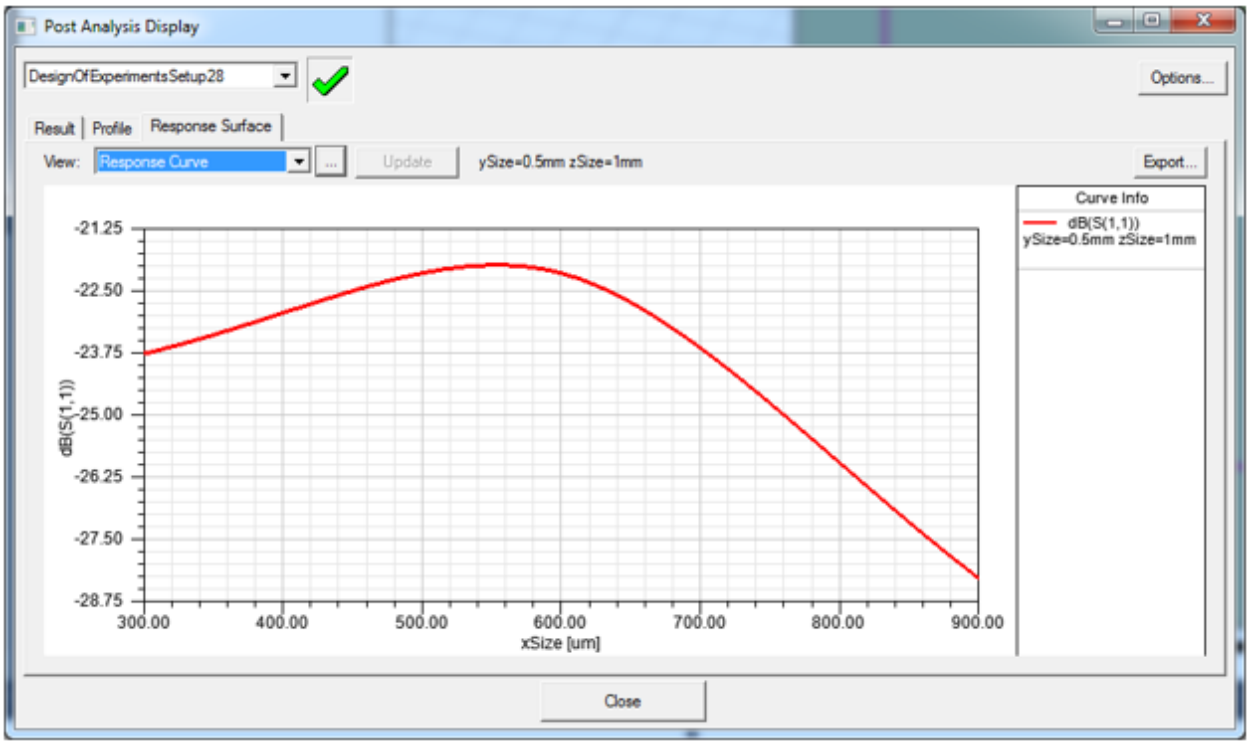
You can add, delete, save, or export response points by using the command buttons in the dialog or you can modify response points manually by modifying point values directly in the grid.

Click **Add to Verify** or **Add to Refine** buttons to insert the selected response point to the verification table or refinement table.

Response Curve

When Continuous Variable is chosen as the X-axis, and Continuous XY plot will be shown.

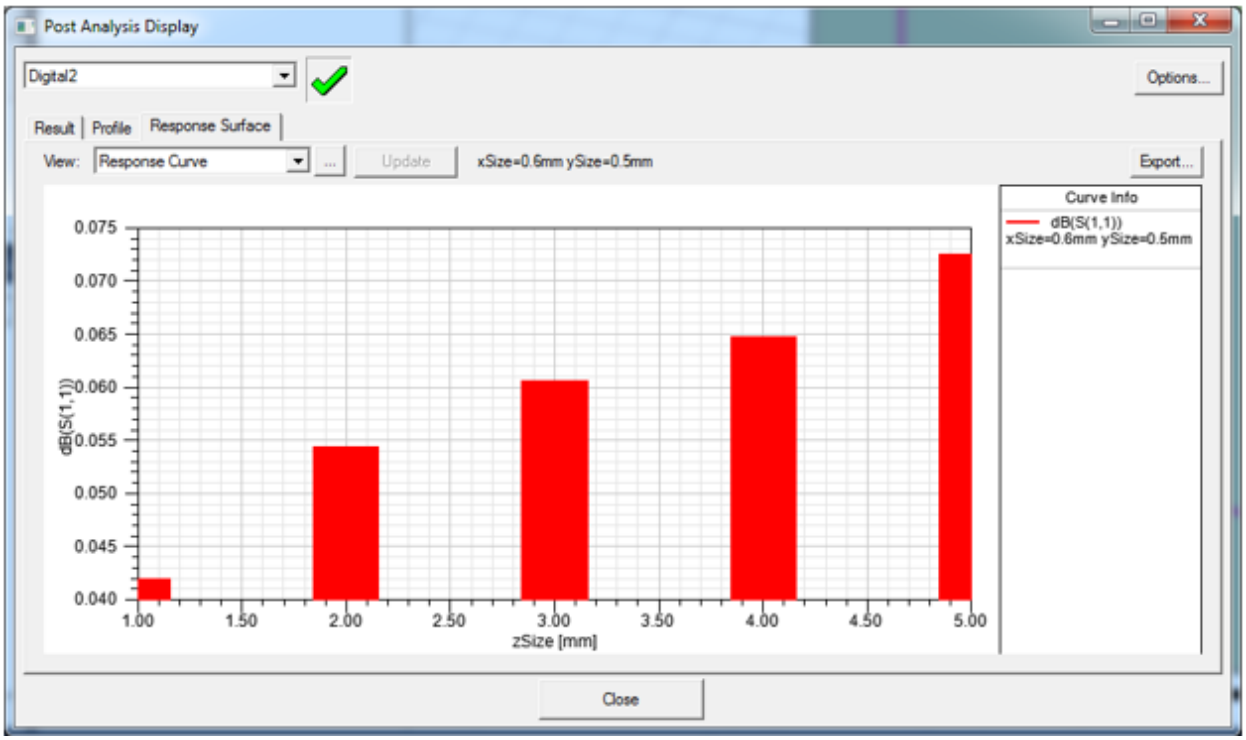
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Discrete Variable

If you choose Discrete Variable as the X-axis, a bar chart plot is shown:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Accumulate Response Curve

It always be unchecked when the tuning dialog started, and won't restore it last check/uncheck state. When it is checked, it will retaining the existing curves, and add new curve to the plot. Uncheck the check box won't clear the accumulated curves, just stop to accumulate new curve. When the Axis variable is changed, all accumulated curves will be cleared.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

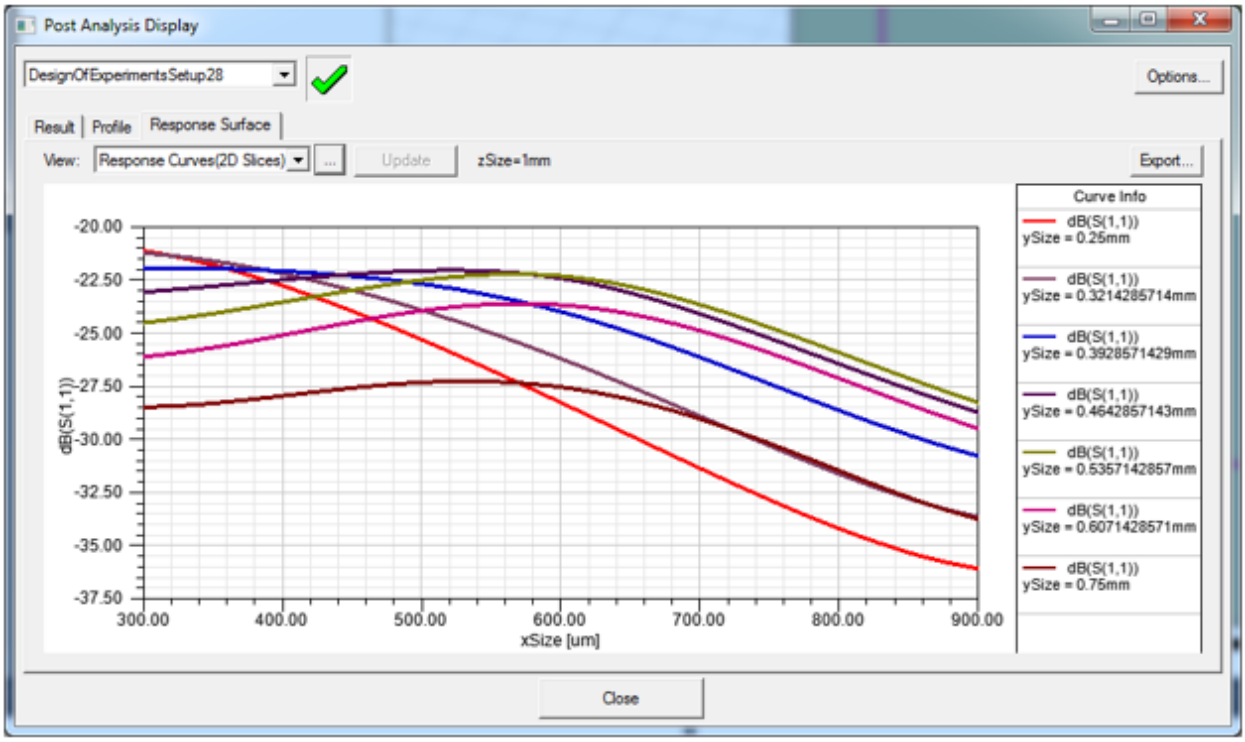
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

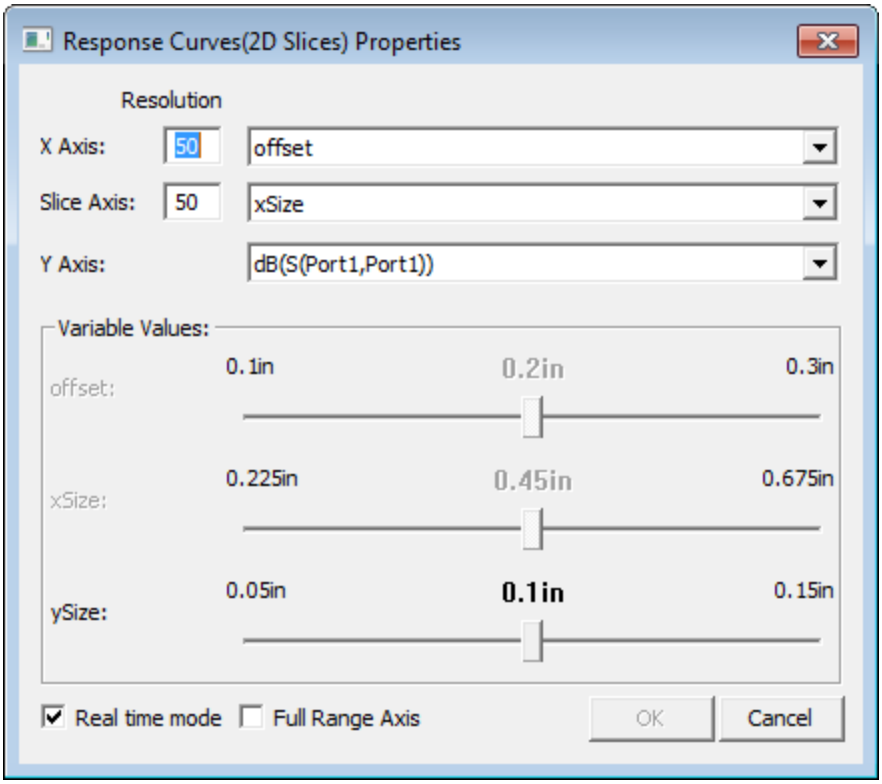
Response Curve (2D Slices)

If you have included more than one variable in the setup, the Response Curves (2D) view is available:



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

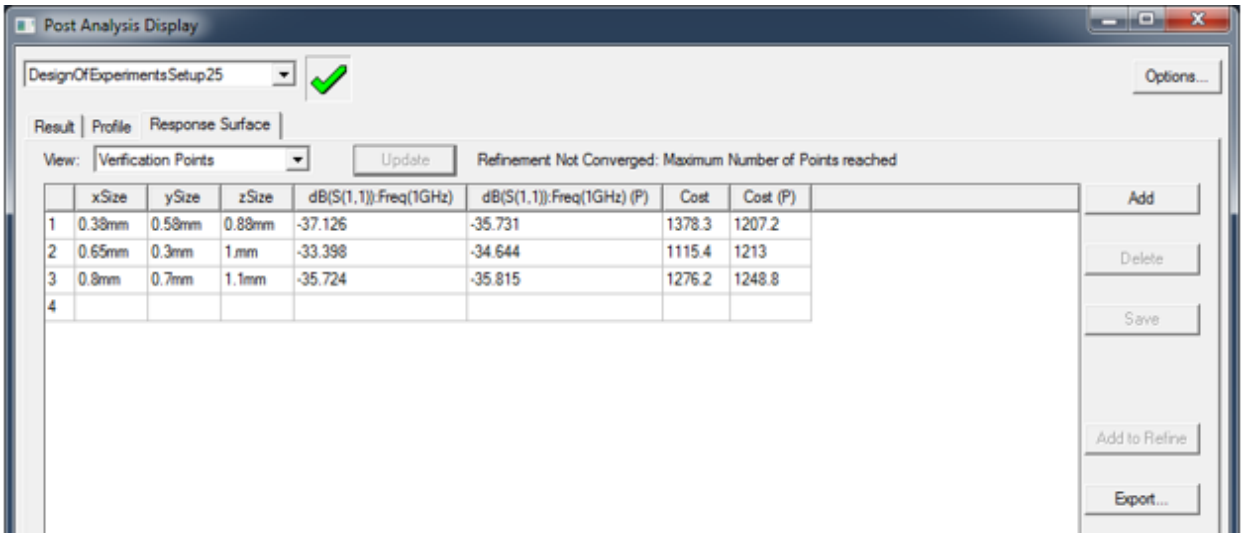
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exported files can be imported into a report.

Verification Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Verification points are shown in the Verification Points table:



Verification points enable you to verify that the response surface accurately approximates the output parameter values; they compare the predicted and observed values of the output parameters.

You can add, delete and modify Verification points manually:

- Same as add/delete/modify refinement points.
- Insert from Response Points Table.

Click **Add to Refine** button to insert the selected response point to the refinement table.

A design point update (that is, a "real solve") calculates each verification point. These verification point results are then compared with the response surface predictions and the difference is calculated.

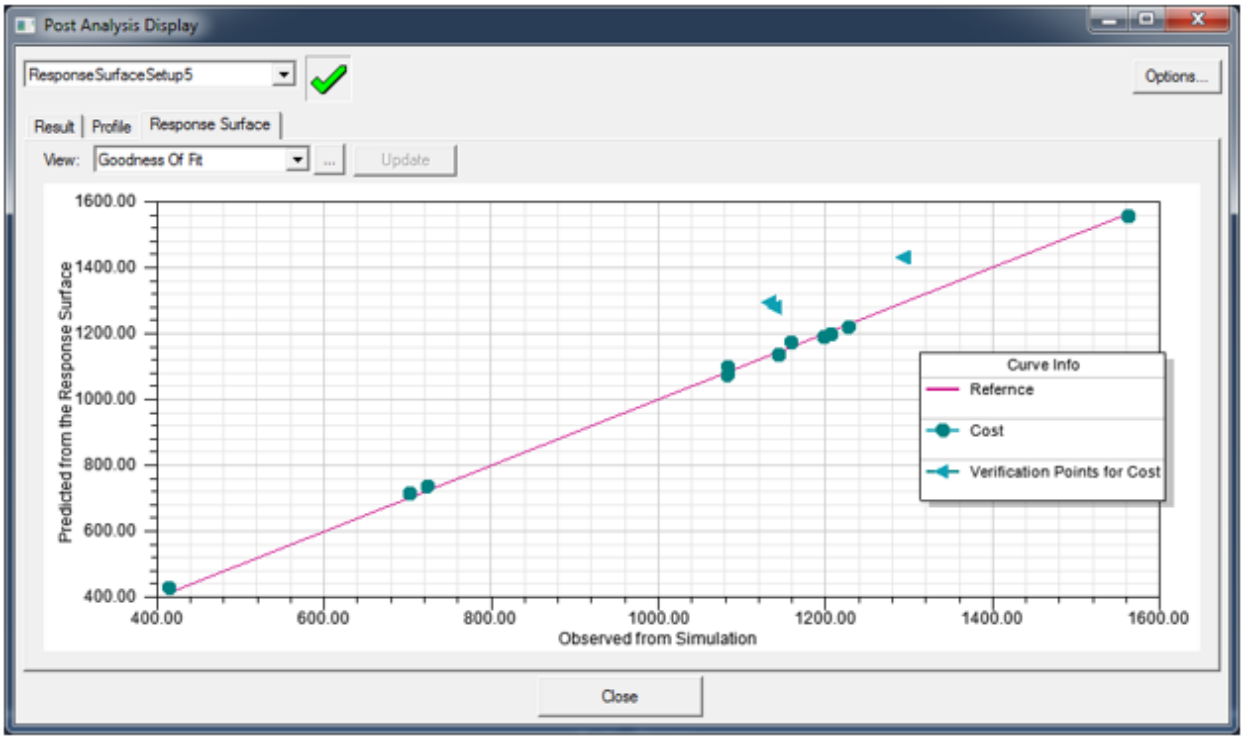
Verification points are useful in validating any type of response surface. In particular, however, you should always use verification points to validate the accuracy of interpolated response surfaces, such as Kriging or Sparse Grid.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Goodness of Fit (Predicted vs Observed Chart)

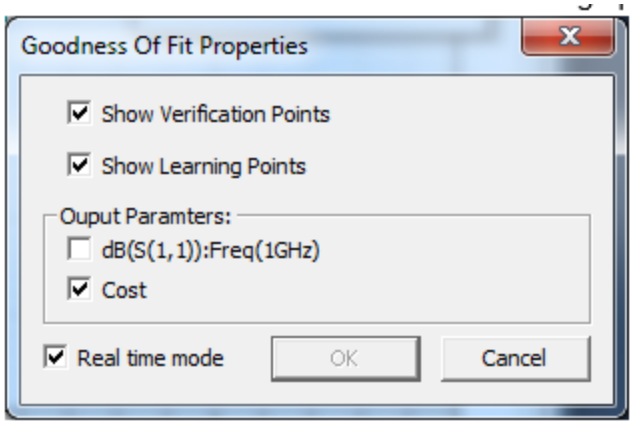
From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup.Response surfaces are built from design points in the Design of Experiments (DOE) and refinement points (collectively, called "learning points"). The Goodness of Fit calculations compare the response surface outputs with the DOE results used to create them.

The closer the points are to the diagonal line, the better the response surface fits the points.



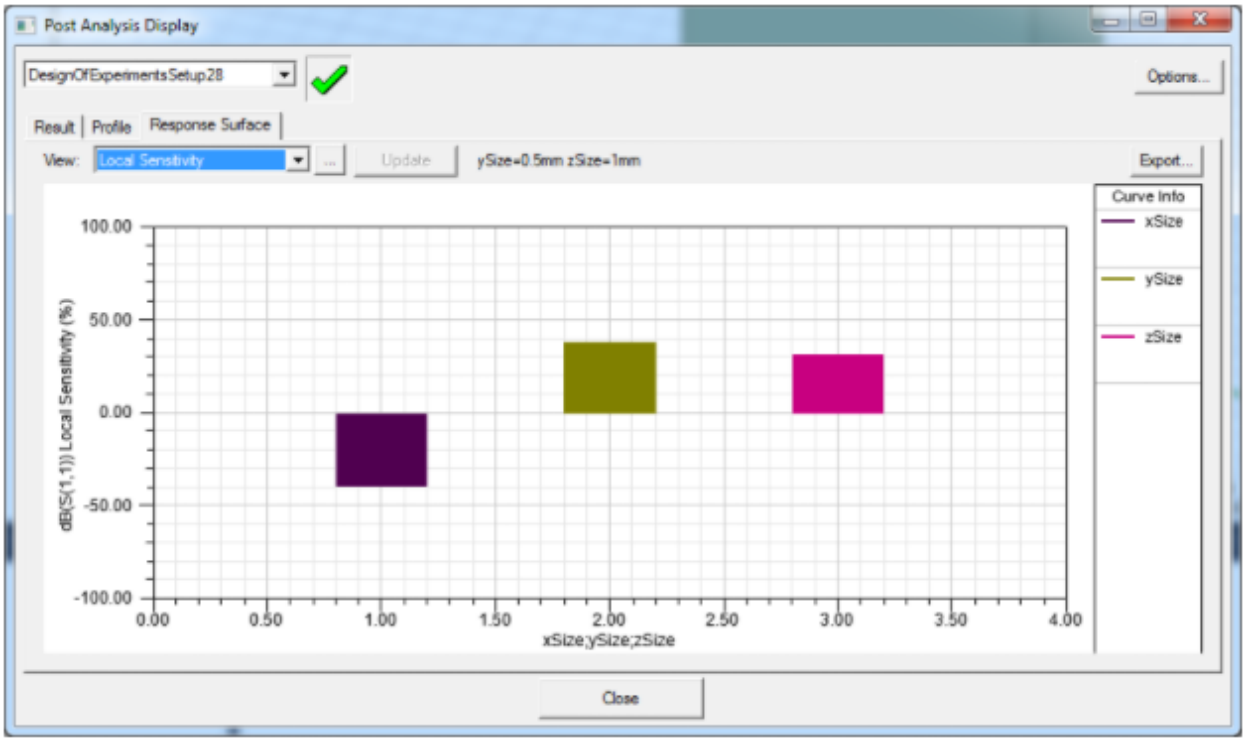
You can view Goodness of Fit information for any of the output parameters in a response surface. To do so click the ellipsis button [...] button to bring up this dialog box:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



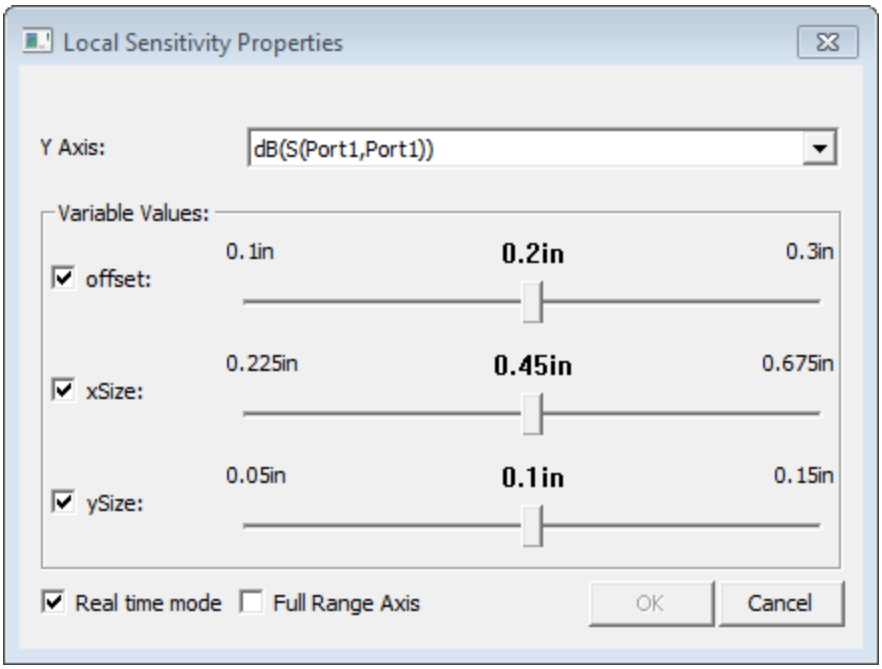
Local Sensitivity Charts

From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Charts allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity chart can be a powerful exploration tool. For each output, it allows you to see the weight of the different input. This chart calculates the change of the output based on the change of each input independently, at the current value of each input parameter in the project.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

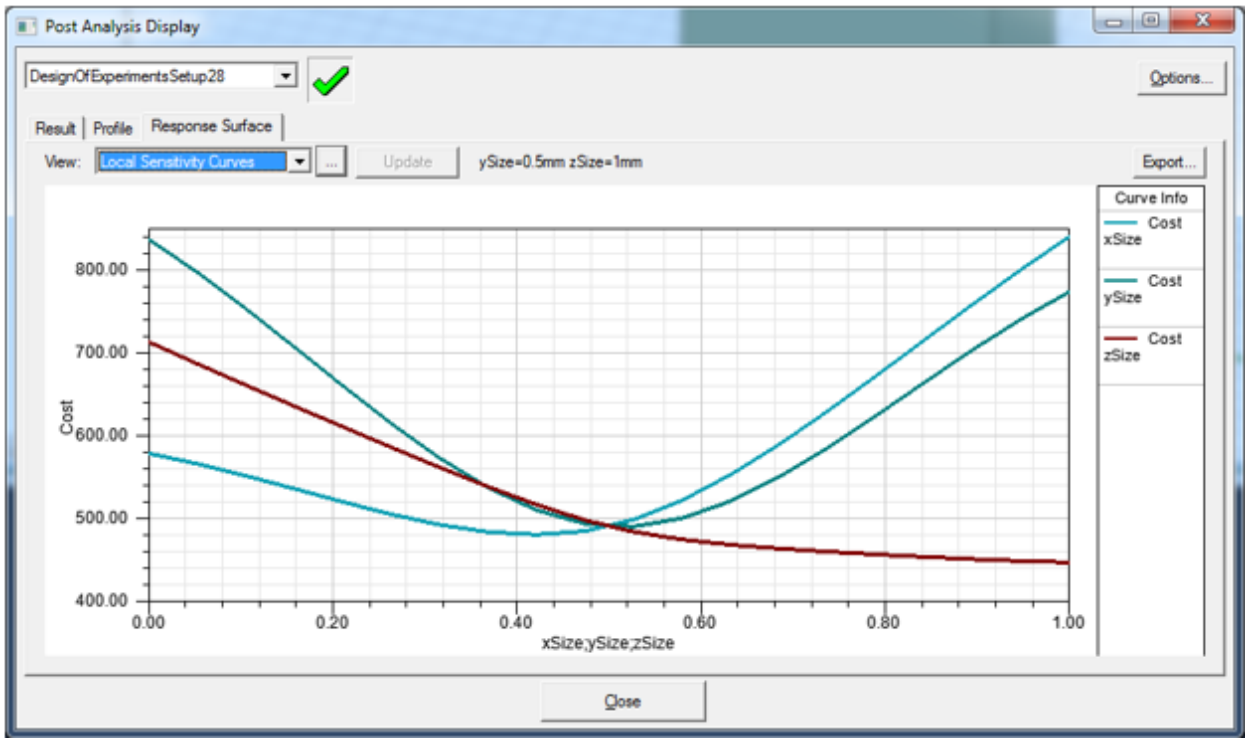
Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog that lets you adjust the variables selected and the values applied.



Local Sensitivity Curves

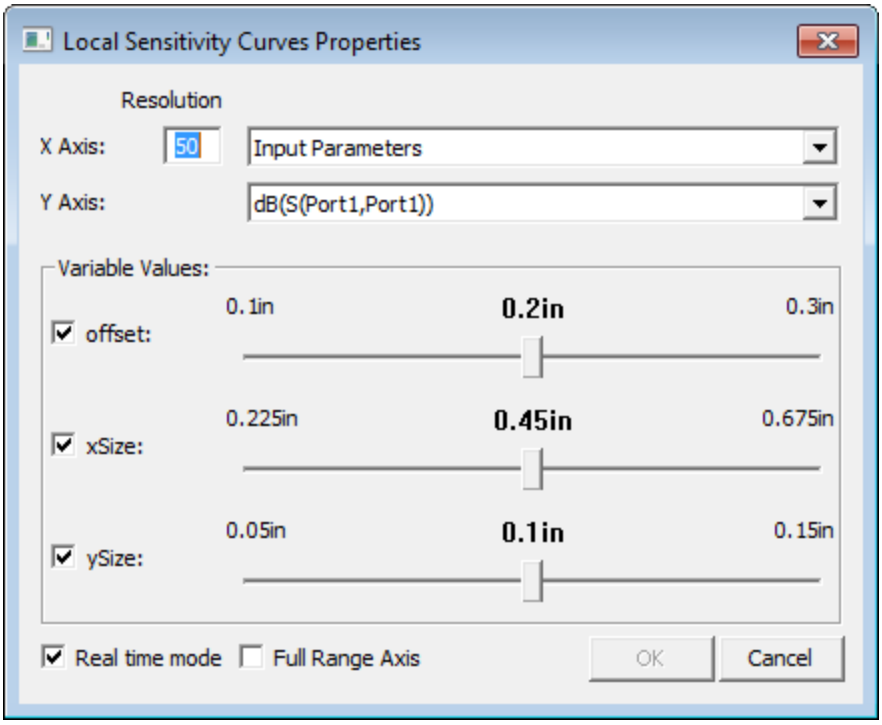
From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Curves allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity Curves chart helps you to focus your analysis by allowing you to view independent parameter variations within the standard Local Sensitivity chart. This multi-curve chart provides a means of viewing the impact of each input parameter on specific outputs, given the current values of the other parameters. The Local Sensitivities Curves chart shows individual local sensitivities, with a separate curve to represent the impact of each input on one or two outputs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog box that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Running an Optimetrics Analysis

Once you have created all necessary Optimetrics based analyses, you have several options for running the simulations.

- To use the **Analyze All** command at the Project or design level to simulate the nominal problem and subsequently run all Optimetrics setups, do the following:
    1. In the Project Manager window, right-click on the **project** or **design** name.
    2. Click **Analyze All** from the shortcut menu.
- To use the **Analyze All** command from the Optimetrics menu to simulate only the Optimetrics based setups, do the following:
    1. In the Project Manager window, right-click on **Optimetrics**.
    2. Click **Analyze** > **All** from the shortcut menu.
- You can choose to analyze only the setups related to a specific Optimetrics type of analysis. In order to simulate setups of a specific type, do the following:
    1. In the Project Manager window, right-click on **Optimetrics**.
    2. Click **Analyze** > **All{*TYPE*}** from the shortcut menu, where *TYPE* is the specific analysis type of interest, Parametric, Optimization, Sensitivity, or Statistical.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

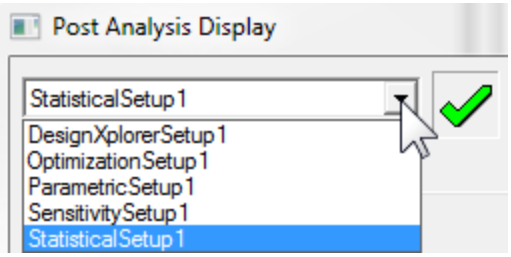
Viewing Analysis Results for Optimetrics Solutions

To view data specific to an Optimetrics solution:

1. In the project tree, right-click the Optimetrics setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Use the drop-down menu to select from available setups.



3. Click the **Results** tab to view results in plot or table form.

   When you view results in Table format, you can sort the results based on each column. Click a column's header to sort. Click again to invert the current sort.

4. Click **Options...** to open a dialog box that allows you to specify the Maximum number of significant digits to display when showing the analysis result. The default is 4.
5. Select the **Profile** tab to view start, stop, and elapsed times for each variable, and the analysis machine for each variation. Click a column heading to sort the table by variation number, variable value, start, stop, elapsed time, or machine.

See the help topics in this section for more details about viewing Optimetrics analysis results.

Viewing Solution Data for an Optimetrics Design Variation

To view the or computing resources used for any design variation solved during an optimization analysis, you must first select the design variation in the **Set Design Variation** dialog box. This dialog box is accessible from the **Solutions** window and via the **Results** > **Apply Solved Variation** command.

1. Click the **Mechanical** menu and then select **Results** > **Solution Data**.

   The **Solutions** dialog box appears.

2. Click the browsing dots beside the **Design Variation** box (...).

   The **Set Design Variation** dialog box appears.

3. Clear the **Use nominal design** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click the design variation for which you want to view the solution data, and then click **OK**.

   The solution data is displayed in the table.

Viewing an Optimetrics Solution's Profile Data

At any time during or after the Optimetrics solution process, you can see an overview of the computing resources or profile data that was used by Mechanical as it solved each design variation. Optimetrics writes the variation information to the profile table before the solve. It then updates the entry with end data (end time, elapsed time, etc.) once the solve variation is completed.

1. In the project tree, right-click the Optimetrics solution setup of interest, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Click the **Profile** tab.
3. Select the Optimetrics setup with the results you want to view from the drop-down menu at the top of the dialog box.
4. Optionally, to examine more detailed profile data for a specific design variation, do the following:
   a. Click a design variation in the table.
   b. Click **Solver Profile**.

   The **Solutions** dialog box appears with the profile data for the selected design variation.

   The profile line for the matrix solver is in the following format:

   Solver *123*

   where:

   - *1* is the precision type: M (mixed) or D (double)
   - *2* is the matrix data type: R (real) or C (complex)
   - *3* is the symmetry type: S (symmetric), A (asymmetric), H (hermitian)

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the parametric setup with the results you want to view from the drop-down menu at the top of the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation. The variation column in the table lists the entries in order. Clicking the Vision header inverts the order. Clicking other headers sorts the entries by value, and clicking again inverts the order.

4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.

5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

Plotting Solution Quantity Results vs. a Swept Variable

To plot solution quantity results versus a swept variable's values on a rectangular (x - y) plot:

1. In the project tree, right-click the parametric setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. If it is not already selected, select **Plot** as the view type.
3. Select the variable with the swept values you want to plot on the x-axis from the **X** drop-down menu.
4. Only one sweep variable at a time can be plotted against solution quantity results. Any other variables that were swept during the parametric analysis remain constant.

   Optionally, to modify the constant values of other swept variables, do the following:

   a. Click **Set Other Sweep Variables Value**.

      The **Setup Plot** dialog box appears. All of the other solved variable values are listed.

   b. Click the row with the variable value you want to use as the constant value in the plot, and then click **OK**.

5. Select the solution quantity results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the view window.

6. Right-click in the plot area to get the shortcut menu where you can set modify the plots display properties, print, copy to the clipboard, or export the data to a file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Cost Results for an Optimization Analysis

To view cost values versus completed iterations in data table format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The cost value at each solved design variation is listed in table format.

3. Optionally, click a design variation in the table, and then click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Cost Results for an Optimization Analysis

To view cost values versus completed iterations in rectangular (x-y) plot format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.

   A plot of the cost value at each iteration appears.

Viewing Output Parameter Results for a Sensitivity Analysis

To view actual output parameter values versus design point in data table format:

1. In the project tree, right-click the sensitivity setup for which you want to view the parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The following values are listed in table format:

   - The regression value of the output parameter at the design point is listed in the **Func. Value** column.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The first derivative of the regression is listed in the **1st D** column.
- The second derivative of the regression is listed in the **2nd D** column.

3. Click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Output Parameter Results for a Sensitivity Analysis

To plot output parameter results versus sensitivity variable values on a rectangular (xy) plot:

1. In the project tree, right-click the sensitivity setup for which you want to view the output parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.
3. Select the sensitivity variable with the sweep values you want to plot on the x-axis from the **X** drop-down menu.
4. Select the output parameter results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the **Post Analysis Display** dialog box.

   The plot displays actual output parameter results for each solved design variation. It also displays a parabola that best fits these results. The parabola is a more accurate representation of sensitivity around the design point than any individual solved design variation.

Viewing Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. To view the results in tabular form, select **Table** as the view type.

   The distribution results for the selected solution quantities are listed in table format for each solved design variation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

5. To view the results in graphic format, select **Plot** as the view type.
6. Type the number of bins you want to plot on the x-axis.
7. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

8. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window changes to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. If it is not already selected, select **Plot** as the view type.
4. Type the number of bins you want to plot on the x-axis.
5. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

Saving Field Solutions for Optimetrics Analyses

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when you request that fields be saved for each solved point in a sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

You can change the general options for a particular design type (HFSS, Q3D, and so on) so that any new Optimetrics setup you define will default to saving the fields. The instructions to do so are included on this page.

Alternatively, you can choose to save the fields on an individual setup basis, without enabling the option to save fields by default. See the subtopics listed in the Related Topics section at the bottom of this page for instructions to do so for each type of optimetrics setup.

To save the fields for all design variations, change the default setting for all projects:

1. Using the menu bar, select **Tools** > **Options** > **General Options**, or from the **Desktop** ribbon tab, click **General Options**.

   The **Options** dialog box appears.

2. In the tree on the left side of the dialog box, expand the branch for the product/design type of interest (HFSS, Q3D, Mechanical, and so on).
3. Select **General**.

   The appropriate group of options appears on the right side of the dialog box.

4. Select **Save Optimetrics field solutions**.
5. Repeat steps 2 through 4 for all products in which you want to save field results by default for all new Optimetrics setups.

   The **Save Fields** option will now be selected by default when you create a new Optimetrics setup in each product for which you selected the **Save Optimetrics field solutions** option.

Saving Field Solutions for a Design of Experiments Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a Design of Experiments setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the Design of Experiments setup, by default field solutions will not be available.

To save the fields for all design variations solved during a Design of Experiments analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Design of Experiments** from the shortcut menu to create a new Design of Experiments setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Design of Experiments setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Design of Experiments Setup** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the design of experiments setup.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Parametric Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a parametric setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the parametric setup, by default field solutions will not be available.

To save the fields for all design variations solved during a parametric analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Parametric** from the shortcut menu to create a new parametric setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Parametric setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Sweep Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the parametric setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for an Optimization Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

To save the fields for all design variations solved during an optimization analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Optimization** from the shortcut menu to create a new optimization setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Optimization setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Optimization** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the optimization setup.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving Field Solutions for a Sensitivity Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a sensitivity analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the sensitivity analysis, all field solutions are deleted.

To save the fields for all design variations solved during a sensitivity analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Sensitivity** from the shortcut menu to create a new sensitivity analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Sensitivity analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Sensitivity Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the sensitivity analysis.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Statistical Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every design variation solved in a statistical analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the statistical analysis, all field solutions are deleted.

To save the fields for all design variations solved during a statistical analysis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Statistical** from the shortcut menu to create a new statistical analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Statistical analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Statistical Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the statistical analysis.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Adding an Expression in the Output Variables Window

When you are in the **Output Variables** window (after clicking **Edit Calculation** from one of the setup analysis windows), do the following to specify an expression:

1. Type a name for the expression in the **Name** text box.
2. Do the following in the **Calculation** section of the window to insert a quantity into the expression:
   a. Select the **Report Type** and **Solution** from the drop-down menus.
   b. Select a **Category**, **Quantity**, and **Function** from the lists, and click **Insert Quantity Into Expression**.
   c. If you want to insert a specific predefined function, select one from the **Function** drop-down menu, and click **Insert Function**.
3. You can also type numbers or expression by hand directly into the **Expression** area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying Meshes in Optimetrics Sweeps

An option in the Optimetrics Analysis setups allows you to request Ansys Electronics Desktop to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. For example, with this option selected a sweep on a scan angle would not need to generate meshes for each solution. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

To copy and reuse meshes on geometrically equivalent parametric variations:

1. Define a variable for the kind of Optimetrics sweep you intend to set up.
2. Select Ansys Electronics Desktop and then select the appropriate **Optimetrics** > **Add** command to display the associated **Setup** dialog box.
3. Click the **Options** tab in the setup dialog box.
4. Select **Copy geometrically equivalent meshes**.

   Ansys Electronics Desktop copies the mesh for a particular parametric sweep for reuse on each geometrically equivalent sweep variation.

> **Note:**
>
> This option is available with all Optimetrics setups, and is applied when these analyses generate geometrically equivalent values. However, it is most relevant to parametric sweep, where such equivalences are more likely to occur.

The **Copy geometrically equivalent mesh** option is not recommended for use when the frequency is varying, since meshing is frequency-dependent. You may wish to turn this option off when the first geometrically equivalent variation requires numerous passes after the initial mesh, but the other geometrically equivalent variations require fewer additional passes, so that it is cheaper to start with the initial mesh each time.

Excluding a Variable from an Optimetrics Analysis

To exclude a variable from being optimized or included in a sensitivity or statistical analysis:

1. Do one of the following:
   - In the **Setup Optimization** dialog box, click the **Variables** tab.
   - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
   - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

   All of the independent variables that were selected for the optimization analysis are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Clear the **Include** option for the variable you want to exclude from the analysis.

   The **Override** option is now selected. This indicates that, for this optimization analysis, the variable is not included.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then clear the **Include** option for the variable you want to exclude.

3. Click **OK**.

Linear Constraints

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via Finite Element Analysis and assigned a cost value; therefore, the cost function is defined over the design space

$$\text{cost}(x): R^n \rightarrow R$$

where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the R^n space. This subset is called the feasible domain and is defined via linear constraints.

You may constrain the feasible domain of a design variable by defining linear constraints for the optimization process. The feasible domain is defined as the domain of all design variables that satisfy all upper and lower bounds and constraints. Linear constraints are defined by the following inequalities:

$$\sum_i \alpha_{ij} x_i < c_j \forall j$$

where

- α_{ij} are coefficients.
- c_j is a comparison value for the j^{th} linear constraint.
- x_i is the i^{th} parameter.

Setting a Linear Constraint

A linear constraint defines the linear relationship between variables. Setting linear constraints in Optimetrics is useful for establishing limitations involving linear combinations of variable values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click **Add**.

   The **Edit Linear Constraint** dialog box appears.
4. Click a **Coeff** text box and type a positive or negative coefficient value.
5. Click a condition, **<** (less than) or **>** (greater than), from the drop-down menu.
6. Type the inequality value, which should be a constant value, in the text box to the right of the condition.
7. Click **OK**.

   You return to the **Linear Constraint** dialog box. The left-hand side of the constraint appears in the **LHS** (left-hand side) column. The condition is listed in the **Condition** column, and the inequality value is listed in the **RHS** (right-hand side) column.

Modifying a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to modify, and then click **Edit**.

   The **Edit Linear Constraint** dialog box appears.
4. Optionally, click a **Coeff** text box and type a new coefficient value.
5. Optionally, click a different condition, **<** (less than) or **>** (greater than), in the pull-down list.
6. Optionally, type a different inequality value in the text box to the right of the condition, and then click **OK**.

   You return to the **Linear Constraint** dialog box. The new coefficient value, the condition, and the inequality value appear in the **LHS** (left-hand side), **Condition**, and **RHS** (right-hand side) columns, respectively.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deleting a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to delete, and then click **Delete**.

   The constraint is deleted.

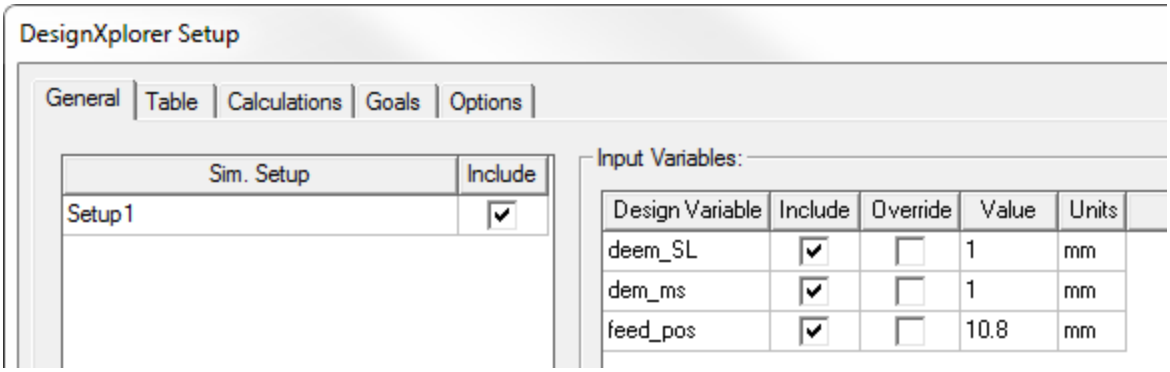
Link to DesignXplorer

You can export a .xml file containing information on a Mechanical setup, optimization variables, and output variables that enables Ansys Design Xplorer to manage the simulations (for example, for design of experiments and optimization). Design Xplorer will launch Ansys Electronics Desktop simulations of design variations and evaluate the outputs.

To do so:

1. Click your product on the menu bar and then **Optimetrics Analysis** > **Add Design Xplorer Setup** or right-click on **Optimetrics** in the **Project Manager** window, and select **Add Design Xplorer Setup** from the short-cut menu.

   This opens the **Design Xplorer** dialog box with the **General** tab selected. it lists the setups available in the current project, and the input variables it contains.



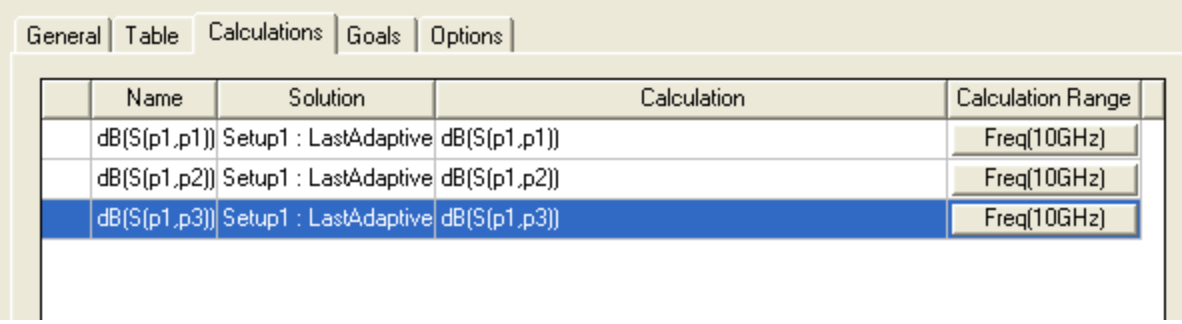
2. Select **Include** for the simulation setups you want to use.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Check the Design variables to use. You can also chose to Override the value of a design variable. You can edit the Value and Units fields. Unchecking **Override** returns the values to their original state.
4. To set up any output calculations, click the **Calculation** tab and click **Setup Calculations**.

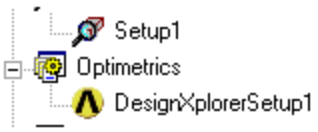
   This opens the **Add/Edit Calculation** dialog box. Here you can define the simulation results of interest. The dialog box contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**. See: Setup Calculations for Optimetrics for details.

   Use the **Add Calculation** button to add expressions to the Calculations table.



5. When you have added the calculations of interest, click **OK** to save the setup.

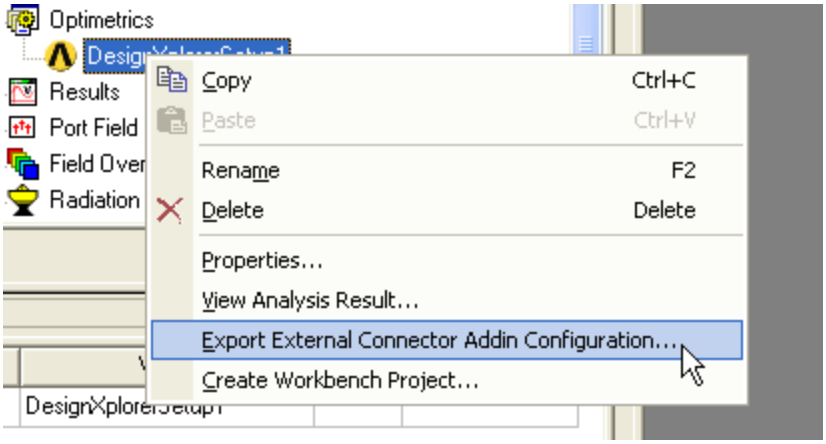
   An icon for the Design Xplorer setup appears under Optimization in the Project tree.



6. To create a .xml file with the setup information for Design Xplorer, first Save your project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Then right-click the setup and select **Export External Connector Addin Configuration**.



   This displays a browser dialog that you can use to navigate your file system and name and saves the .xml file. This file contains information regarding the path along with the setup, variables, and simulation results that you specified.

8. If you have an Ansys Workbench installation, you can perform additional steps. You should have provided a path to the Workbench installation in the **Tools** > **General Options** dialog box Miscellaneous tab, to provide a path.
9. Then click **Create Workbench Project**.

   This lets you name a Workbench project containing the information in the setup. The Ansys Workbench will be launched with the connection to the project established. To this connection, you can add a Design Xplorer Setup. See the documentation of Ansys Workbench for details on Design Xplorer.

Modifying the Value of a Fixed Variable

If you are not including a variable in an optimization, sensitivity, or statistical analysis, Optimetrics uses that variable's current value during the analysis.

To override the current value of a fixed variable for an Optimetrics setup:

1. Do one of the following:
   - In the **Setup Optimization** dialog box, click the **Variables** tab.
   - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
   - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.
2. Click **Set Fixed Variables**.

   The **Setup Fixed Variables** dialog box appears. Under **Fixed Variables**, all of the current independent variable values are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Value** text box of the variable with the value you want to override.
4. Type a new value in the **Value** text box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this Optimetrics setup; the current variable value set for the nominal design is ignored.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new value in the **Value** text box.

5. Optionally, click a new unit system in the **Units** text box.
6. Click **OK**.

To revert to a default variable value, clear the **Override** option.

Tuning a Variable

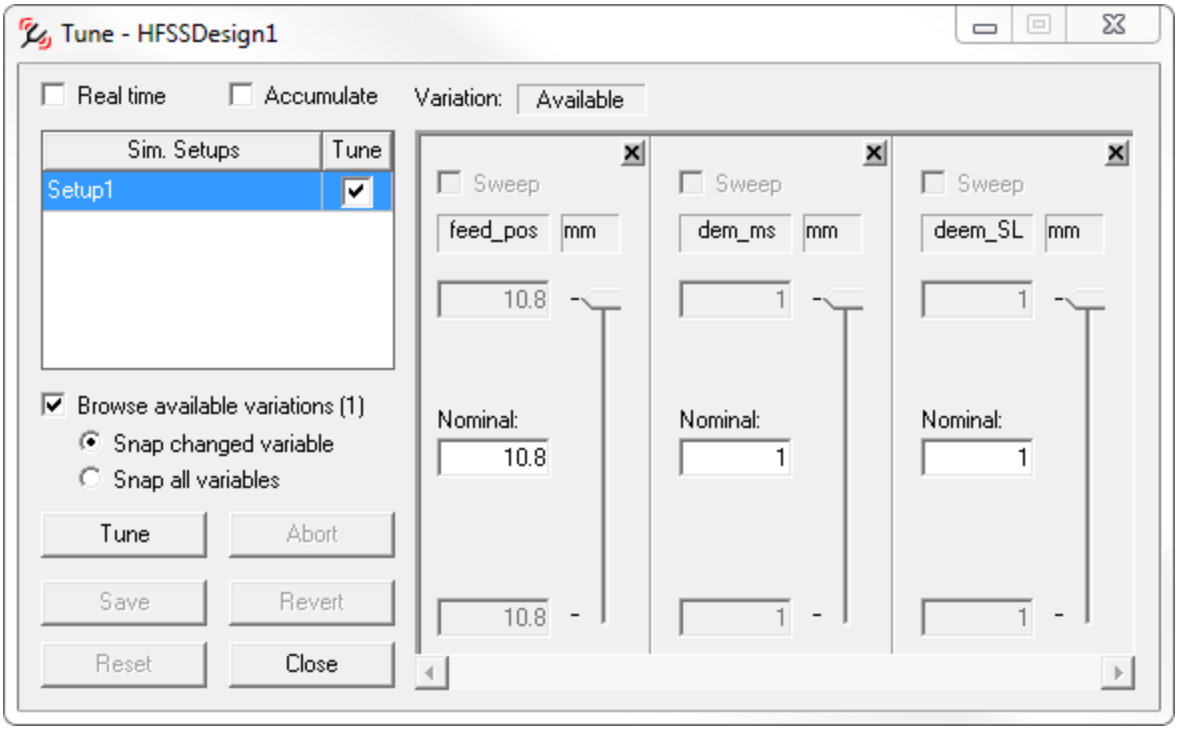
If you want to ensure that tuning does not resolve variations already solved by an optimization setup, you must check **Save Fields and Mesh** in the **Options** tab of that setup.

1. Before a variable can be tuned, you must specify that you intend for it to be used during a tuning analysis in a Project or Design **Properties** dialog box.
2. After running the simulation, click the product in the menu bar and then select **Tune**.

   Alternatively, right-click **Optimetrics** in the Project Manager and choose **Tuning** from the shortcut menu.

   The **Tune** dialog box appears, listing the variables which have been included for tuning.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Clear the **Real Time** option.

   Clearing the **Real Time** option enables the **Tune** button. If this option is selected, a simulation begins immediately after you move the slider. Otherwise, you use the **Tune** button to apply the current values to a simulation.

4. If you want to see updates to an open Report plot while tuning a post processing variable, you must select the **Browse available variations** check box. Selecting **Browse available variations** disables the sweep check box, and the fields for minimum and maximum variable values. This feature lets you see the effect of changes to the post processing variables on plotted results.

   Clearing **Browse available variations** enables the Sweep check box, the minimum and maximum fields, and changes the **Nominal** field to **Step**. (See step 6.)

5. In the **Sim. Setups** column, select the solution setup you want to use when it solves the specified design variation.

   The analysis is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

   Checking the Tune box for a Sim Setup enables the Real Time check box, the Browse available variations check box, and the Snap radio buttons. Clearing the Tune box disables those selections.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the **Nominal** text box for the variable you want to tune, type the value of the variable you want to solve, or drag the slider to increase or decrease its value.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

   Alternatively, if you want to solve a range of values, specify a linear range of values with a constant step size:

   a. Select the **Sweep** check box. (You must have cleared the **Browse available variations** check box).
   b. In the text box below the **Step** value, type the starting value in the variable range.
   c. Type the step size, or difference between variable values in the sweep definition, in the **Step** text box. The step size determines the number of design variations between the start and stop values. The model is solved at each step in the specified range, including the start and stop values.
   d. In the text box just below the variable name, type a stopping value in the variable range.

7. If you have cleared the Real Time check box, click **Tune** to apply the changes you have made to the variable values.

   > **Note:**
   >
   > Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

8. Changing a variable value with the sliders of by typing in the text field enables the **Save** and **Reset** buttons.

   Clicking **Save** opens a **Save As** dialog box with a name field and an **Apply tuned values to design** check box.

   Clicking **Reset** changes the variable values back to what they were originally.

9. If you have changed one or more included variables, clicking **Close** on the **Tuning** dialog box opens the **Apply Tuned Variation** dialog box. This lists the included variables and the values for each tuning. If you have tried multiple values, they are listed, and the current value is highlighted. Select another value to change the highlight. Click **OK** to apply the highlighted values to the design, or **Don't Apply** to ignore the changes from the original variable values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also appear.

Click **Cancel** to close the dialog box and go back to the **Tune** dialog box.

Tuning Overview

Tuning a variable is useful when you want to manually modify its value and immediately perform an analysis of the design. For example, it is useful after performing an optimization analysis, in which Optimetrics has determined an optimal variable value, and you want to fine tune the value to see how the design results (for example, traces in a report) are affected.

A design can be updated after a tuning analysis to reflect a design variation solved during a tuning analysis and the results, including field solutions if you select **Save Fields and Mesh** on the **Options** tab of the associated setup dialog box.

Applying a Tuned State to a Design

You can apply the variable values solved during a tuning analysis to the nominal design in one of the following three ways:

- When closing the **Tune** dialog box
- When saving a tuned state
- When reverting to a tuned state

When closing the **Tune** dialog box:

1. Click **Close** to exit the **Tune** dialog box.

   The **Apply Tuned Variation** dialog box appears.

2. Click the design variation you want to apply, and then click **OK**.

   The variable values from the solved design variation become the current variable values for the nominal design. If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also be apparent.

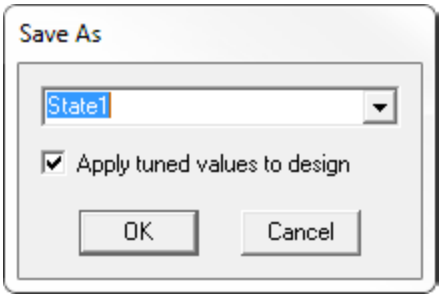
Saving a Tuned State

You can save the settings in the **Tune** dialog box, including the variable values you specified for a tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. After tuning a variable, click **Save** in the **Tune** dialog box.

   A **Save As** dialog box appears.



2. Type a name for the tuned state in the text box.
3. Select **Apply tuned values to design** if you want to update the model to the new variable values.
4. Click **OK** to return to the **Tune** dialog box.

Reverting to a Saved Tuned State

You can revert to a group of saved settings in the **Tune** dialog box, including the variable values you specified for a specific tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. In the **Tune** dialog box, click **Revert**.

   The **Revert** dialog box appears.

2. Type the name of the tuned state you want to apply or click a name in the drop-down menu.
3. Select **Apply tuned values to design** if you want to update the model to the selected tuned state's variable values.
4. Click **OK** to return to the **Tune** dialog box.

Resetting Variable Values after Tuning

If you want to reset variable values to the values they were set to when you started the current session of the **Tune** dialog box:

- After tuning a variable, click **Reset** in the **Tune** dialog box.

  Solutions for the design variations solved during tuning analyses remain available for post processing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16 - High Performance Computing

Ansys Electronics Desktop affords High Performance Computing (HPC) options that allow you to speed up processing. HPC leverages multiple cores through matrix multiprocessing, distributed frequency points (called spectral decomposition method or SDM), domain decomposition (DDM), parallel hybrid FEM/IE solving or the finite antenna array DDM. In addition, hierarchical HPC solving is possible where frequency points can be distributed with each frequency point using multiple cores or machines for large scale DDM analysis at each frequency point, all in parallel. Mechanical intelligently determines which jobs are to be performed and how to distribute them for the simulation. It automatically apportions the jobs during the simulation process and makes optimum use of the available resources. This computing technology enables generating accurate solutions for large, complex, higher-fidelity models.

See the following topics for more information and configuration tips:

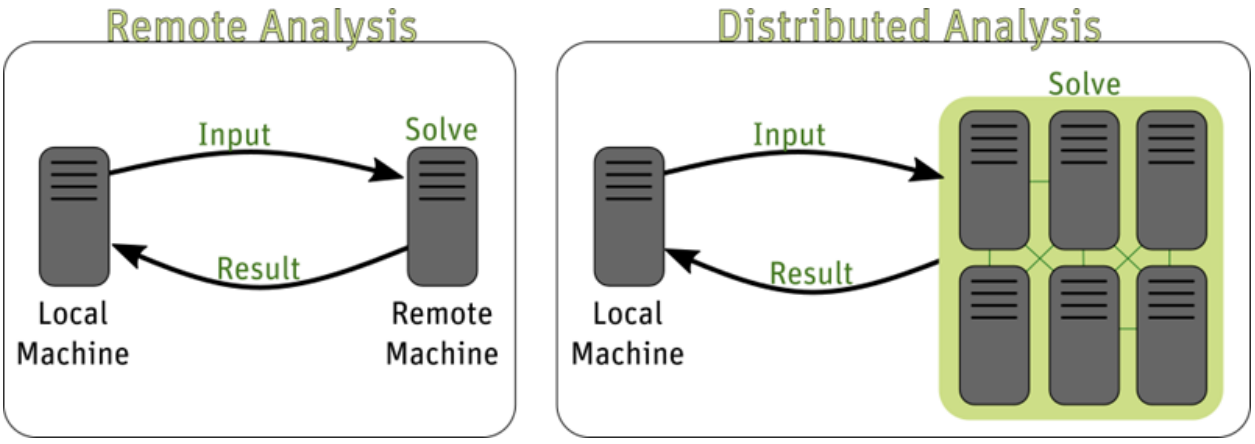
- Remote Analysis
- Distributed Analysis
- Ansys Cloud Direct Support for HPC Job Management
- Large Scale DSO for Parametric Analysis
- Interactive Scheduler Jobs
- HPC Integration
- Multi-Step Job Submission
- Distribution Command Line Options
- GPU Acceleration

Remote Analysis

It is possible to solve a project on a different machine from the one on which you are running Ansys Electronics Desktop. This is particularly useful when you want to take advantage of a more powerful machine but it is not convenient to access that machine. This process involves configuring the machine that will perform the solve (the remote machine), as well as the machine from which the simulation is to be launched (the local machine).

This can also be extended into distributed analysis, where a specified analysis, if supported, is concurrently solved on multiple machines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Important:**
>
> In both Remote and Distributed Analysis, communication between machines can drastically affect performance. Use of a high-speed network system, such as Gigabit or Infiniband, is recommended for optimal performance.

The rest of this topic covers the following:

- Prerequisites for Remote and Distributed Analysis
- Configuring the Local Machine to Solve Remotely
- Remote Analysis Options
- Running Remote Analysis

The **Export Options Files** command writes XML files containing options settings at all levels to the specified directory. **Tools** > **Options** > **Export Options** makes it easier for different users to use Ansys EM tools installed on shared directories or network drives. For some example use cases, see: Example Uses for Export Options Features.

Prerequisites for Remote and Distributed Analysis

1. You must have installed Ansys Remote Simulation Manager (RSM) or a supported High Performance Computing (HPC) management software program (See: High Performance Computing (HPC) Integration).

   For a list of currently supported HPC software, select the topic, *Ansys 2025 R1 - Job Schedulers and Queuing Systems Support* from the Platform Support page.

2. Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

3. If you use RSM, it must be accessible from all remote machines. In addition, the Ansys Electronics Desktop engines must be registered with each initialization of RSM.

   To do this on each machine:

   - From Windows, click **Start** > **Programs** > **Ansys EM Suite 2025 R1** > **Register with RSM**. You can also run **RegisterEnginesWithRSM.exe**, located in the product subdirectory (for example, C:\Program Files\ANSYS Inc\v251\AnsysEM\RegisterEnginesWithRSM.exe).

     In each case, you see a dialog box confirming the registration. Click **OK** to confirm.

   - From Linux, run **RegisterEnginesWithRSM.pl**, located in the product installation directory (for example, /opt/ansys_inc/v251/AnsysEM/RegisterEnginesWithRSM.pl).

     If the RSM service cannot run due to permission issues for the configuration file, it issues an error message and exits.

     If your product is not registered with RSM, the analysis will run locally.

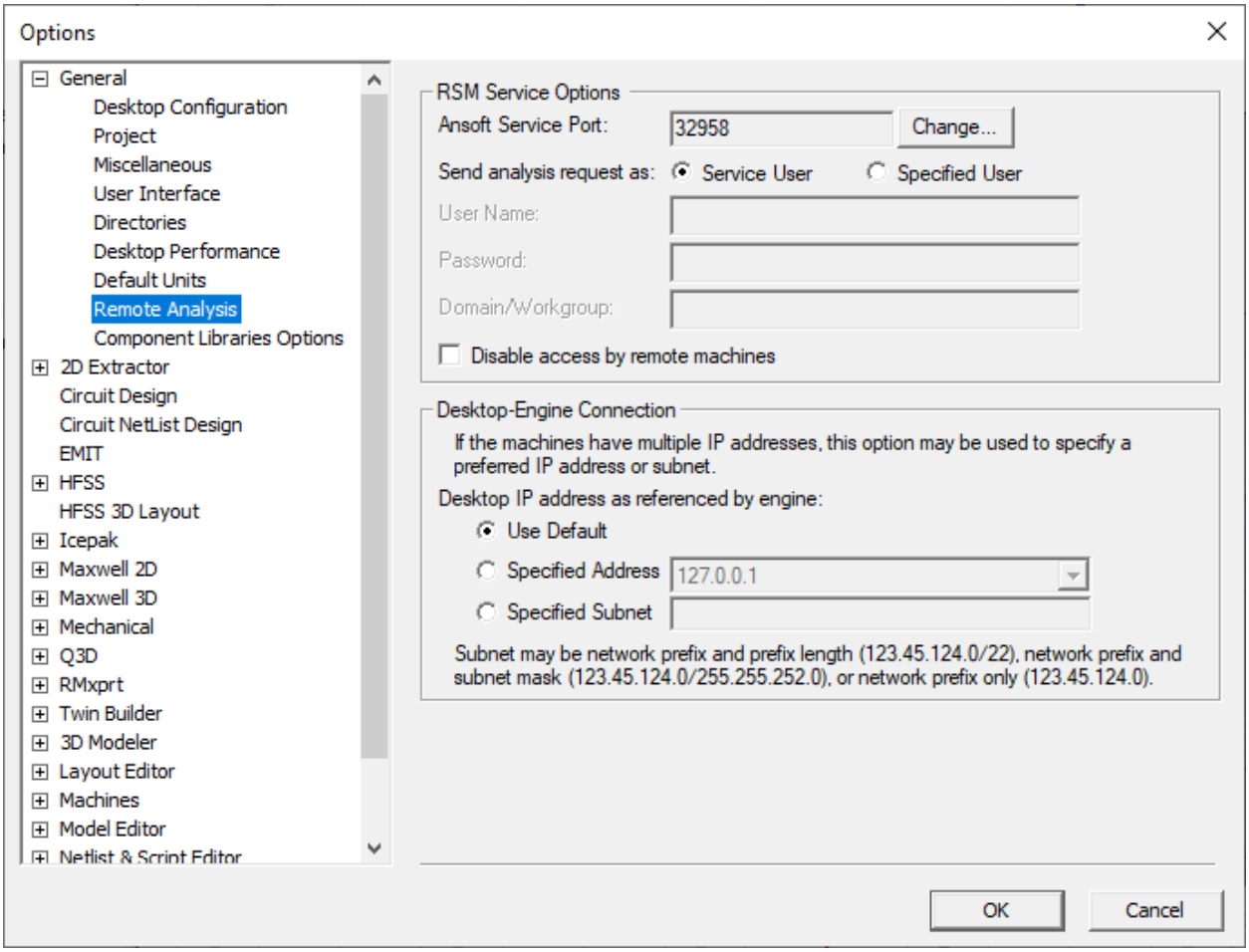
Configuring the Local Machine to Solve Remotely

To set Mechanical Analysis options, see: Distributed Analysis.

Remote Analysis Options

You can also set Remote Analysis Options from **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select whether to run simulation processes as the user running RSM (Service User), or sd a Specified User. If you select Specified user, you must provide the User Name, Password, and any Domain/Workgroup on which this user is defined. If the name or password is incorrect, the Messages window issues a warning, and the solver attempts to perform the analysis as the Service User.

Running Remote Analysis

When you run a simulation remotely, you should see a message in the Progress window identifying the design name and the specified remote machine. You will see Progress messages as the simulation continues. When the simulation is complete, you will see a message in the **Messages** window.

Setting up Linux for Mechanical

When running the Ansys Electronics Desktop on Linux, in order to solve Mechanical designs using a remote Linux machine, you must configure the machines to share login credentials and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enable an automatic login on the remote machine.

Perform the following procedure to configure your Linux machines. The procedure refers uses the following variables:

- machineA = your Linux machine running the Ansys Electronics Desktop
- userA = your username for machine A
- machineB = the remote Linux machine
- userB = the username for machineB

1. Log on to *machineA* as *userA* and create a pair of authentication keys without entering a password.
    - Enter the following command line in the bash prompt: *usera@machineA*:~> ssh-keygen -t rsa
    - When "Enter file in which to save the key (/home/a/.ssh/id_rsa):" appears, press enter without specifying a file name.
    - When "Enter passphrase:" appears, press enter without specifying a password.
    - When "Enter same passphrase again:" appears, press enter without specifying a password.
2. Create a directory on *machineB* as *userB*.
    - Enter the following command line in the bash prompt: *usera@machineA*:~> ssh *userb@machineB* mkdir -p .ssh
    - When "*userb@machineB*'s password:" appears, enter the password and press enter. Entering this password once at this point allows access.
3. Add *usera*'s new public key to *userb* to register the credentials with *machine B*.
    - Enter the following command in the bash prompt: *usera@machineA*:~> cat .ssh/id_rsa.pub | ssh *userb@machineB* 'cat >> .ssh/authorized_keys'
    - When "*userb@machineB*'s password:" appears, enter the password and press enter.

> **Note:**
>
> To verify automatic access to machineB, enter the following in the bash pompt: *usera@machineA*:~> ssh *userb@machineB*. If this is not successful, try the following:
>
> - Place the public key in .ssh/authorized_keys2.
> - Switch the permissions of .ssh to 700.
> - Switch the permissions of .ssh/authorized_keys2 to 640.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troubleshooting

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

Problem:

When you try to solve from local to remote machine, a COM engine process starts on the remote machine, but the user interface hangs indefinitely.

This occurs if the remote solve option is enabled after the COM daemon is started, or when the option "Don't allow exceptions" is selected for the Windows firewall.

Resolution:

Remote solve needs either firewall exceptions to be turned on, or firewall to be completely turned off.

Problem:

When you try to solve from a local to a remote machine, you receive the following error message:

[error] Unable to locate or start COM engine on 'nomachine' : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication. (10:57:13 PM Aug 13, 2019)

Resolution:

This message can happen if the machine is not present, the network connection is down, if there are firewall issues, or if the service is not running.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Problem:

A solve that is distributed to multiple hosts using MPI fails because the AnsysEM installation path is different on different hosts.

Resolution:

Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

Problem:

The command `ansoftrsmservice stop` will not work as root.

Resolution:

Use systemctl command to stop the service:

```
sudo systemctl stop ansoftrsmservice
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Windows

Error:

"Unable to locate or start COM engine on <remote node> : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Make sure the user listed in the service is an administrator.
5. Make sure the COM engine is registered with the RSM Service. From the Windows menu, choose **Start** > **All Programs**> **Ansys EM Suite 2025 R1** > **Register with RSM** to register the engines.
6. If none of these steps fixes the problem, contact Ansys Support.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine from the Windows menu, select **Start** > **All Programs** > **Ansys Electromagnetics Suite 2025 R1** > **Register with RSM**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Linux

Error:

"Unable to locate or start COM engine on <remote node>: Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Check to make sure Remote Simulation Manager is running:
   a. Go to the rsm subdirectory of the Remote Simulation Manager installation directory, <RSM installdir>/rsm.
   b. Type
   `./ansoftrsmservice status`
   c. If the status query indicates that the service is stopped, type ./ansoftrsmservice start.
5. Make sure the COM engine is registered with RSM. Type:
   `./RegisterEnginesWithRSM.pl status`
   from within the Electronics Desktop installation directory. If the status query indicates "Not registered", type:
   `./RegisterEnginesWithRSM.pl add`
6. If none of these steps fixes the problem, contact Ansys Support.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine, go to the Ansys Electromagnetics product installation directory and type:
   `./RegisterEnginesWithRSM.pl add`

Distributed Analysis

Distributed analysis allows users to split certain types of analyses and solve each portion of an analysis simultaneously on multiple machines. Simulation times can be greatly decreased by using this feature.

Mechanical supports the following form(s) of distributed analysis:

- Distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line.

> **Note:**
>
> Communication between machines in remote analysis and distributed analysis can drastically affect performance. Use of a high-speed network system, like Gigabit or Infiniband, is recommended for optimal performance.

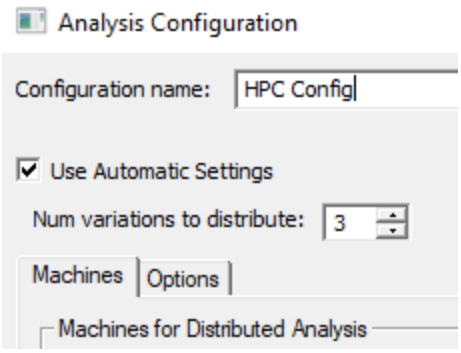
All types of distribution use MPI, except when only distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line. MPI may also be used if Auto mode is specified, and rows of a parametric table are distributed. MPI must be correctly configured if the distributed analysis uses MPI. See Setting HPC and Analysis Options for setting MPI Licensing and the appropriate Ansys Windows or Linux Installation Guide for details on installing MPI.

> **Note:**
>
> MPI is not supported currently for Mechanical designs. Therefore, distributed solutions are limited to parametric analyses, which do not use MPI. This limitation is not to be confused with solving multiple threads of a process on a single multi-core computer. Parallel processing of this type is supported by the Mechanical solvers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beta Feature: Parallel Component Mesh Adapt for 3D Component Array

If you enable the Beta Feature for Parallel Component Mesh Adapt, you can have parallel mesh adaption for 3D Component arrays. This feature works only with Use Automatic Settings enabled on the active Analysis Configuration.



When running a suitable design with this feature enabled, the Solution Profile will show multiple frequencies being adapted at the same time.

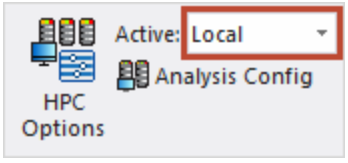
Configure a Distributed Analysis

To configure a distributed analysis, you must select a distributed machine configuration containing a list of machines to use for a simulation, based on memory and CPU considerations (See: Selecting an Optimal Configuration for Distributed Analysis). To create an new distributed machine configuration or to edit an existing one, see Editing Distributed Machine Configurations.

Before you can select a configuration, it must be active. See: Setting HPC and Analysis Options.

To select an existing, active configuration:

- From the **Simulation** tab, use the **Active** drop-down menu to select a configuration.

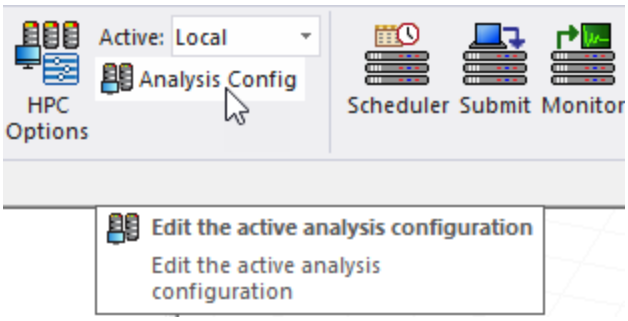


Editing Distributed Machine Configurations for Mechanical

To create a new distributed machine configuration, or edit an existing machine configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

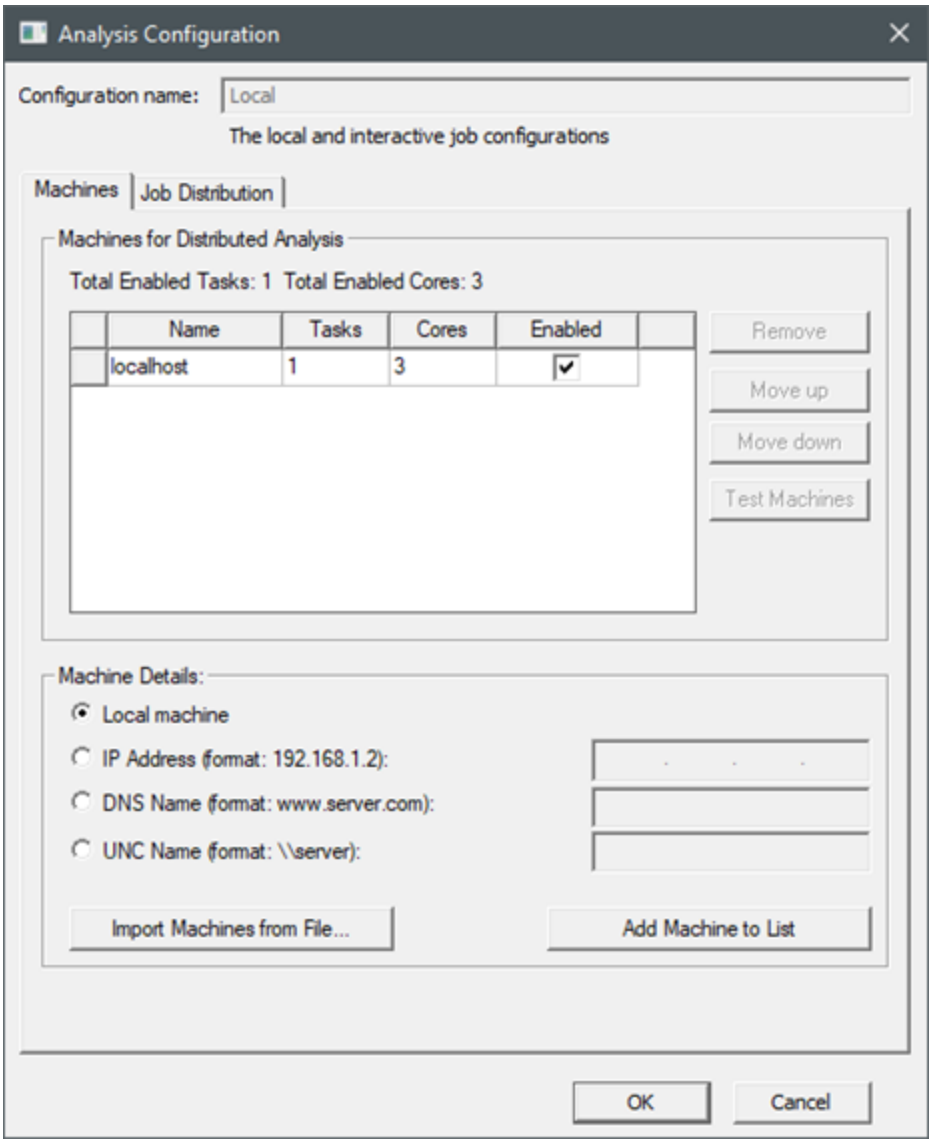
1. Click **Tools** > **Edit Active Analysis Configuration** to open the **Analysis Configuration** dialog directly or click **Analysis Config** on the **Simulation** tab of the ribbon toolbar.



This opens the **Analysis Configuration** dialog box directly. You can also access this dialog from the **HPC and Analysis Options** dialog box by clicking the **Add...**, **Edit...**, or **Copy...** buttons.

If you have selected **Add...** from the **HPC and Analysis Options dialog** box, the fields are empty. If you have selected **Edit...**, used **Tools** > **Edit Active Analysis Configuration**, or clicked **Copy** from the **HPC and Analysis Options** dialog box, the fields show the selected configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Specify the name of the new or edited configuration. It cannot be empty and cannot be a previously used name or a reserved word.
3. Optionally, click **Use Automatic Settings**. The **Job distribution** tab and **Tasks** column are no longer available.

Machines Tab

This tab contains the machine list for the analysis configuration. Here you can provide machine information, either by specifying Remote machine details, or by importing a list of machines from a file. You can then remove, order, test, and enable machines on the list. Control buttons let you **Add machine(s) to list** or **Remove** machines from the list.

Import Machines from File...

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can import a machine list from a file, and an enhanced the file format handles the new flexibility. Each line of the file can contain a machine specifier of the form:

`<MachineName>:<NumTasks>:<NumCores>.`

> **Note:**
>
> This same format is used with the "-machinelist file=" command line option.

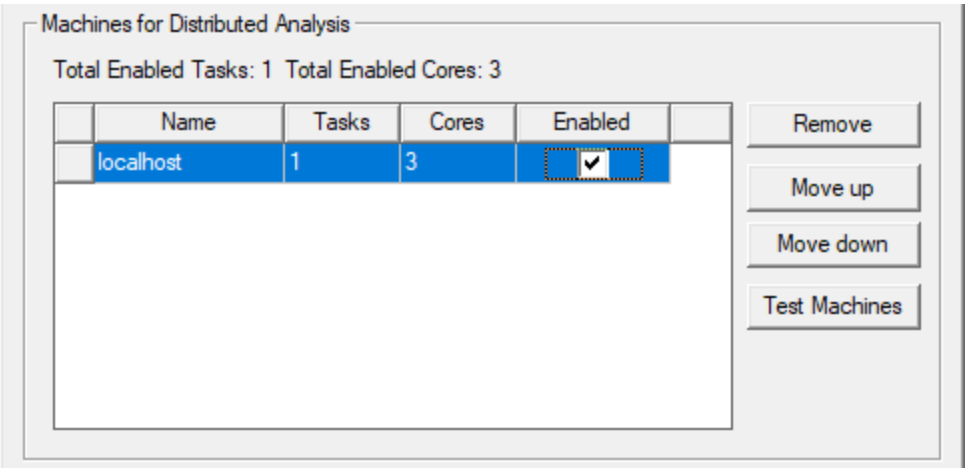
Local Machine Radio Button

To streamline the common case of running jobs on the current machine, use the dedicated radio button to specify the local machine.

4. For each machine to manually add to the list, under *Machine Details*, specify an IP address, a DNS name, or a UNC name. Then, click **Add Machine to List**.

   The remote machines must have the same Ansys Electromagnetics Suite version installed in the same OS and version, and have the RSM service active.

   Once you have specified the remote machine details, either directly or by **Importing Machine from a File**, you select a machine from the table to enable the buttons to **Remove** a machine, or to **Move** a machine up or down on the list.



   The displayed list always shows the order in which you entered them irrespective of the load on the machines. To control the list order, select one or more machines, and use the **Move up** or **Move down** buttons. **Move up** and **Move down** are enabled when you select one or more adjacent machine names.

Enabled Machines

Each machine on the current list has an **Enabled** check box. Here you can enable or disable the listed machines according to circumstance. Above the table, the dialog gives a count of the total enabled tasks, and the total enabled cores.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- For single-level job distributions when Optimetrics is enabled under the *Job Distribution* tab, the number of **Tasks** controls the number of Optimetrics variations to solve in parallel.
- For Mechanical designs, both the solver and the mesher use the **Cores** setting to control the number of parallel processes.
- The number of cores you specify must be equal to or greater than the number of tasks you specify.

For distributed tasks in Mechanical, the software will allocate the cores on a given machine to that machine's tasks. For example, if you want to solve 5 Optimetrics variations, each being allocated 2 cores, you must specify 5 tasks and 10 cores. The software will automatically allocate 2 cores to each of the 5 variations.

Consider a given set of variations (for example, different thermal properties or object dimensions). Ideally, you would make assignments so that each task has the same number of cores. In other words, it is ideal for the number of cores specified to be a multiple of the number of tasks (such as 3 Tasks/6 cores or 4 tasks/12 cores). This is because the solvers attempt to make each task computationally balanced for the greatest solution efficiency. For example, consider two machines, one with eight cores, and another with four. Assuming that the memory is proportionally equivalent, you could assign 2 tasks and 8 cores to machine 1 and assign 1 task and 4 cores to machine 2, giving all tasks the same number of cores (4).

However, balancing the computing resources is a recommendation, not a requirement. You may be limited by the number of available cores on your machines. Similarly, you may wish to limit the number of cores based on the allowable work load on a computer, keeping cores available for other applications to use. As long as the specified number of cores is equal to or greater than the specified number of tasks, the solution will run, whether or not the computing resources are balanced between the tasks.

If you select a distributed configuration (rather than Local) from the Toolbar menu and you launch multiple analyses from the same UI. Mechanical selects the machines that are running the fewest number of engines in the order in which the machines appear in the list. For example, if the list contains 4 machines, and you launch a simulation that requires one machine, Mechanical chooses the first machine in the list. If another simulation is launched while the previous one is running, and this simulation requires two machines, Mechanical chooses machines 2 and 3 from the list. If the first simulation then terminates and we launch another simulation requiring three machines, Mechanical chooses 1, 4, and 2 (in that order).

Test Machines

When multiple users on a network are using distributed solve or remote solve, they should check the status of their machines before launching a simulation to ensure no other Ansys EM

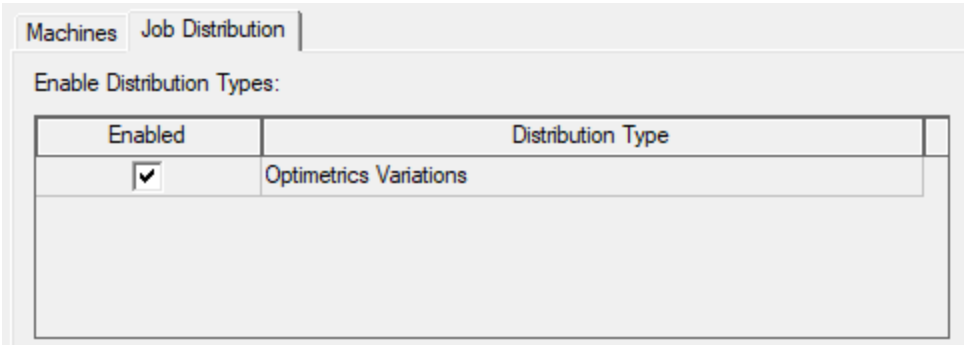
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

processes are running on the machine. To do this, you can select one or more machines and click **Test Machines**. A **Test Machines** dialog box opens.

The test goes through the current machine list and gives a report on the status of each machine. A progress bar shows how far testing has gone. An Abort button lets you cancel a test. When the test is complete, you can OK to close the dialog box. If you need to disable or Enable machines from the list based on the report, you can do so in the **Distributed Analysis Machines** dialog box.

Job Distribution Tab

Use this tab to enable specific types of job distribution and to enable multi-level solves. The Job distribution types listed are design type specific and will differ between products.

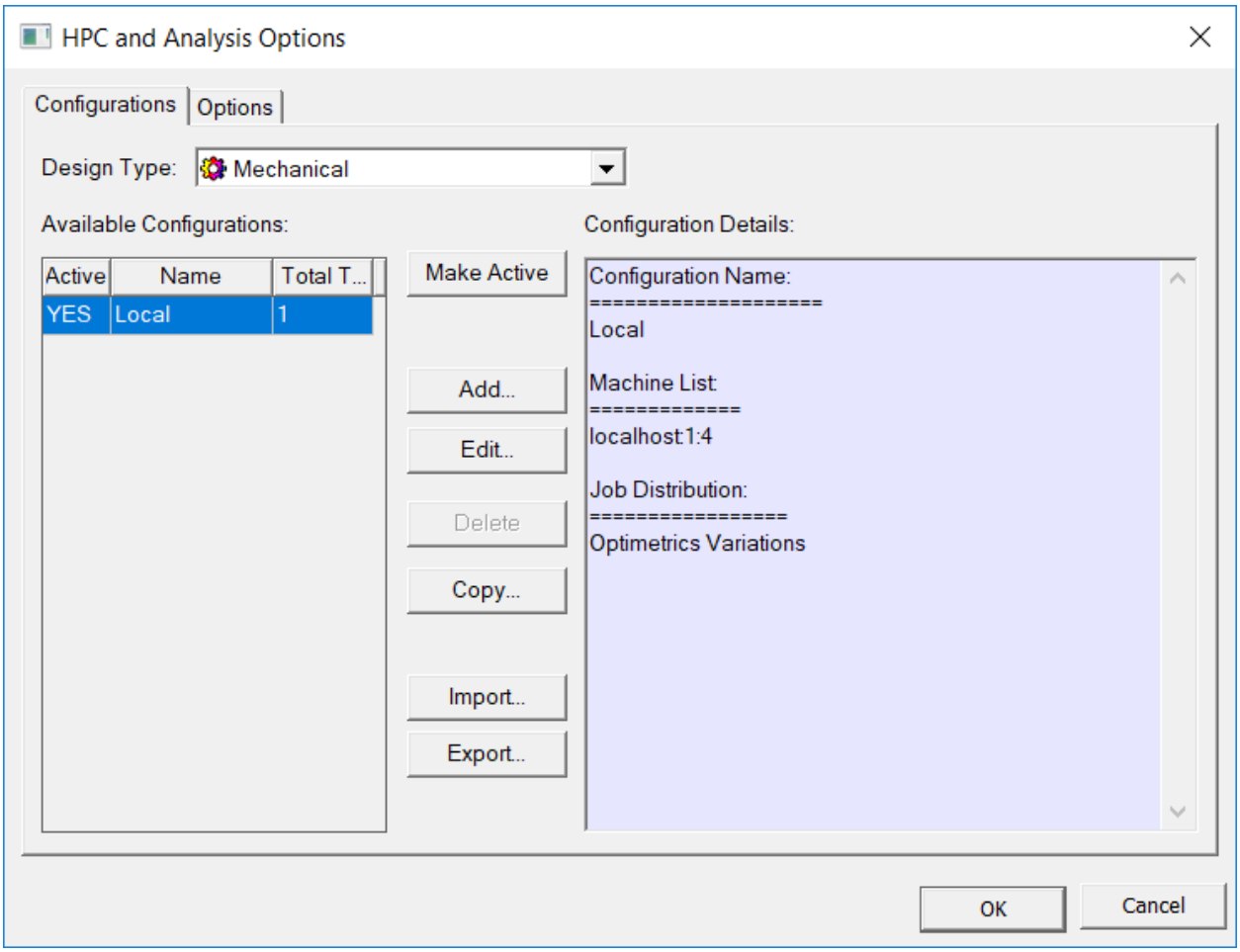


Use the check boxes to enable/disable available distribution types. The job distribution list box allows you to specify which job distribution types to allow for the current analysis configuration. At solve time, the Ansys Electronics Desktop automatically select the best distribution type from the enabled distribution types. By enabling/disabling distribution types, you can control the job distribution.

Just because you enable a distribution type does not mean it will be used. It must be also allowed by the solve setup. Note that the enabled distribution types will apply to all setups of the given design type, so it is possible for different setups in a design to be solved using different distribution types.

The distribution types that you enable here are listed in the **HPC and Analysis Options** dialog box in the *Configuration Details* section. To access this dialog box, click **Tools** > **Options** > **HPC and Analysis Options** from the menu bar or click the **HPC and Analysis Options** command on the **Simulation** ribbon tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adding Configurations or Accepting Edits

Click **OK** to accept the changes and close the **Analysis Configuration** dialog box. Only machines checked as **Enabled** appear on the distributed machines Configuration Details Machine list.

Regardless of the machine(s) on which the analysis is actually run, the number of processors and the default process priority settings are now read from the machine from which you launch the analysis. See Setting HPC and Analysis Options.

For more information, see distributed analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The option is only active if there are multiple rows listed in the parametric table, there are multiple frequency sweeps listed under a given analysis setup, and the number of distributed analysis machines is two or greater.

Selecting Optimal Configurations for Distributed Analysis

This section provides some basic guidelines for configuring a distributed analysis. In this section, assume that a fixed number of machines and cores are available, and that the goal is to use them as efficiently as possible.

In the majority of situations, configuring a Distributed Solution Option (DSO) in combination with the multiprocessing option to take optimal advantage of the available hardware results in improved speed.

> **Note:**
>
> This topic assumes that there is enough hard drive space and memory for DSO simulations. For multiple DSO simulations on a single machine, the total memory needed is the sum of the memory used by each simulation. For example, consider a discrete frequency sweep in which each frequency point needs 3.5GB. A quad core system with 8GB of RAM would rely heavily on swap, which is highly inefficient. In this case, updating the number of tasks in the machine list is ideal.

Large Scale DSO for Parametric Analysis

Large Scale DSO for parametric analysis operates through a non-graphical batch application called desktopjob. You can run the desktopjob command line to perform parametric analysis DSO. The command-line interface supported by this batch program is similar to the command line used for regular DSO jobs.

Large Scale DSO is used for large scale parallel jobs, which either fail or scale poorly as regular DSO jobs. A Large Scale DSO job does not support the output of full parametric results, but produces reduced datasets corresponding to predefined rectangular plots. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. These CSV outputs can be used directly in downstream applications (for example, Microsoft Excel). They can also be imported as dataset solutions for post-processing. Non-rectangular plots of the design are not extracted. In order to produce a new output you must rerun the analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For a machine with *n* cores, it should be expected that running *n*, single core, distributed simulations in parallel will encounter additional overhead due to the need to spawn *n* unique solve processes. Therefore, it should not be expected to observe an *n* times speed up over the time taken to run *n* analyses in series with a single core. The relative impact of this overhead increases as the size of the simulation and time to complete a single solve shrink.

The basic Large Scale DSO process involves

1. Preparing the model for Large Scale DSO Analysis.
2. Submitting the Large Scale DSO job through **Tools** > **Job Management** or via command line.
3. Monitoring the job's progress.
4. Postprocessing the results.

For details, refer to the following sections:

- Prerequisites for Large Scale DSO
- Job Management Interface for Large Scale DSO
- Command Line Syntax
- Deployment/Configuration
- Job Outputs
- Job Monitoring
- Process Limits for Large Scale DSO on Windows
- Known Issues/Troubleshooting for Large Scale DSO

Large Scale Distributed Solve Operation could submit a parametric setup to be solved in multiple machines, each machine may launch multiple EM-Desktop processes to solve the assigned variations (Design Points). Variations are distributed to each task (EM-Desktop process) equally, regardless of the machine hardware and each variation’s complexity. In practice, some tasks may finish earlier than others, in some extreme case some tasks may hours behind fastest task. DSO can redistribute tasks when a task finishes before other task. Variations are removed from slow tasks and reassigned to fast tasks. If you abort a task, they can be reassigned to the running task, when the running task finish its original assignment. For more information, see Large Scale DSO theory.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

Aborting a Large Scale DSO Simulation

To abort the whole Job, select the **Abort** button on the Job Monitor dialog.

To abort using the Job progress bar, click the button next to the Job progress bar, Click the **Abort** menu item in the popup.

To abort all tasks in a Node(host), click the button next to the node progress bar., and click the **Abort** menu item in the popup. Aborted Variations will be redistributed to other running Nodes, if redistribution is turned on.

To abort an individual task, click the button next to the node progress bar, and click the “Detail” menu item in the popup. In the Task status dialog box, click the **Task** button in the grid. In the variation status dialog box, click **Abort** button. Aborted Variations will be redistributed to other running tasks, if redistribution is turned on.

To terminate a hanging EM-Desktop process, the hanging ansysedt.exe process won’t respond to the first abort command. Send a second abort command to terminate the hanging process. Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

For an EM-Desktop process crashed or killed by Windows Task manager or other tools, the task status will be shown as aborted, Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

Prerequisites for Large Scale DSO

General Prerequisites

- Ansys Electronics Desktop must be installed on the cluster which runs either a supported scheduler or Ansys RSM.
- The cluster is compatible with Large Scale DSO requirements.
- Every node of the cluster supports the disk space (in temp directory) and memory requirements of multiple engines that run in parallel.
- All the machines allocated to Large Scale DSO job must all come from the same platform, either Windows or LINUX.
- On the Windows platform, Ansys RSM is started as an admin account, rather than as a system account.

> **Important:**
>
> Large Scale DSO does not support RSM Service running with system login credentials.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On each machine of the cluster, the 'desktopjob' application is registered with Ansys RSM service using the command shown below:

  Windows: `<installation-directory>/<platform> desktopjob.exe -regserver`

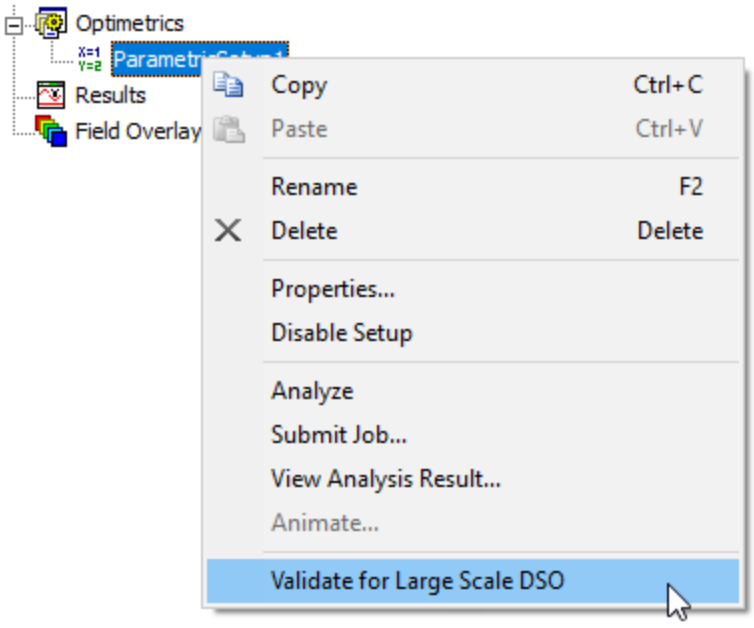
  LINUX: `<installation-directory>/<platform> desktopjob -regserver`

Job Management Interface for Large Scale DSO

Large Scale DSO jobs run only in non-graphical batch mode, irrespective of the scheduler environment. This is in contrast to a Regular DSO job, which can run in graphical mode. This consideration implies that projects corresponding to Large Scale DSO job must be saved and closed prior to job submission. Secondly, the command to submit a Large Scale DSO job is only available through **Tools** > **Job Management** or via a command window, while a Regular DSO job can be run in an RSM environment by right-clicking directly on the parametric setup. The Job Management window is accessed by running Ansys Electromagnetics product Desktop on the designated Postprocessing Node of the cluster. The Desktop provides UI commands for scheduler selection, job submission and job monitoring/control.

When you have selected the scheduler, perform the following steps to submit a Large Scale DSO job:

1. Set up and prepare the model on your local workstation. Right-click the desired solution setup and select **Validate for Large Scale DSO**.



2. Correct any errors and save the project.
3. Close Electronics Desktop.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Copy the project (or folder, if the project references external files) from a personal workstation to a shared drive on cluster.

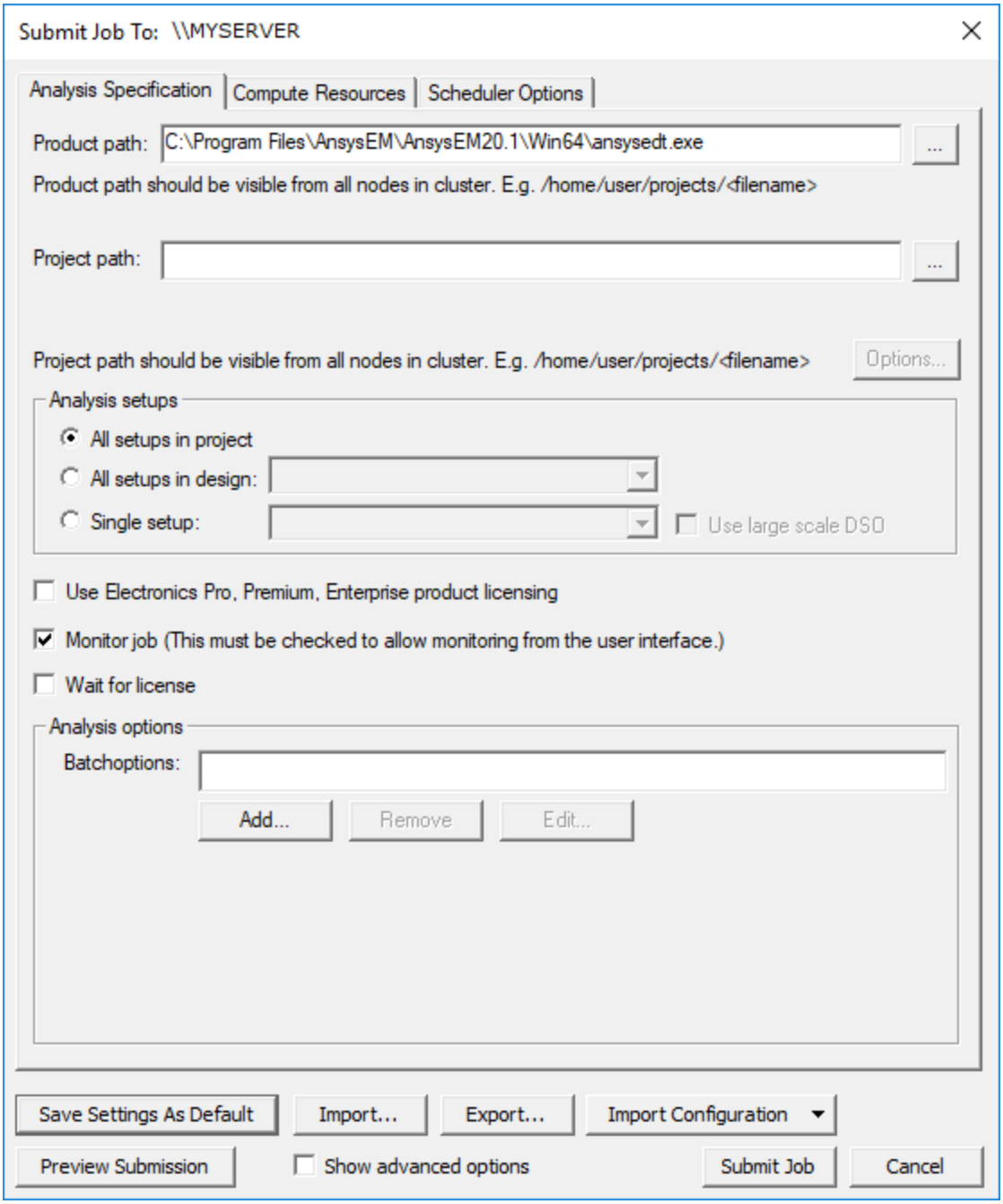
   In the RSM environment, you must specify a machine list (See: HPC and Analysis Options).

   In a Linux scheduler environment, a cluster must have a designated Postprocessing Node.

5. Open a remote desktop session (or equivalent) on the node corresponding to the first machine in the machine list (the designated Postprocessing Node on Linux).
6. Launch Electronics Desktop on that node and open the project.
7. Verify that the model has been prepared correctly.
8. Close the project.
9. Submit the job one of several ways:
   - Click **Tools** > **Job Management** > **Submit Job**.
   - Click **Project** > **Submit Job**.
   - Click **Mechanical** > **Submit Job**.
   - Select the **Simulation** tab and click the **Submit** icon.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Options vary slightly depending on the selected scheduler. See: HPC Integration.

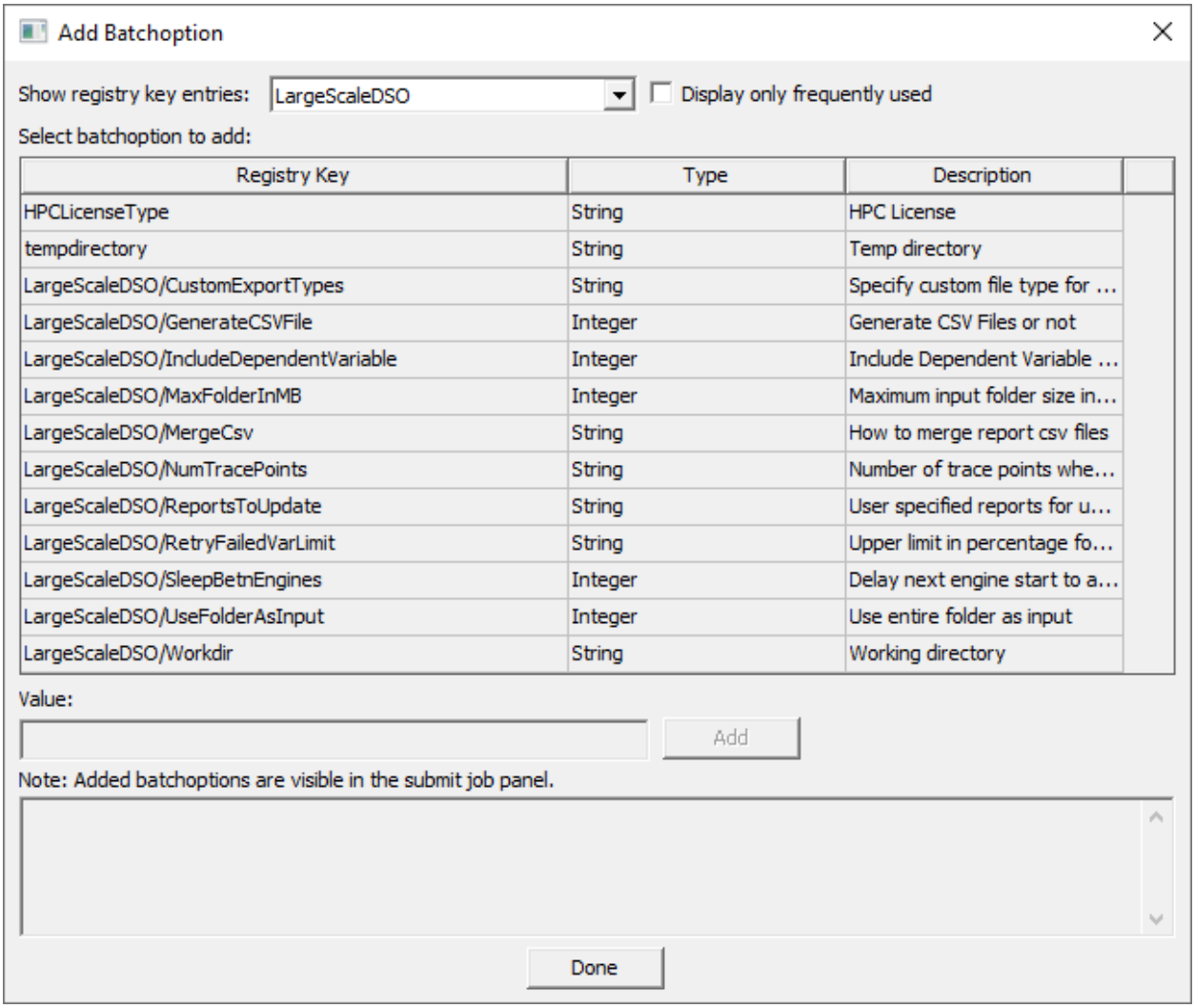
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Specify information for all fields:
    - Most options will not be enabled until you select a **Project path**.
    - Under **Analysis setups**, select setups for analysis.

      For Large Scale DSO, select a **Single setup** from the drop-down menu and ensure the **Use large scale DSO** check box is selected.

    - Determine whether to **Use Electronics Pro, Premium, Enterprise product licensing**, whether to **Monitor job**, and whether to **Wait for license.** If you wish to monitor jobs through **Tools** > **Job Management**, you must enable job monitoring.
    - Add **Batchoptions**, if desired.

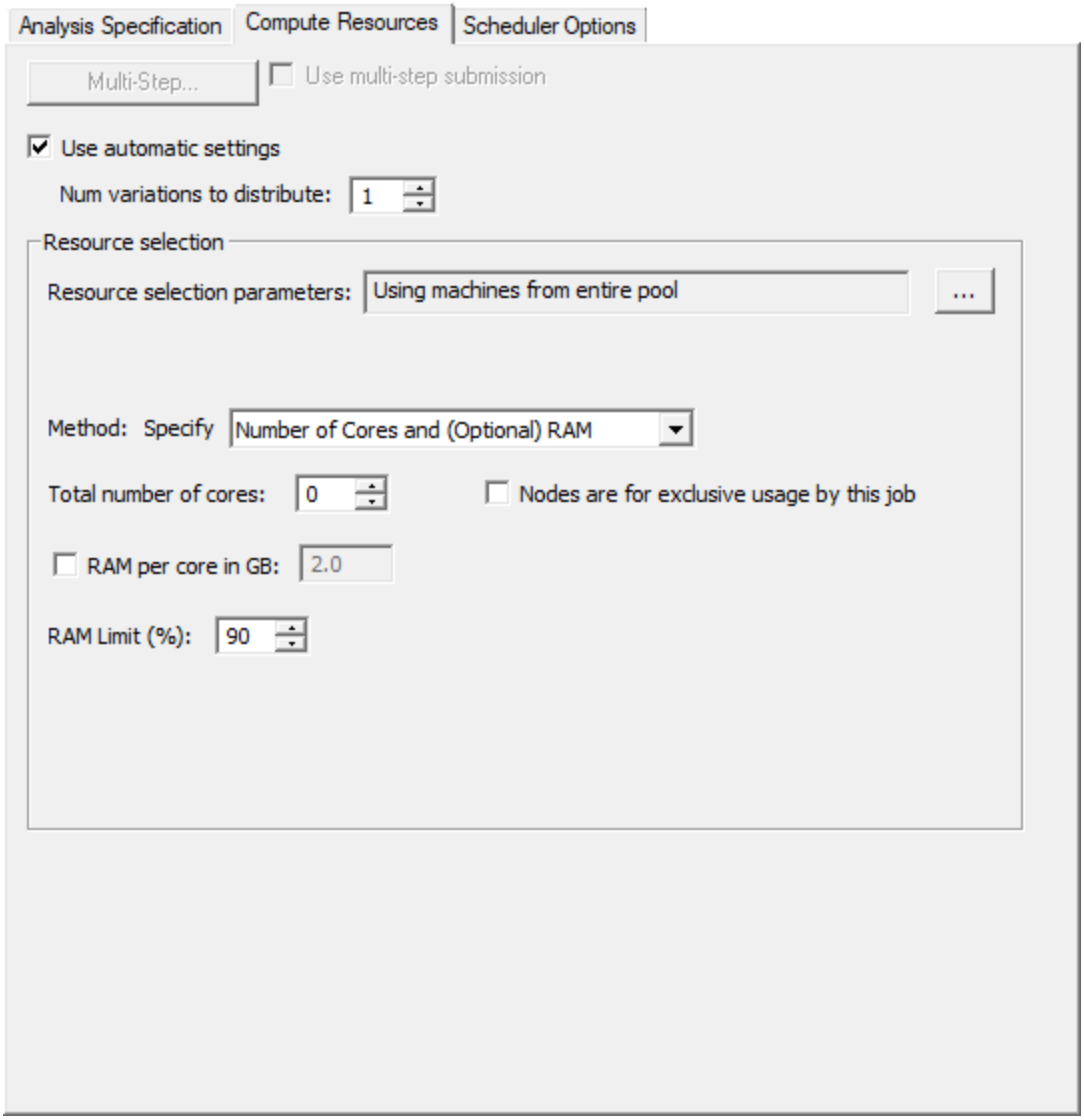
      The following shows batchoptions specific to Large Scale DSO.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Select the **Compute Resources** tab.

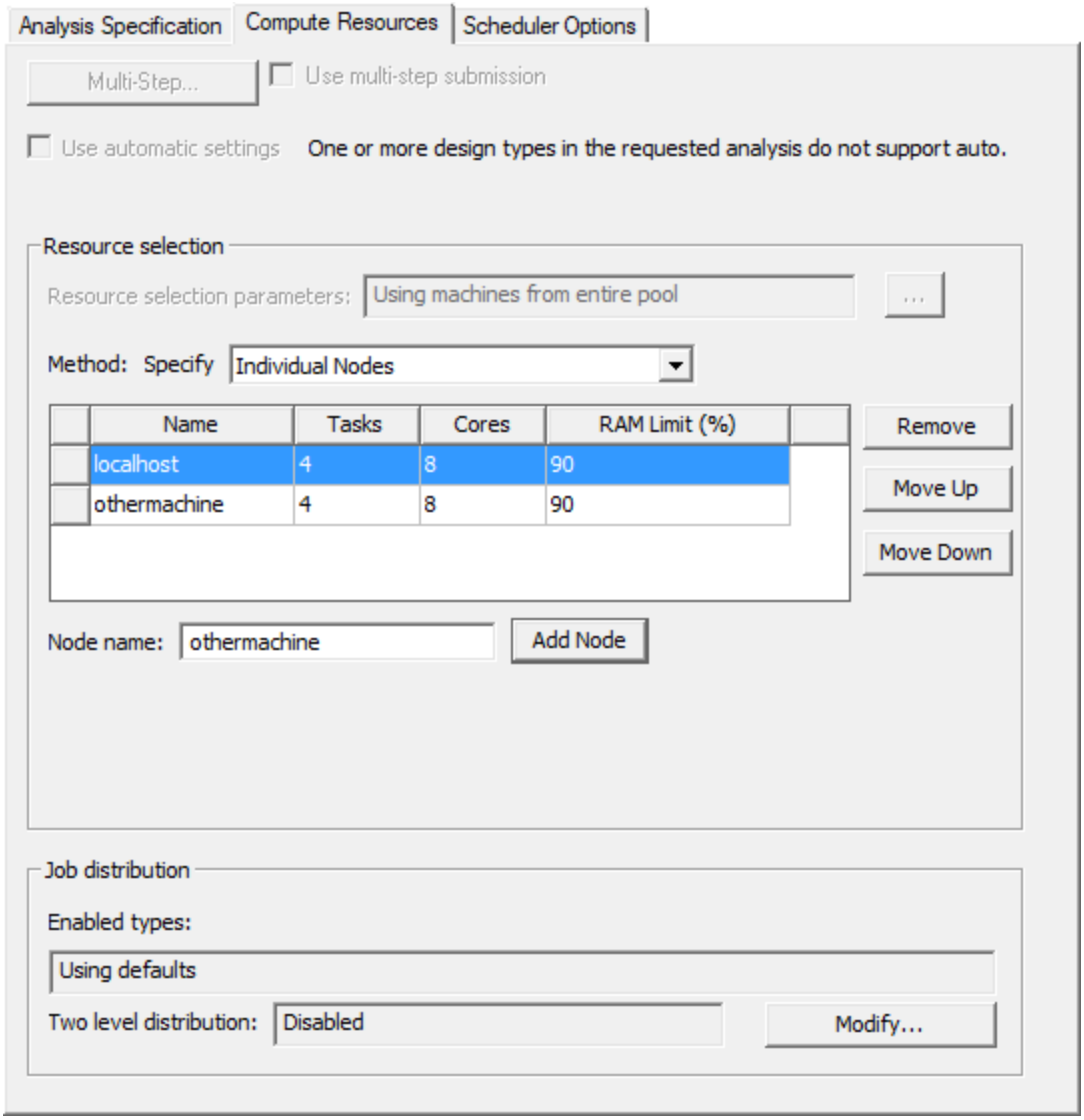
    When available, **Use automatic settings** may be selected. If it is selected, also select the Number of variations to distribute.



    The values you specify represent minimal requirements for each condition that can interact in leading to the total resources the Scheduler derives from them. A submission preview shows the number of resources assigned.

    When automatic settings are not available or not selected, additional options appear:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For RSM Large Scale DSO jobs submitted from the Job Submission panel, localhost must be the first node in the resource selection panel. Otherwise, the job will fail.

In the **Job distribution** area, you can enable or disable **Two-level distribution** by clicking **Modify**. See: Two-level Distribution Guidelines.

> **Note:**
>
> The **Enable two level** option under *Distribution levels* is not currently supported for Mechanical designs. Keep the default **Single level only** option for *Mechanical – Modal* or *Mechanical – Thermal* solutions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. If desired, click **Preview Submission** to view a summary of the commands to be sent to the scheduler. The text can be copied to the clipboard.
13. To submit the commands to the scheduler, click **Submit Job**.

> **Note:**
>
> The RSM environment does not support queuing, so clicking **Submit Job** starts it immediately.

14. If you enabled job monitoring, you can monitor the job via **Tools> Job Management > Monitor Jobs**. See: Monitoring Jobs.

Large Scale DSO Command Line Syntax

Large Scale DSO operates through a non-graphical batch application called `desktopjob`. You can run the `desktopjob` command line to perform parametric analysis DSO. The command-line interface supported by this batch program is consistent with the command line used for current DSO jobs. `desktopjob -help` lists all available command-line options as shown below:

Command Line Syntax:

```
desktopjob.exe <options> <project-path-on-shared-drive>
```

Note that the project path can be to an archive file.

Options:

- **-help**

  Prints the help text.

- **-cmd**

  Specifies the command to run.

  Available choices: dso

- **-ng**

  Runs the analysis in non-graphical mode.

- **-monitor**

  Outputs progress and messages to standard output/error.

- **-waitforlicense**

  Queues the job until licenses are available.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-preserve**

  Preserves the local storage space of the distributed job for investigation into the job's run. If local storage directory (for example, the temp directory) is provisioned by scheduler, ensure it is also configured to preserve the job's local storage. *This storage should be deleted manually*.

- **-batchoptions**

  Overrides the Tools/Option entries through either a batchoptions file or batchoptions string.

  -batchoptions specific for Large Scale DSO include:

  To retry failed variations, new batch option LargeScaleDSO/FailedVarRetryCount.

  For each task to re-simulate its failed variations when its assigned variations are finished, specify the FailedVarRetryCount batch option with positive integer. Default 0; Value-Zero(0) will disable the re-simulation for failed variations. Value ranges from 0 or positive integer.

  Redistribution batch option, LargeScaleDSO/VarRedistribution .Value 0 disables redistribution (default). Value 1 enables redistribution.

  Redistribution limit batch option, LargeScaleDSO/RedistributionLimit. Minimum estimated remaining time (in minutes) for variations to redistribute to another task. Value must be positive integer. Default 3 minutes

  Example:

  ```
  -batchoptions <config-file-on-shared-drive>
  -batchoptions "'name1'='val1' 'n2'='v2' "
  ```

- **-jobid**

  Specify a custom job ID for the job. The job's output is organized into a folder with job ID name. This parameter is ignored when run under a scheduler.

- **-machinelist**
  - In the context of **Ansoft RSM**:

    Specify machines for distributed analysis. Machine list is specified either inline (as a comma separated machine names) or through a file. Multiple cores are specified by repeating the name of machine or by embedding number of cores in the machine name, using a colon separator.

    Example 1:

    ```
    -machinelist "list=m1,m1,m1,m2,m2,m3"
    ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2:

```
-machinelist "list=m1:3,m2:2,m3"
```

Example 3:

```
-machinelist "list=m1:1:3,m2:2:2"
```

Example 4:

```
-machinelist "file=machines.txt"
```

- In the context of **a scheduler such as LSF**:

  Specify the portion of total machines for distributed analysis. Use remaining for overhead or shared memory multiprocessing.

  Manual Example:

  ```
  -machinelist "Num=10"
  ```

  Auto Example:

  ```
  -machinelist "NumCores=40"
  ```

  Note: In a scheduler environment, the machinelist argument can be omitted, but the results may not be as expected or desired. If there is no machinelist argument, the scheduler-assigned resources will be used, assuming one core per task. E.g., if the scheduler assigns 40 cores, there will be 40 variations solved in parallel, with only one core used per variation. Since one core per task may not make sense in many cases or may not be what the user desires, a warning message that machinelist is missing will be output, but the job will proceed.

- **-auto**
  - Run the leaf jobs in auto mode. See Submitting Large Scale DSO Job Examples.
- **-numdistributedvariations**:

  Specify the number of parallel leaf jobs to run in auto mode. Required when running under a scheduler. If not specified when not running under a scheduler, then the number of tasks for each machine must be specified in the machine list. The total number of tasks will be the number of parallel variations (leaf jobs) to run.

  Example:

  ```
  -machinelist NumCores=40 -auto -NumDistributedVariations 10
  ```

- **-usefolderasinput**

  Choose this option if the job's input represents an entire folder rather than just the project file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-maxfolderInMB**

  Specify the maximum size input folder that is allowed for a valid job (in MBytes). By default, the maximum size allowed for input is 10MB. Specify a value of 0 to remove this size restriction and enable inputs of any size. This option applies when `-usefolderasinput` is used.

- **-workdir**

  Specifies the shared drive folder for status and result files generated by analysis. By default, the results folder of input project is used as the work directory.

- **-meregecsv: [acrossDPs | singleDP | both]**

  across DPs: Merge report csv files for all design-points (variations). One file is created per trace, across all variations.
  singleDP: Merge csv files within a single design-point (variation). One file is created per variation, per a set of traces that can be merged.
  both: Merge all traces that have the same primary sweep for all design-points (variations) into one csv file.
  Interpolation note: If primary sweep values are not uniformly spaced, mergecsv is enabled with traced values and are re-sampled uniformly using '-batchoptions' syntax as shown below:

```
-batchoptions "'LargeScaleDSO/NumTracePoints'=500"
-batchoptions
"'LargeScaleDSO/NumTracePoints'='PrimarySweepName:200'""
-batchoptions
"'LargeScaleDSO/NumTracePoints'='ReportName1:Trace1:100;ReportNam
e1:Trace1:100;ReportName1:TraceName2:200'"
```

- **-abort**

  Abort a running job identified through the job's working directory. Example: `-abort <projectresultsfolder-path>/>jobid>`. For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

- **-repackageresults**

  Choose this option to add simulations results to the input archive file. Note: this option only applies if an archive file is provided as input.

- **-batchsolve**

  Solves the specified parametric setup.

  Syntax for the setup:

  `<design-name>:Optimetrics:<parametric-setup>`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO Job Outputs

A Large Scale DSO analysis does not support the output of full parametric results. Instead, it extracts a subset of results using predefined rectangular plots, which are created by the user before running the job. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. The outputs can be either imported as datasets for post-processing in the desktop also as function of parametric variations, or used directly in downstream applications (for example, Excel, or custom programs that parse .csv files).

> **Note:**
>
> Non-Rectangular plots of the design (for example, statistical eye and digital plot) are not extracted.

Output Location and Organization

The results of a Large Scale DSO job are located in the `<workdir>/<jobid>/results` folder. If `workdir` is not specified on the job command line, it is same as the input project's results folder. For example, the default `workdir` corresponding to `\\shared\projects\tee.aedt` is `\\shared\projects\tee.aedtresults`. Within this results folder, there is one folder per variation. The name of the variation's folder is an integer number corresponding to variation's index in the parametric table. For example, a variation folder named '4' has results for the fifth row of parametric table, while a variation-folder named '0' has results for the first row of the table. Each variation's folder contains a CSV file for each trace.

CSV File Contents

The initial header rows of the CSV file define the solved variation. For each such row, the first column contains a variable name and the second contains a variable value. The row following variation rows has the name of primary sweep and the name(s) of extracted quantities. Subsequent rows contain data—quantity values as a function of primary sweep.

These examples provide context:

- **Traces of an S-parameter Report** – The data portion of the CSV file contains two columns of data: the first contains Freq values and the second contains values for the trace's s-parameter component.
- **Trace of a Far Field Report** – Suppose there is a far field report with a trace (magrE), whose primary sweep is phi and whose secondary sweep is theta. Further suppose that two values of theta are chosen and all values of phi are chosen. For this trace, the data portion of CSV file contains three columns of data: the first containing phi values, the second containing magrE values for the first value of theta, and the third column containing values for second value of theta. The magrE output columns are titled as 'magrE_crv1' and 'magrE_crv2' respectively.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Advanced Sweeps** – In the case of a trace with special primary sweep (such as the trace of a time domain quantity), one CSV file is created per curve of trace, per variation. These CSV files always have two columns, irrespective of the number of values chosen for secondary/higher sweeps.

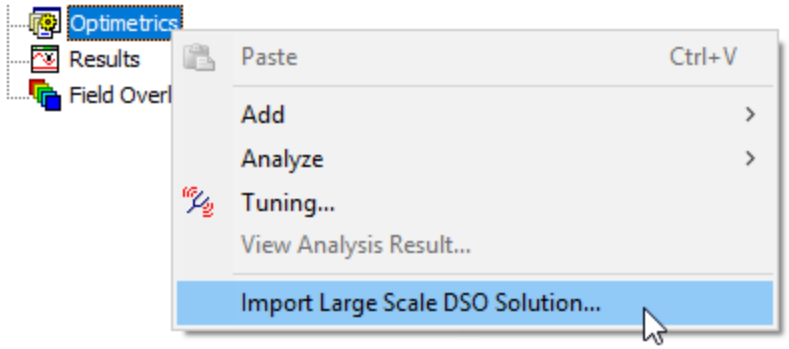
Postprocessing Large Scale DSO Dataset Solutions

In Ansys Electronics Desktop, the **Import Large Scale DSO Solution** command allows you to post-process Large Scale DSO dataset solutions.

Importing a Large Scale DSO Solution

To import a Large Scale DSO solution:

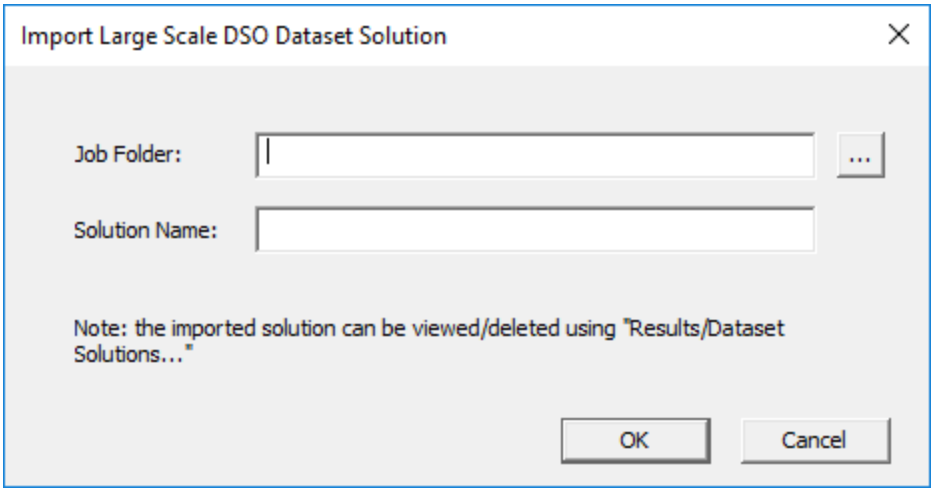
1. In the Project Manager, right-click **Optimetrics** and select **Import Large Scale DSO Solution**.



> **Note:**
>
> You can also access this window by right-clicking **Results** > **Dataset Solutions**.

The **Import Large Scale DSO Dataset Solution** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the ellipsis button (**...**) and browse to select a job folder. To select a results dataset, double-click the results folder name. Results folders are organized by the scheduler prefix (for example, RSM) and job number.
3. Click **Open** to return to the **Import Large Scale DSO Solution** window.
4. Review the **Job Folder** path and **Solution Name**, then click **OK**.

   The dataset is imported.

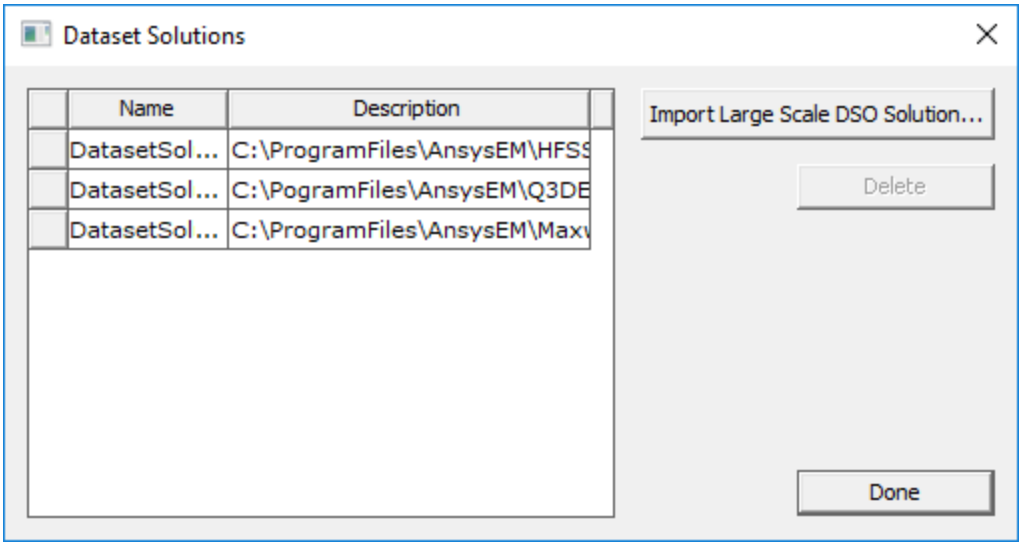
Viewing Imported Solutions

To view a list of imported datasets:

1. In the Project Tree, right-click **Results** and select **Dataset Solutions...**

   The **Dataset Solutions** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting a dataset enables the **Delete** button, which you can click to remove it.

2. Click **Done** to close the window.

Creating a Dataset Report

After importing one or more DSO solutions, you can create a dataset report.

To do so:

1. In the Project Tree, right-click **Results**. Select **Create Dataset Report** > **[Report Type]**.

   The **Report** window for the report type opens. See Creating Reports.

Cloning a Dataset Solution

If you reopen a project that was solved using Large Scale DSO, you can quickly clone the solution report:

1. In the Project Tree, under **Results**, right-click the report and select **Clone from Dataset Solution** > **[Solution Name]**.

   The cloned solution appears. You can now reuse the existing report definition rather than creating a new report.

Large Scale DSO Job Monitoring

The **Monitor Job window** allows you to monitor the progress and status of Large Scale DSO jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Resources for Large Scale DSO Monitoring

Large Scale DSO avoids detailed intra-variation monitoring, as it increases network traffic for large-scale jobs. Additional monitoring resources include:

- **Cluster Monitoring Tools** – Standard cluster monitoring tools are ideal for job-neutral resource monitoring as they use negligible network bandwidth.
- **Detailed Monitoring of Analysis of a Variation** – For detailed monitoring, you may want to examine a job's log files. Large Scale DSO writes detailed logs about the machines where engines are running and the local storage location of per-engine distributed databases. You can log in to individual machines for deeper probing of each distributed engine.

  The following logs are available:

  - **Per-Node Logs** – There is one desktopjob.log file per node assigned to the job. This log contains information regarding the node such as name, local storage folder, and number of engines started on this node. It is located in <workdir>/<jobid>/r<nodeIndex>. For example, <workdir>/<jobid>/r0 contains the desktopjob.log corresponding to the engines running on the first node of job, while <workdir>/<jobid>/r2 contains the log corresponding to engines running on the third node.
  - **Per-Engine Logs** – There is one desktopjob.log file per distributed engine. It is located in <workdir>/<jobid>/r<nodeIndex>/r<taskIndex>. For example, <workdir>/<jobid>/r0/r0 contains the log corresponding to first engine running on first node, while <workdir>/<jobid>/r1/r2 contains the log corresponding to the third engine running on the second node. Engine unique information (such as local storage of this engine) is logged here.
  - **Parametric Analysis Log** – This log file is located in'<workdir>/<jobid>/r<nodeIndex>/r<taskIndex> and corresponds to Desktop's local-machine parametric batchsolve. It is available only at the end of analysis and contains information regarding the variations solved by this engine and any info/warning/error messages.
  - **Root Log** – This is the top-level desktopjob.log file that logs job distribution information such as hierarchical activation and the list of nodes assigned to this job.

For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

Large Scale DSO Deployment/Configuration

LINUX Cluster Configuration

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 'Temp directory' configuration.

  Temp directory is either on 'local storage' or on storage that has equivalent speed characteristics. The I/O rates of the storage should be invariant to network traffic

  Temp directory on a host has sufficient space to hold results database for the variations that are solved on it. (Note: This storage is freed at the end of the analysis.)

  The amount of required space depends on the number of engines per node and the cumulative variations solved on this node

  The amount of required space depends on the project's compression-options. For example, if 'Save Fields' of a parametric setup is OFF, the space requirement is smaller by the amount of space taken up by field solution data.

- Ansoft RSM environment: In the case of supported scheduler environments, there is no extra configuration needed. In the case of Ansoft RSM environment, following additional steps are needed:

  Ansoft RSM must be running on all the nodes of cluster. The credentials of 'RSM service' allow read/write to shared drive. Reason: the remote engine processes are launched using the credentials of RSM service

  Registration of 'desktopjob.exe' with RSM service: 'desktopjob' program must be registered with Ansoft RSM using 'desktopjob -regserver". To ensure that the registration is successful, check that the 'desktopjob' entry in '<RSM-installation-folder>/AnsoftRSMService.cfg' file is valid.

> **Note:**
>
> LINUX specific critical note: Edit AnsoftRSMService.cfg and replace 'desktopjob.bin' with 'desktopjob'

Major limitation: In the Ansoft RSM environment, Large Scale DSO can only be enabled for one product.

Troubleshooting hints (Ansoft RSM environment only): "shared drive read/write" requirement is a new constraint introduced in Large Scale DSO. So if user runs into a situation where Regular DSO jobs run and Large Scale DSO jobs fail, one possible cause for the failure: RSM service does not have privileges to read and write to project folder located on shared-drive.

Windows Cluster Configuration

All the above steps apply, except for steps that are stated as LINUX-specific. Additional instructions:

- Ansoft RSM and Ansys Electromagnetics products are either installed locally on each node of cluster OR installed on a single shared-drive available to all nodes of cluster.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Registration of 'desktopjob.exe' with RSM service.
- Network installation: desktopjob.exe is registered with RSM service once, on any of the nodes of cluster
- Local installation: Since each node has it's own RSM installation, desktopjob.exe must be registered with RSM on each node.

> **Important:**
>
> Ansoft RSM service must be started using the credentials of a non-system 'admin' account, which has read/write permissions to project's shared drive. If RSM service runs as 'system' user, large-scale-dso jobs will fail

Heterogeneous Cluster configuration

Limitation: Currently heterogeneous cluster (with both linux and windows nodes) is not supported. This is due to the shared drive requirement.

Large Scale DSO Tutorial Example

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

This section provides an example of the use of Large Scale DSO to distribute parametric variations of an HFSS model across the nodes of a cluster or to multiple cores of a single machine.

This example presumes that your configuration satisfies the Prerequisites for Large Scale DSO. For this example, pre-suppose that we have a Windows cluster. Further suppose that the shared drive folder, which contains the input projects and computed results is at the location *\\sjo7na1\hfssprojs*.

Major steps for Large Scale DSO Example

1. Prepare the model for Large Scale DSO Analysis
2. Submit the Large Scale DSO Job
3. Post process the results

Prepare the Model for Large Scale DSO Analysis

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

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Launch the desktop.
2. Create the input project on the shared drive.

   For this example, start with the standard HFSS OptimTee.aedt example and copy it to the shared drive.

   Copy "<installation-directory>\<platform>Examples\RF_Microwave\OptimTee.aedt" to "\\sjo7na1\hfssprojs\OptimTee.aedt"

3. set up the parametric table.

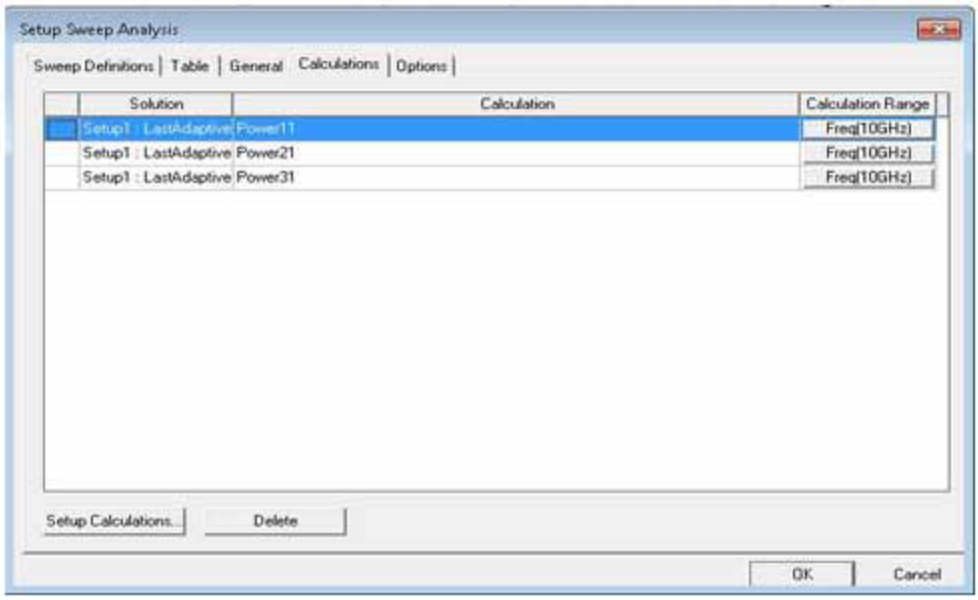
   For this example, use the existing 'ParametricSetup1' as the parametric setup to solve.

4. Prepare the outputs.

   Outputs from Large Scale DSO come from pre-defined rectangular plots that are created before the Analysis command is issued. Follow the steps below:

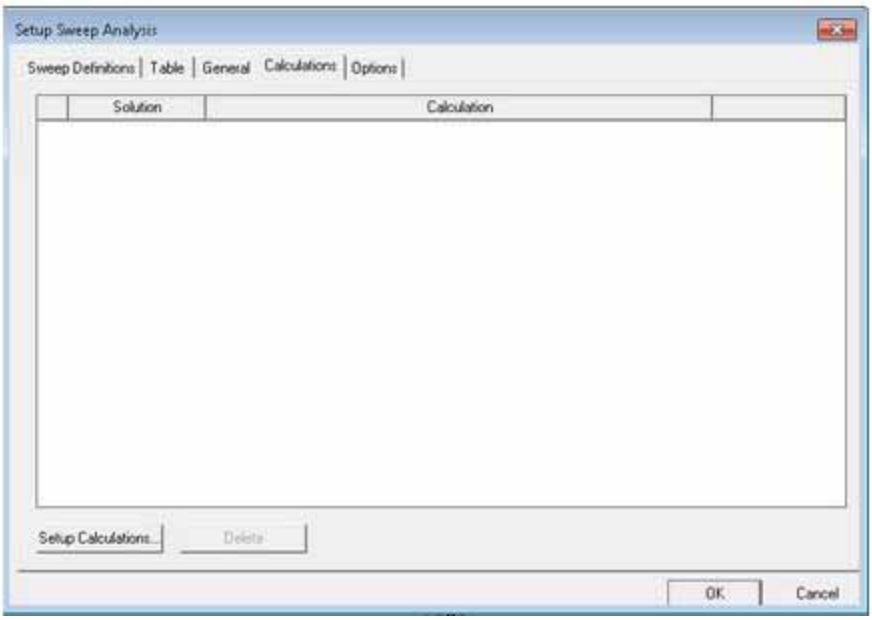
   a. Because these DSO outputs come solely from Rectangular Plots, delete all other postprocessing setups, and then turn off Save Fields And Mesh as shown below.

      Open ParametricSetup1 for editing. You will see that the OptimTee parametric setup contains three calculations, as shown below.



   b. Delete all three calculations. When you have done so, the Setup Sweep Analysis Calculations tab looks like this:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



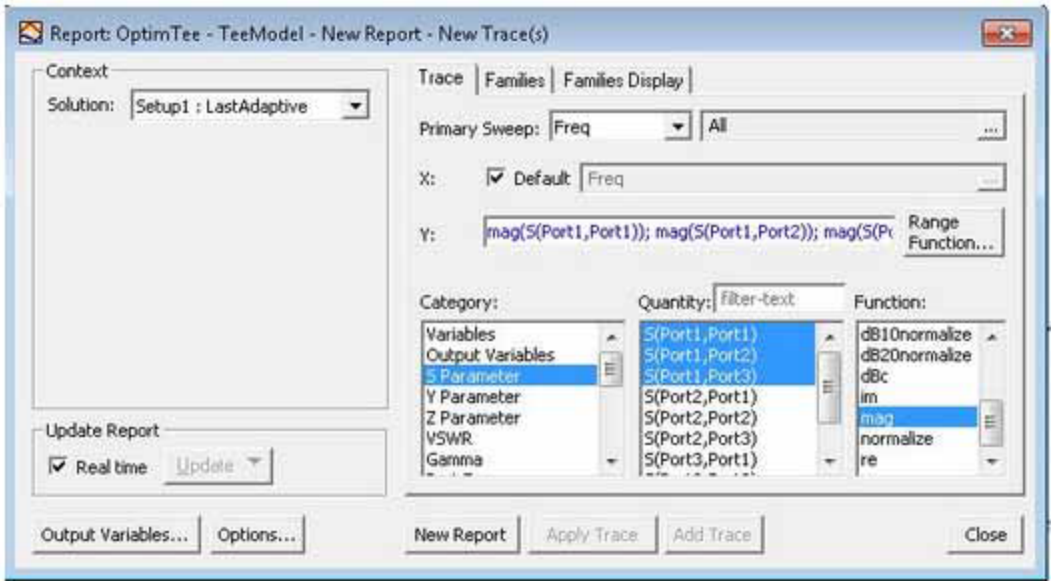
c. Click on the **Options** tab and uncheck Save Fields And Mesh, as shown below:

5. Use the Reporter to define outputs.

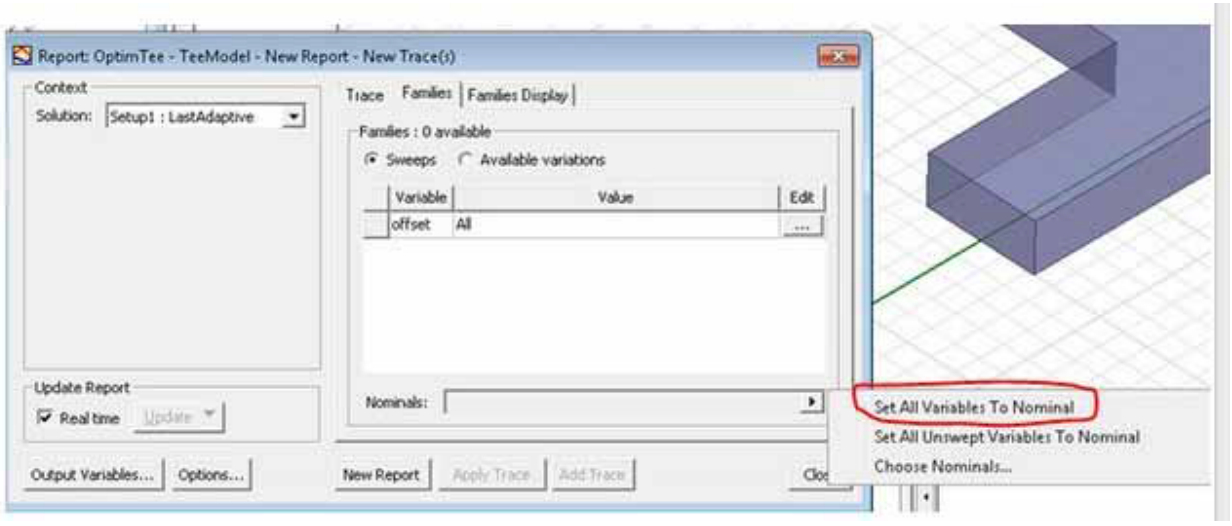
   For this example, you add six traces that correspond to six csv outputs of the Large Scale DSO job: mag(S11), mag(S12), mag(S13), Power11, Power12, and Power13.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. As shown below, select three quantities: mag(S11), mag(S12), and mag(S13).



b. Click on the Families tab and ensure that all variables are set to Nominal, as shown below.



> **Note:**
>
> For Large Scale DSO, outputs are not extracted correctly unless all variables on the **Families** tab are set to nominal.

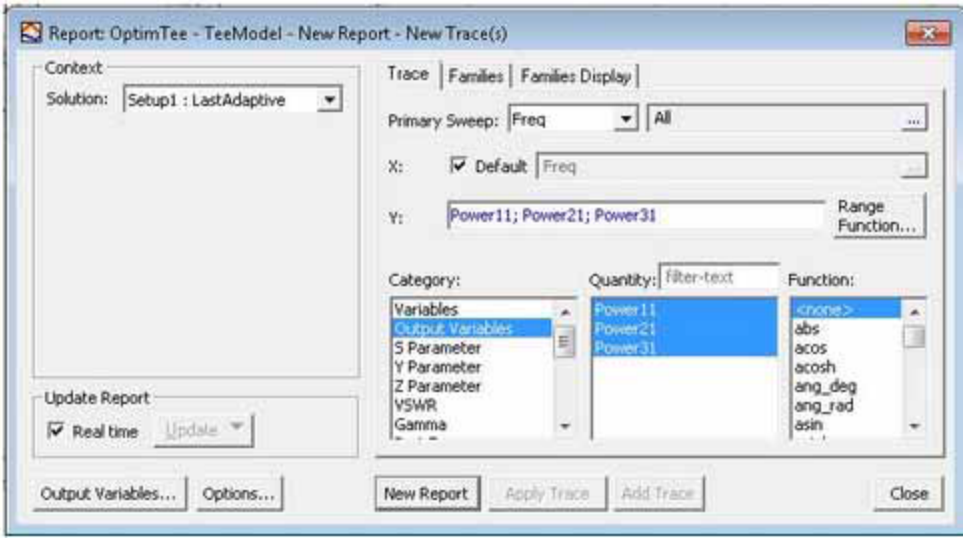
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click the **New Report** button.

   Use the Report to create a Power distribution plot that has three power distribution traces.

d. To create an S-parameter plot that has three traces, click **Results> Create Modal Solution Data Report> Rectangular Plot**.

e. As shown below, select three quantities, Power11, Power21, and Power31.

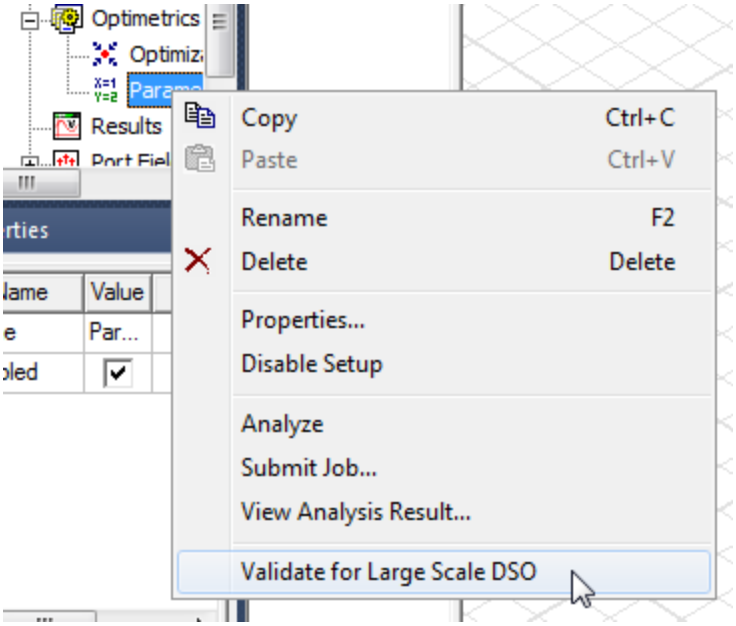


f. Click on the Families tab and ensure that all variables are set to Nominal.

g. Click the **New Report** button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Right-click the parametric sweep, and select Validate for Large Scale DSO.



7. A dialog reports any errors. These may occur if the steps just taken are neglected:

   ```
   [error] Please remove all the calculations in Parametric Setup/Calculation page.
   ```

   ```
   [error] Please turn off "Save Fields And Mesh" in Parametric Setup/Options page.
   ```

   ```
   [error] No rectangular plot exist in the design. Please create a rectangular report.
   ```

8. Correct any errors, if necessary to pass validation.
9. Save and Close the project.

Next

Submit the Large Scale DSO Job: Examples

Submit the Large Scale DSO Job: Examples

This section includes examples of submitting a Large Scale DSO job using Ansoft RSM and a Scheduler.

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

Using the Ansoft RSM Environment

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Before submitting the job make sure that the input project is not open in any Ansys Electronics Desktop window.
2. For this example, suppose that there are two quad-core machines on your cluster with the names "m1" and "m2". Further suppose that there are two engines per machine, for a total of four parallel engines. Let the number of processors allocated to each engine be 1.
3. From a command prompt, issue the following command:

   ```
   "<installation-directory>\ANSYS
   Inc\v251\AnsysEM\desktopjob.exe" -cmd dso -machinelist
   "list=m1:2,m2:2" -batchoptions
   \\sjo7na\aedtprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"
   \\sjo7na\aedtprojs\OptimTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the following contents:

   ```
   $begin Config
     'Desktop/Settings/ProjectOptions/NumberOfProcessors'=1
     'HFSS/Preferences/SaveBeforeSolving'=0
     'HFSS/Preferences/HPCLicenceType'='pack'
   #end 'Config'
   ```

4. Suppose the above job is assigned ID "jobID".

Using a Scheduler Environment (such as LSF)

1. Suppose you want to solve variations using four parallel engines, each engine being assigned a single core.
2. From a command prompt, run the following command:

   ```
   bsub -n 4 "<installation-directory>\ANSYS
   Inc\v251\AnsysEM\desktopjob.exe" -cmd dso -batchoptions
   \\sjo7na\hfssprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"\\sjo7na\hfssprojs\Opti
   mTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the same contents as the RSM example above.

3. Suppose the above job is assigned an ID "jobid"

For example with 2 nodes of 44 cores each (with multiple distributed variations per machine):

MANUAL: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist num=8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Mechanical/NumCoresPerDistributedTask'=11`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
'Mechanical/'HPCLicenseType'='Pack'" -batchsolve
TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt
```

AUTO: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=88 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Mechanical/'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

For example, with 16 nodes of 44 cores each (note each distributed variation will span 2 machines):

AUTO: `bsub -n 704 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=704 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Mechanical/'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

Next

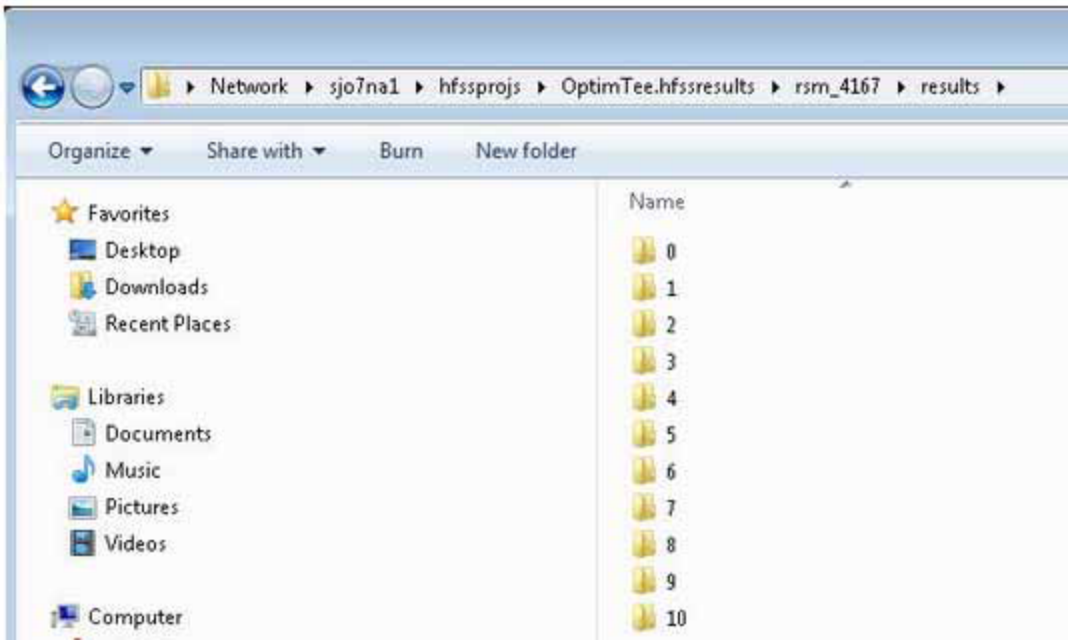
Large Scale DSO Example: Post Process the Results

Large Scale DSO Example: Post Process the Results

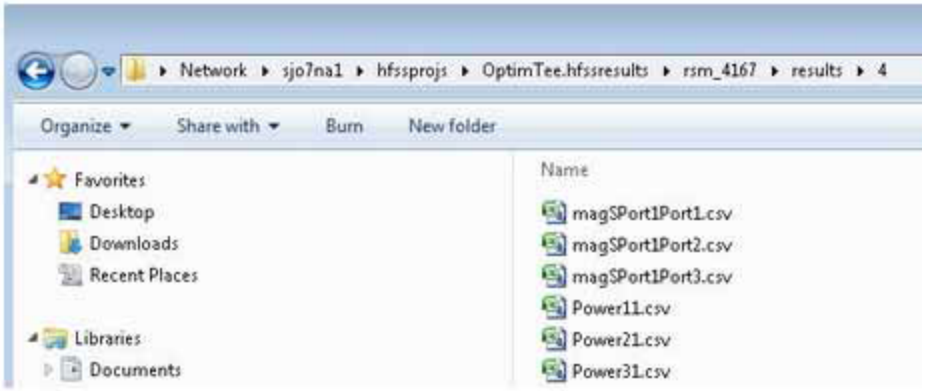
Once the job is done, output is available in the ~\OptimTee.aedt\jobid\results folder. Each variation creates a subfolder, which in turn has one csv file per trace of each report. See the detailed information regarding job monitoring and the location of the analysis logs.

The following figure shows the results for 10 variations as located in 10 folders.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



There are six csv files corresponding to three S-parameter traces and three power distribution traces, as shown below for the fourth variation.

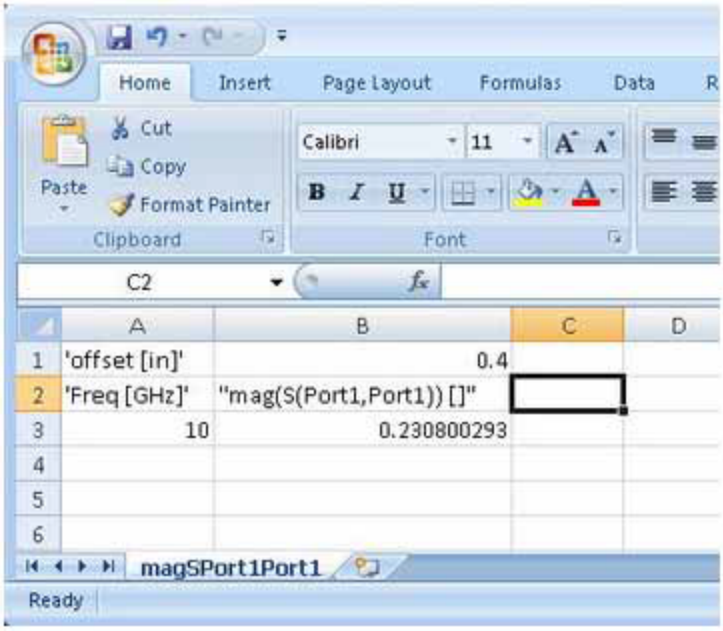


You have three options for postprocessing csv files.

- Import Large Scale DSO Dataset Solution
- Use Microsoft Excel or any other application that has csv post processing functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Parse the csv output into your custom program, for any downstream flow.



Large Scale DSO Known Issues/Troubleshooting

Node Order

For Large Scale DSO jobs that are submitted from job submission panel using RSM, localhost must be the first node in the resource selection node list, otherwise Large Scale DSO solve with RSM will fail. Set the order in the **Submit Job To** window, **Compute Resources** tab:

	Name	Tasks
	localhost	1
	othermachine	1

Parallel Task Limitation for LS-DSO Parametric Variations

There some limitations of running short parametric runs in parallel and using a subset of cores and/or running across multiple machines instead of just one. LS-DSO does help in getting close to linear scalability for parametric runs in many scenarios, it has some overheads and typically they are small in comparison to total time of run. However, a few factors in particular scenarios can make them prominent.

The startup of each task involves copying the project and launching ansysedt to solve a subset of parametric table. Typically, one task solves several variations and this startup cost is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

negligible in comparison to total time. Consider a case where each task is solving only single variation. Additionally the actual solve time of one variation is relatively small, 3-5 minutes. These factors make the startup cost significant. There is also some variance in solve times of different variations. When all variations are solved in parallel, the total time is determined by slowest variation, even though the expectation intuitively might be relative to average time. That could make the perception of overhead worse in this case.

Another important factor is the number of parallel tasks on each machine relative to the total number of cores on the machine. Each task is effectively running an ansysedt of its own and solving subset of parametric table on a copy of project. The solve process does involve significant disk IO. If you run too many tasks on a single machine, they may end up competing with each other for single disk, and that could cause them to slowdown. The conflict in memory access may also become a factor.

In a particular case, 26 tasks were run on one 26 core machine, and this caused significant slowdowns. Spreading the tasks across machines helped scalability as an example with a 32 core machines in a Linux Cloud, we ran this job with 26 tasks on a single machine and then another one over two machines with 13 tasks per machine. The single machine job finishes in about 14 mins, while the job on two machine finishes in about 8.5 mins. This indicates that spreading tasks across machines helps with scalability by minimize file read/write conflicts.

Job Restart

There is no provision for stopping and restarting a job. A new job does not reuse solved results; it always solves all rows in the table. An abort or failure of a job restarts from the beginning, unless a new parametric table with the unsolved rows is created.

Linux-Only Issues

- Deployment/Installation errors (such as mainsoft-related) are not captured. If there is such an issue, the Large Scale DSO job will fail without useful messages in the logs.
- Report-based extraction fails if traces and parametric-setup are not prepared as per the Getting Started guides.
- Job status: The exit code of job doesn't indicate success or failure correctly. The error messages from multiple log files needs to be combined to determine the reason for failure. In many situations, the reason for a failure is apparent only after re-running the job after turning ON the 'debug logging'.
- In some LINUX scenarios, the analysis appears to finish successfully with valid results, except that the exit code is '134'. In this case, although the exit is abnormal, the failed exit code can be ignored.
- Load Balancing: For models with 'unbalanced variations table' (variations that take considerably different amount of time to solve are clustered in few regions of table), job

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will take longer time to solve than a Regular DSO as the job's overall completion time is determined by the slowest solving region. Workaround: rearrange the rows in the parametric table so that each region takes a similar time to solve.

- GM Specifics: the model used for 'Report-based extractor' jobs is NOT compatible with the 'Ansys-extractor-for-GM' jobs. A valid model for Ansys-extractor-for-GM cannot contain any of: reports, overlay plots, Optimetrics calculations.

Interactive Scheduler Jobs

This document includes information, guidelines, and caveats for users running interactive scheduler jobs on Linux.

In most cases, jobs run under a scheduler run as a batch job. These jobs may be submitted using the Ansys Electromagnetics Desktop job submission GUI, using cluster job submission commands on a command line or using a cluster GUI, if available.

Some customers use an alternative method for submission of scheduler jobs. For convenience, we call such jobs "interactive scheduler jobs." In this approach, the user submits an interactive job to the scheduler. From the interactive job prompt, the user launches an Ansys Electromagnetics Desktop product, which starts in interactive (GUI) mode, not batch mode. The user selects a project and then runs one or more analysis commands using the GUI. The intent is that these analysis commands should use all resources allocated to the job, whether on the same host as the GUI or on other hosts.

This approach is supported on Linux, where the user may set up an X Window System server for interacting with the Ansys Electromagnetics Desktop product GUI. The user needs to configure the cluster environment and/or the interactive environment so that the user may view and interact with the product GUI. This approach is not supported on Microsoft Windows.

Specifying Options for Interactive Scheduler Jobs

DSO Configuration for Interactive Scheduler Jobs

Design Type Options for Interactive Scheduler Jobs

Specifying Options for Interactive Scheduler Jobs

Batch scheduler jobs use command line options for specifying options rather than GUI controls. For interactive scheduler jobs, options may be specified on the command line, using GUI controls, or obtained from the registry. The machines specified in the **Configurations** tab of the **HPC and Analysis Options** window will be ignored for interactive scheduler jobs. If the command line used to launch the product contains a list of specific machines (using option -machinelist list=...), then the job will use the specified machines and cores. If the command line used to launch the product does not contain a specific machine list, then the machines and cores allocated to the job by the scheduler will be used for the job. In general, the job distribution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

settings for interactive scheduler jobs should be specified on the command line, instead of using the **HPC and Analysis Options** window.

- The UI shows the settings that will be used for analysis, even if they come from the command line, not the registry.
- If you make changes to the settings in the UI, and the changed settings will be used for the analysis, even if the changed settings conflict with the command line (including batchoptions).
- If you makes no changes to the settings in the UI, then the command line settings (including batchoptions) will be used for analysis.

Batchoptions settings that are automatically generated for batch jobs submitted using the AnsysEM job submission GUI will need to be manually included in the product command line. Some use cases require that certain settings be made on the command line when the product is launched, rather than using the GUI.

Batchoptions for Interactive Scheduler Jobs

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

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Line Example:

```
desktopjob.exe" -cmd dso -jobid RSM_27086 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsAndTraces'"
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
desktopjob.exe" -cmd dso -jobid RSM_25248 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsByRow'"
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

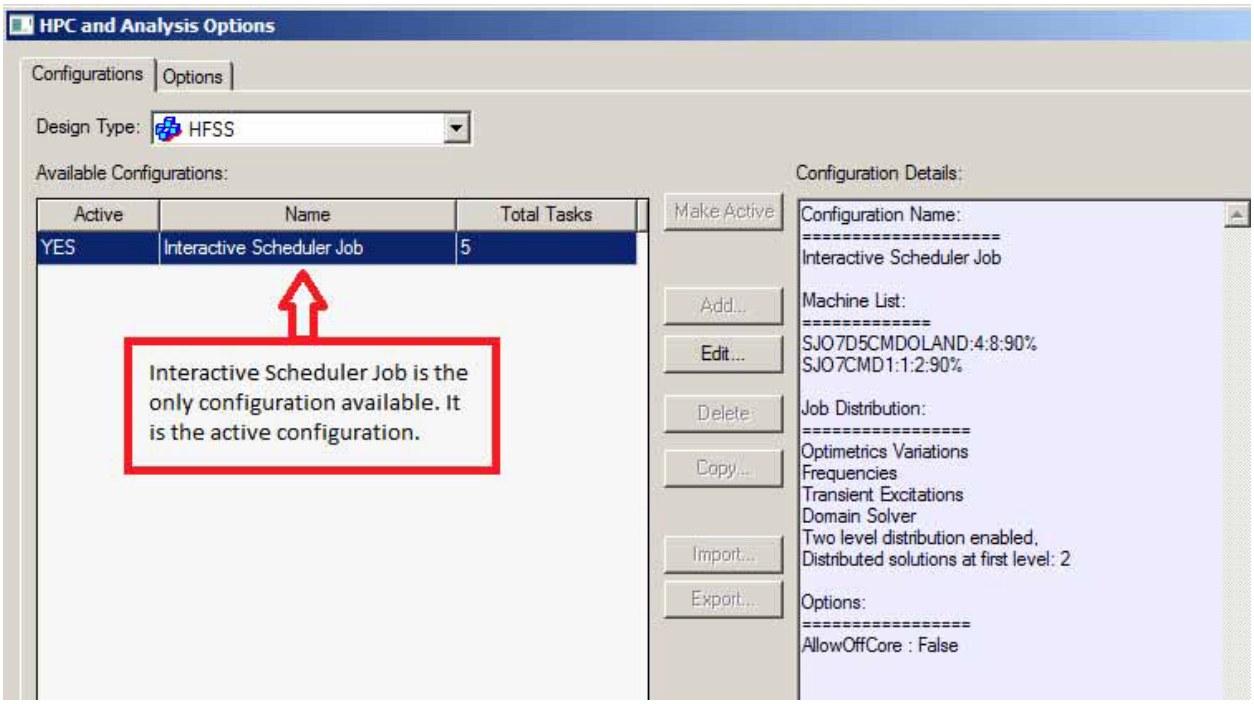
DSO Configuration for Interactive Scheduler Jobs

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

When running an interactive scheduler job, there is only one DSO configuration available for each design type. Each configuration is named "Interactive Scheduler Job". This configuration is always the active configuration for an interactive scheduler job. and it is the only configuration displayed in the list of available configurations shown in the "HPC and Analysis Options" dialog box. No configurations can be added or removed, but the "Interactive Scheduler Job" configuration may be modified using the "Edit" button, which pops up the "Analysis Configuration" dialog.

Because the "Interactive Scheduler Job" configuration is the only configuration accessible for interactive scheduler jobs and it is not accessible in other modes, there is no sharing of the "Interactive Scheduler Job" configuration settings with other modes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



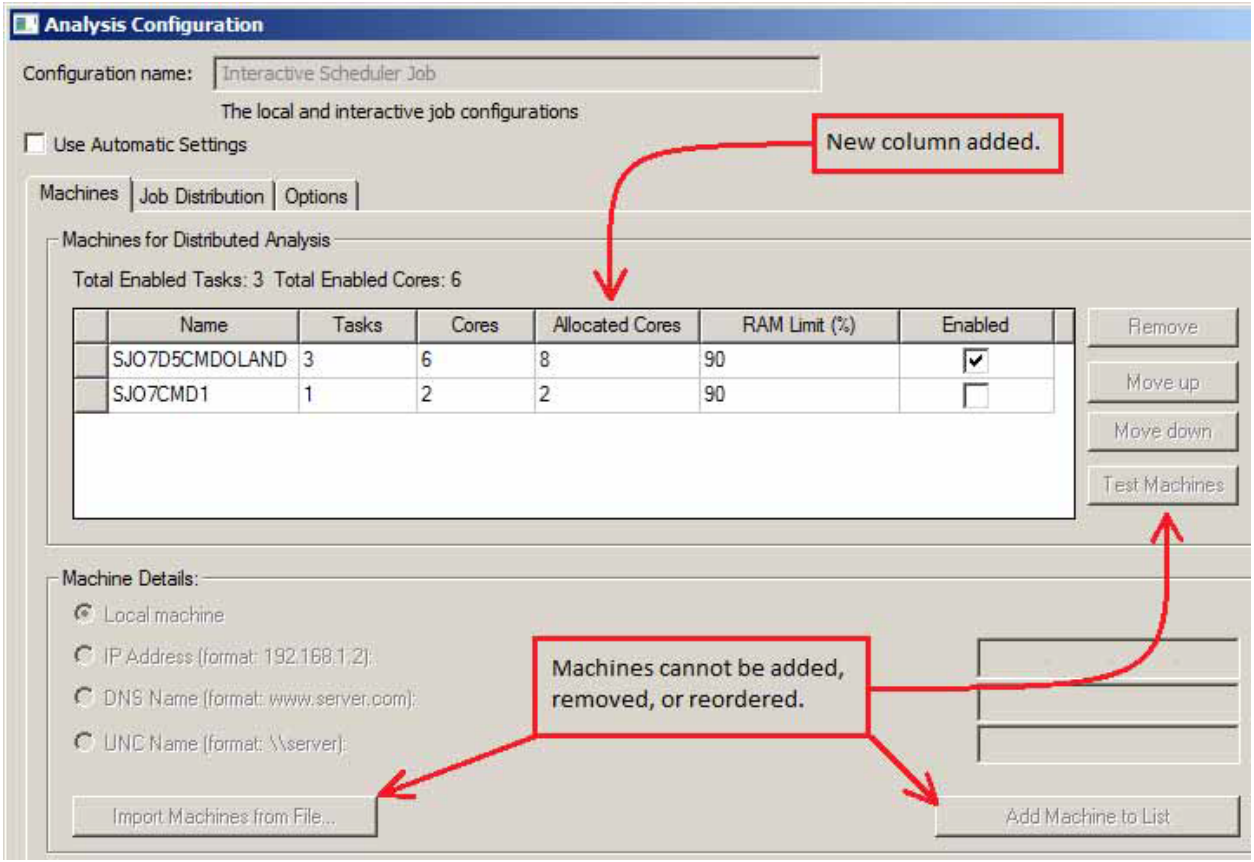
One major difference between interactive scheduler mode and batch mode or normal interactive mode is:

- In interactive scheduler mode, DSO configuration settings and design type options in the UI override command line options and batchoptions.
- When not in interactive scheduler mode, DSO configuration settings and design type options specified on the command line (including batchoptions) override UI settings.

Analysis Configuration - Machines Tab

The most obvious changes for interactive scheduler jobs are visible in the **Machines** tab of the **Analysis Configuration** window. The grid of machine information is prepopulated with the list of machines allocated to the job. It also contains an additional column in interactive scheduler mode. This column, "Allocated Cores", indicates the number of cores allocated to the job by the scheduler on each host. You cannot add or remove machines from the list, or modify the allocated cores for any machines. You can modify the tasks and cores for any machine, or specify that a machine is enabled or disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analysis Configuration - Job Distribution Tab

The **Job Distribution** tab of the **Analysis Configuration** window only appears if the **Use Automatic Settings** check box is not checked. The state of this check box, and the settings shown on this tab are initialized from the command line or from the registry. The command line options that affect the **Use Automatic Settings** check box or the job distribution settings are:

- -distributed
- -local
- -auto
- -machinelist

The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

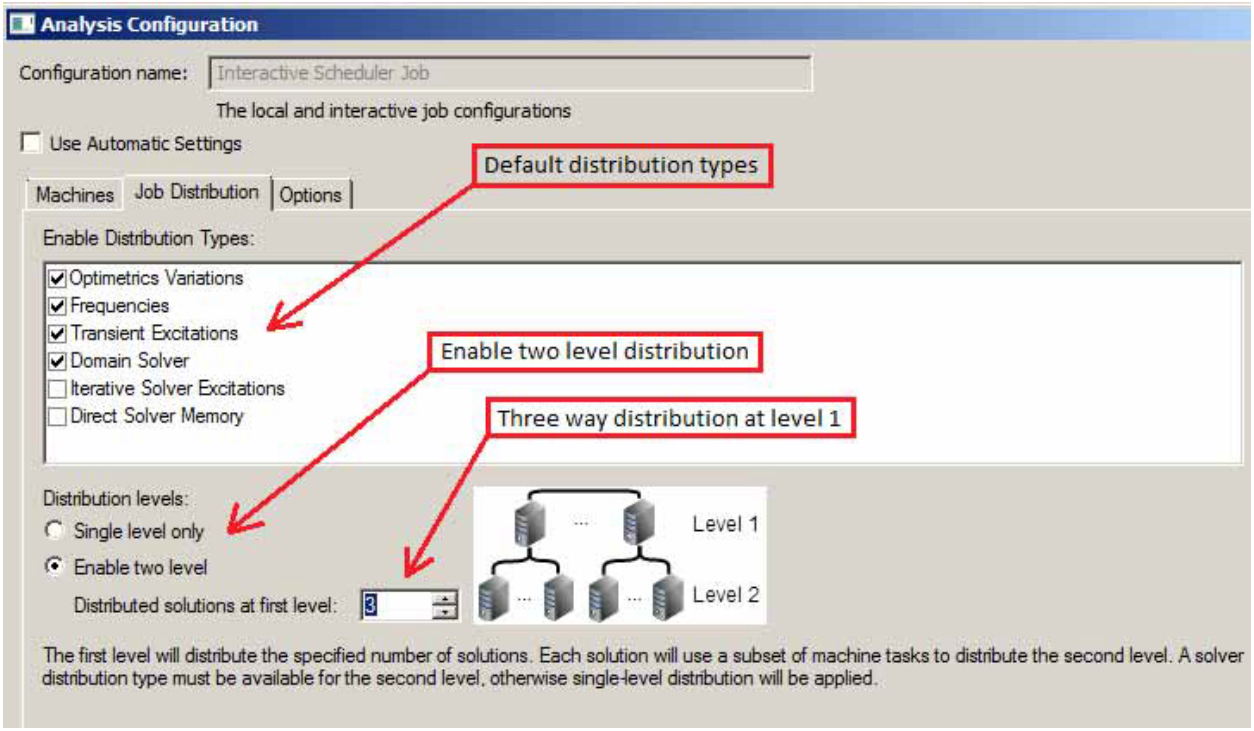
Example

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command line:

```
ansysedt -distributed includetypes=default maxlevels=2 numlevel1=3
```

Initial "Job Distribution" settings:



Analysis Configuration - Options Tab

You can use the **Options** tab of the **Analysis Configuration** window to examine or modify the DSO configuration options. The DSO configuration options are handled like the design type options, except that the Interactive Scheduler Job configuration settings are not shared with other modes.

The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

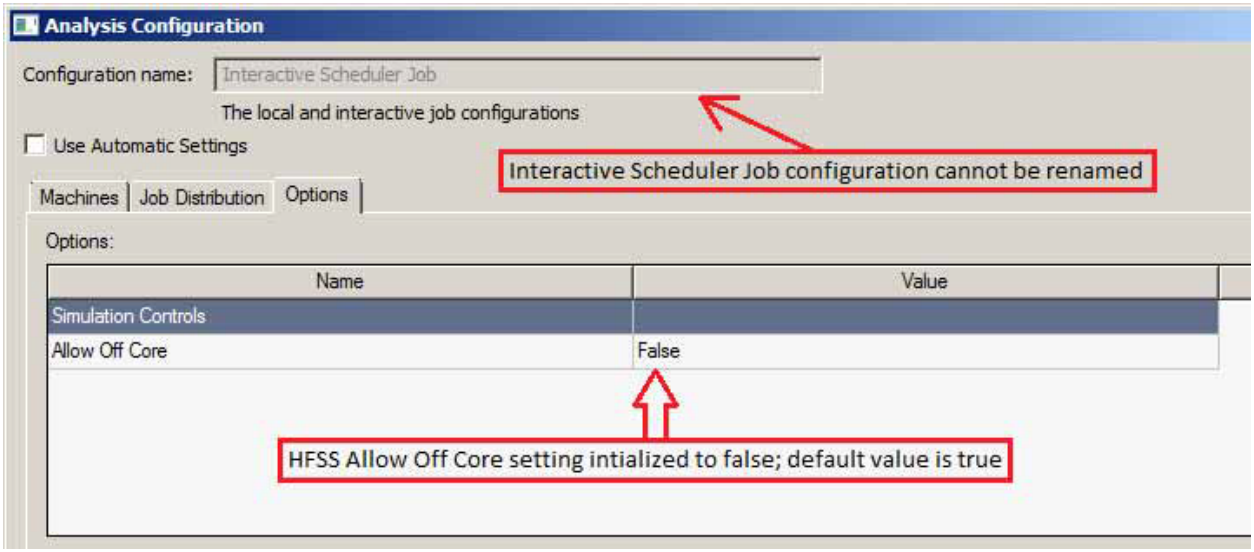
Example

Command line:

```
ansysedt -batchoptions 'HFSS/AllowOffCore'=0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Analysis Configuration Options:



Design Type Options for Interactive Scheduler Jobs

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

You can use the **Options** tab of the **HPC and Analysis Options** window to examine or modify the design type options. The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the current design type.

These settings are shared between interactive scheduler mode and other modes.

Example

The following example includes two command line options, one that is applicable to HFSS ('EnableGPU') and one that is applicable to all design types ('HPCLicenseType'). These opitons would be included as part of the complete job submission command line syntax.

Command Line Excerpt:

```
-batchoptions 'HFSS/EnableGPU'=1 'HPCLicenseType'='Pack'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High Performance Computing (HPC) Integration

Ansys Electromagnetics products offer a direct integration with a number of High Performance Computing (HPC) software programs.

The list of currently supported schedulers includes:

- Ansys Cloud Direct
- Ansys Cloud Burst Compute
- Microsoft Windows HPC Server (Windows Only)
- Grid Engine (GE) (Linux Only)
- Platform's Load Sharing Facility (LSF) (Linux Only)
- PBS Pro or PBS Torque (Linux Only)

Electronics Desktop also supports custom integration.

A job scheduler may also be described as a batch system, a Distributed Resource Management System (DRMS) or Distributed Resource Manager (DRM). The features supported on each scheduler are included in the documents for each. For each job scheduler, the versions or revisions that have been tested are included.

A user may submit jobs using the command line tools or other tools provided by the scheduler. The Desktop includes a GUI to help the user submit jobs to a job scheduler. This generic Job Submission GUI is shared across the Ansys EM products. The general procedure is to specify the scheduler and head node, describe and submit the job, and monitor the results. Using “Ansys Cloud Direct” or “Ansys Cloud Burst Compute will require a login through a browser instead.

Scheduler Terminology

This help uses some specific terminology when discussing HPC and schedulers. See definitions below.

Term	Definition
Core	Unit of processing.
Compute Cluster	Network of machines on which jobs run. Typically consists of head nodes and many compute nodes.
Distributed Processing	Multiple engines are launched simultaneously on the same machine *or* on different machines.
Engine	Electronics Desktop application (aka. executable) launched during analysis commands to generate analysis results.
Job	Application (aka. program, executable) with command line options that uses resources to produce results. For example, `hfss.exe -ng -BatchSolve`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Term	Definition
Machine/Host/Node	Consists of one or more processors, memory, disk, etc.
Multiprocessing	A single engine uses multiple cores on the same machine.
Parallel Job	Job that runs on multiple cores belonging to the same or different machines.
Processor	Consists of one or more cores.
Resource	Machines, licenses, etc. that a job uses.
Serial Job	Job that runs on a single core.
Service	Program that runs in the background (for example, RSM Service) and listens on a port. The OS provides the programming interface through which Applications communicate with services, once the machine and port number are known. Launching an executable on a remote machine requires a service to run on the remote machine.

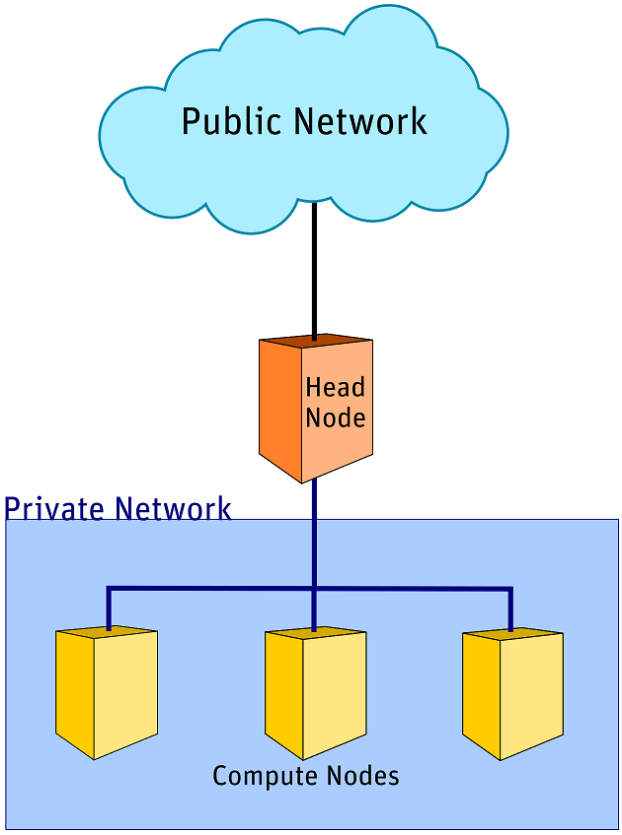
What a Scheduler Does

Schedules are responsible for the following:

- Enabling effective/efficient utilization of cluster's resources consistent with organization's goals
- Maintaining queue(s) of jobs
- Maximizing throughput of jobs by processing all jobs as quickly as possible (typically, one job per CPU)
- Allowing a choice of various scheduling policies (for example, First Come First Serve, Priority-Based, and Preemption)
- Providing a suite of tools or utilities (graphical or command line) for end users to submit, monitor, suspend, and abort jobs
- Managing a compute cluster by running various interacting services on head nodes and compute nodes
- Providing a programming interface to access services

Scheduler-Managed Compute Cluster

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Head node(s) typically maintain queues. Compute nodes are typically on a high speed network to improve scalability of parallel jobs. Services running on nodes interact with each other to manage resources. End user tools communicate with services to manage jobs.

Configuring Electronics Installation for HPC

In HPC applications, Ansys Electromagnetics Suite must be available on each cluster host where jobs may be run.

- On the Linux platform, Ansys EM may be installed on a shared drive that is accessible to all machines in the cluster.
- On the Windows platform, Ansys EM must be installed separately on each host of the cluster.

Ansys EM must be accessible using the same path on each host. All cluster users running Ansys EM jobs must have permission to read and execute the files in the installation directory and its subdirectories.

The temp directory selected during installation must be readable and writable by all user accounts used to run the Ansys Electromagnetics Suite. This temp directory path should be the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

same on all machines of the cluster and should be local to every machine. For example, C:\temp on Windows, /tmp on Linux.

Because HPC is offered as a direct integration, you need only install the Ansys EM software. No additional configuration is required.

Firewall Configuration

If firewall is turned *off* between the machines of the cluster, there is no need for any configuration.

If firewall is turned *on*:

- For a Windows cluster: Configure firewall by adding exceptions that allow Ansys EM programs and services to communicate with each other. If you are using a standard Windows Firewall, this is automatically done for you by the installation program. If you are using a third-party firewall software, it needs to be configured in a similar manner.
- For a Linux cluster: Open up the firewall for range of ports denoting ephemeral (or dynamic) ports. Check with your system administrator on how this can be done on each machine of cluster.

Installation Directory Examples

Microsoft Windows

Install the Ansys Electromagnetics Suite in directory "C:\Program Files\ANSYS Inc\v251\AnsysEM" on each node of the cluster. The same directory pathname must be used on all hosts.

Linux

Install the Ansys Electromagnetics Suite in a common directory that is accessible using the path /opt/ansys_inc/v251/AnsysEM on each execution node of the cluster.

Integration with Microsoft Windows® HPC Scheduler

The Windows HPC scheduler is only supported on Windows.

Jobs may be submitted in any of the following ways:

- Using Windows HPC GUIs from Microsoft: Job Manager or Cluster Manager
- Using Windows HPC command line tools
- Using Ansys Electronics Desktop UI commands for scheduler selection, job submission and job monitoring/control.

General Guidelines for Submitting Ansys EM Jobs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Job submitted to Windows HPC Cluster is defined by Job properties, Task List and Task properties. Priority, resource requirements, node preferences, etc. come from Job properties. In the case of Ansys Electromagnetics jobs, Task List consists of a single task. Properties of this task specify the command line that runs Ansys Electromagnetics desktop in non-graphical mode to perform analysis of a project.

Specifying the Number of Compute Resource Units for HPC Jobs

You can either select **Use automatic settings** on the *Compute Resources* tab, or you either enter the number of tasks and total cores per machine, or individual nodes. HFSS, HFSS 3D Layout, and Icepak have **Use automatic settings** selected by default. This release permits options in setting whether nodes are exclusive for the submission. See Windows HPC Non-exclusive Jobs.

Ansys EM Project File and Project Directory for use with Windows HPC Scheduler

Ansys Electromagnetics Suite tools write their results to a subdirectory of the directory containing the Ansys EM project file. The Project Directory (the directory containing the project file) must be accessible to all of the cluster hosts that may run Ansys EM jobs. The user account for the job must have permission to read the project directory, and to create and modify files and subdirectories of this directory. The pathname of the project file must be accessible to all cluster hosts using the same path name, which is generally expressed as a UNC pathname.

Example:

The project file is on the user's workstation (with hostname user1_PC) in directory C:\user1\projects\new\project1.aedt, and the directory C:\user1\projects is shared with sharename projects.

Correct

When submitting the job, you should use the following pathname to specify the project file:

```
\\user1_PC\projects\new\project1.aedt
```

Incorrect

If a local pathname is used, the cluster hosts will not be able to find the user's project on the workstation

```
user1_PC: ' C:\user1\projects\new\project1.aedt '
```

Submitting and Monitoring Ansys EM HPC Jobs

Jobs may be submitted to the Windows HPC Scheduler using any of the following methods:

- Using the *Submit HPC Job* dialog box
- Using the Windows HPC Job Manager GUI

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the Windows HPC Command Line Tools
- Using the Windows PowerShell

Client Utilities from the Microsoft HPC Pack, must be installed on the submit host to use any of these methods to submit a job to a cluster. The *Submit HPC Job* dialog box will be unable to contact the cluster head node if the client utilities are not installed.

This document covers the first method. See the Microsoft documentation for information on the other three methods.

- Submitting and Monitoring Jobs for Windows HPC
- Specifying the Number of Compute Resource Units for HPC Jobs

Jobs may be submitted from any Microsoft Windows host meeting the following requirements:

- For submitting jobs to the Windows HPC scheduler, the Desktop process must run on a node that is configured for submission of jobs to the Windows HPC cluster. That is, the Windows HPC Client Utilities must be installed on the node, and network communication from the Desktop node to the head node of the cluster must be allowed. For Ansys Electromagnetics Suite 2025 R1, Windows HPC Server 2008 R2 (or later) client utilities are required. Using a computer on the network is not supported for submission of jobs to the Windows HPC cluster.
- When submitting jobs to a Windows HPC cluster, the user must also specify the head node of the cluster to which the jobs will be submitted. When the user selects the "Windows HPC" scheduler in the "Choose scheduler" list, the Head Node edit control is enabled. The user may enter the Windows HPC cluster head node name into the edit box. Alternatively, the head node may be selected using a "Browse for Computer" browser by pressing the ellipsis [...] button.
- The Windows HPC Pack client utilities are installed on the submission host.
- Network communication between the submission host and the Windows HPC Cluster head node is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.
- The submission user is permitted to submit jobs to the Windows HPC Cluster.

Job Monitoring

1. Windows HPC Jobs may be monitored using the *Monitor Job* dialog box, which is brought up by the **Tools > Job Management > Monitor Jobs...** command. This dialog bos may also be brought up by checking the **Begin monitoring this job now** check box when a job is successfully submitting using the job submission dialog box. You can also select the **Simulation** tab of the ribbon and select the **Monitor** icon. You can monitor this job either automatically (by checking the option) or through the **Tools> Job Management> Monitor Jobs...** command. For more details, see Monitor Jobs window.

In addition to the above requirements to allow job monitoring the following is also necessary:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Network communication between the submission host and all Windows HPC Cluster nodes where the job may run is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.

Cluster Configuration

Any job running on a Windows HPC Cluster that is distributed over multiple compute hosts requires network communication between processes running on these hosts. The cluster must be configured to allow this communication. Any firewall or other security software must be disabled or configured to allow communication between any of the compute hosts were a job could run.

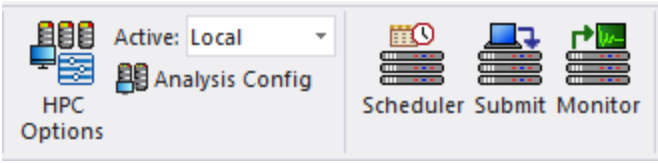
Job Submission User Profile on Cluster Compute Nodes

In order for a job to run correctly, the submission user's profile must be accessible and properly initialized on the cluster compute nodes where the job runs. If the Ansoft/temp subdirectory of the user's "My Documents" directory does not exist or is not accessible on the compute cluster nodes where a job runs, the batchoptions for the job will not be processed correctly, resulting in job failure. One way to ensure that this directory is created on each compute host is for the submission user to login to each compute host and run the product GUI one time.

Submitting and Monitoring Jobs for Windows HPC

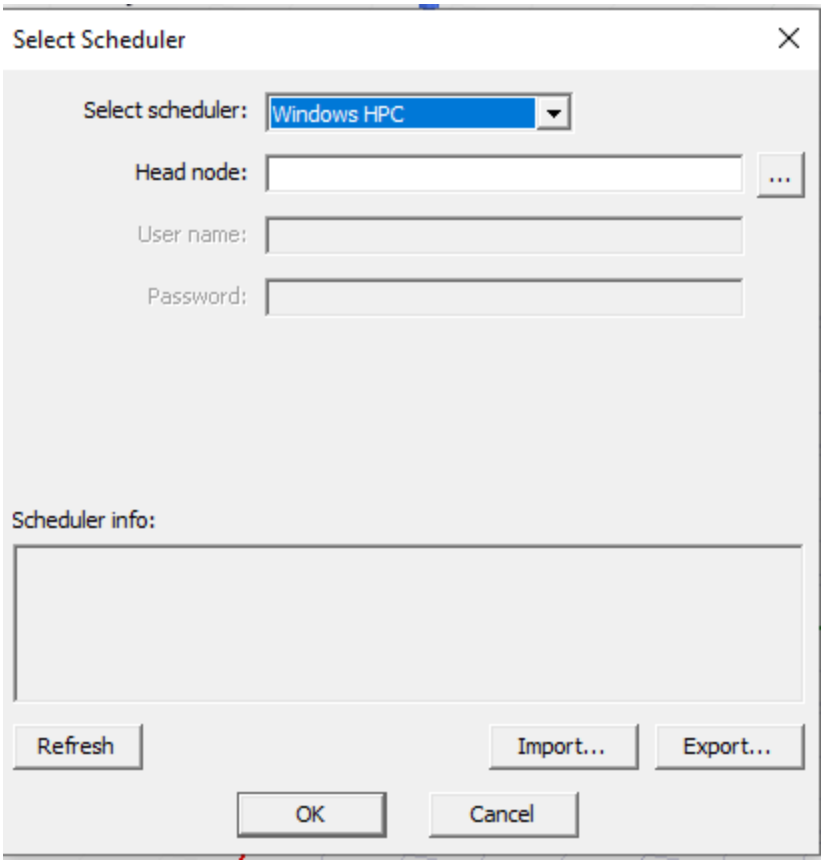
In order to submit jobs using **Windows HPC**:

- Click **Tools** > **Job Management** > **Select Scheduler**, or
- Select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



This opens the **Select Scheduler** dialog box. Specify **Windows HPC** as the scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Windows HPC, specify the head node of the cluster.

After specifying the cluster head node, click **Refresh**. This verifies that the head node may be contacted, and displays the scheduler name, a brief description (including the head node name), and the version of the Windows HPC head node.

Pressing **Cancel** discards changes made in this dialog box. Pressing **OK** verifies that the head node can be contacted before accepting the changes. If no problem occurs, the dialog box will close. If there is a problem contacting the head node, the dialog box will not close and the changes will not be accepted.

After setting the job submission node, perform *one* of the following:

- Select **Tools** > **Job Management** > **Submit Job...**
- Click **Project** > **Submit Job...**
- Click **Mechanical** > **Submit Job...**
- Select the **Simulation** tab and click the **Submit** icon.
- Use a shortcut menu to select **Submit Job**.

This opens the **Submit Job To** dialog box.

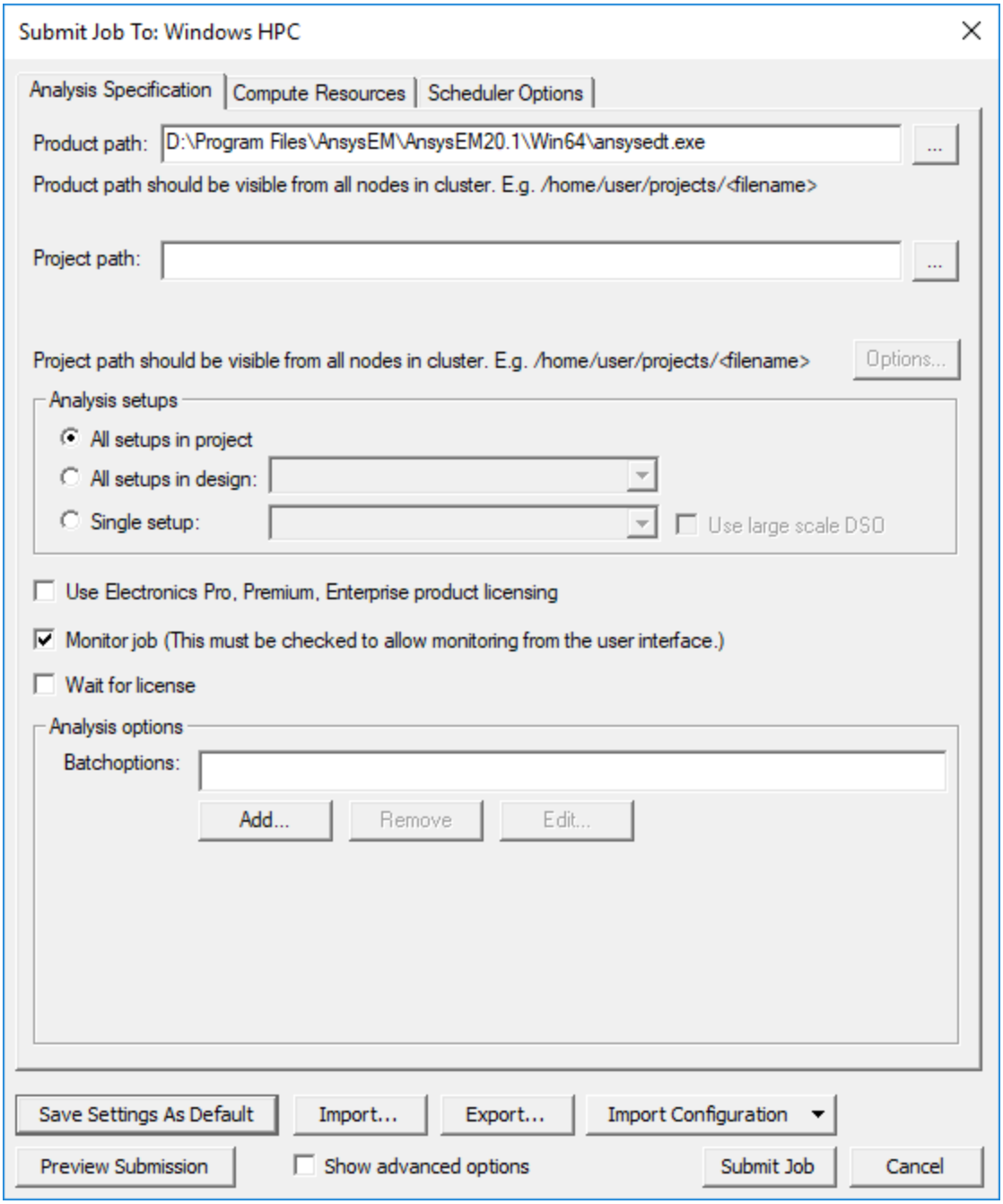
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Submit Job To** dialog box contains three tabs:

- **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
- **Compute Resources**– this tab can be populated either by automatic settings, by predefined Analysis Configuration, or specifying parameters in the fields for resource selection, for job parallelization and enabled forms of parallelization.
- **Scheduler Options**– contains fields for Job name and priority. The customization options shown by checking advanced are not used for Windows HPC.

On the **Analysis Specification** tab, enter path names for the Product and Project. These must be UNC paths that are accessible from each compute host used for Ansys Electromagnetics jobs. The Project can be an archive. The submission user must have permission to write to the directory containing the project file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can select which setups are analyzed in the **Analysis Setups** area of the dialog box. There are radio buttons to select:

- All setups in the project
- All setups in a specified design (you select the design from the drop-down menu)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

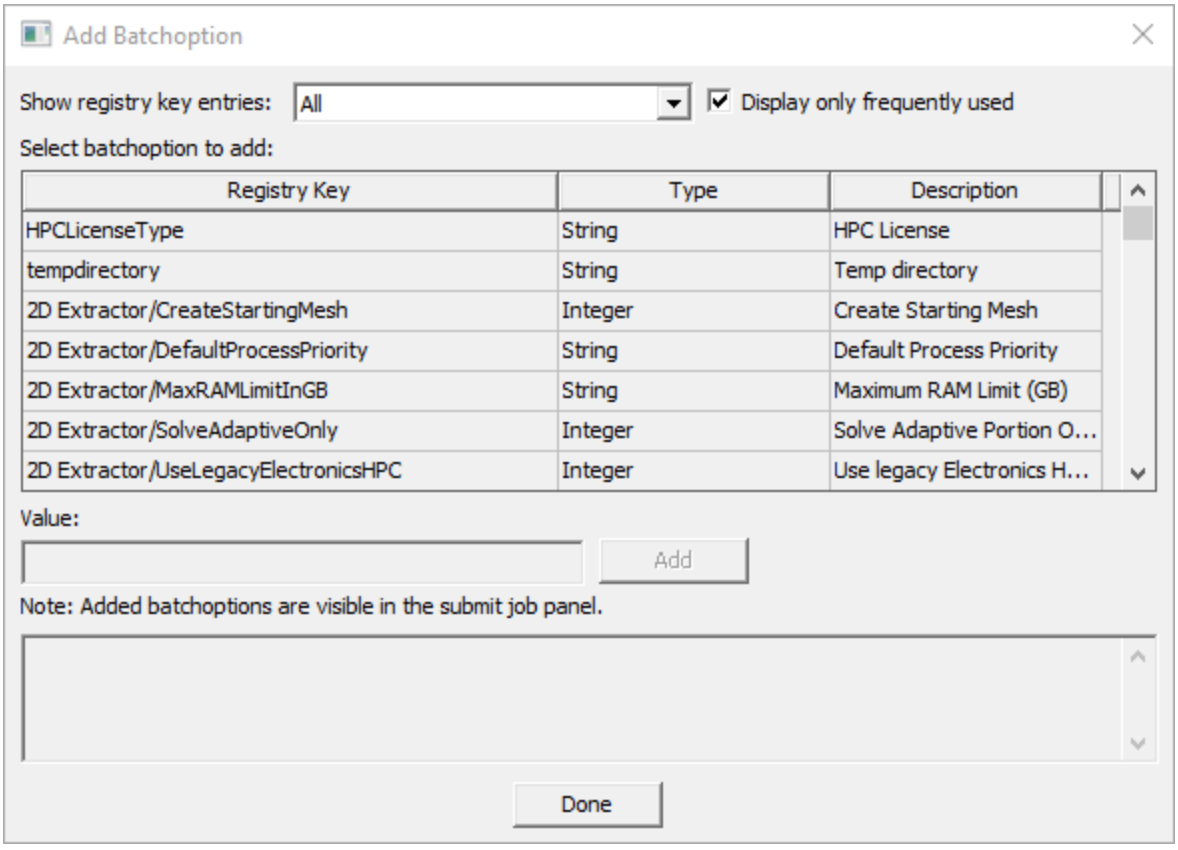
- Single setup (if you selected the Submit Job command from the shortcut menu, the setup name populates the field)

If you specify multiple setups, they will be processed sequentially in the order displayed in the edit box.

The Analysis options include:

- Monitor job – you must enable this option to monitor the job from the user interface.
- Wait for license – specify whether to wait until a license if available before starting a simulation.
- Batch options – optionally specify -batchoptions in the text field. See detailed discussion of -batchoptions beginning under Running Ansys Electronics Desktop from a Command Line.

The **Add...** button opens the **Add Batchoption** dialog box.



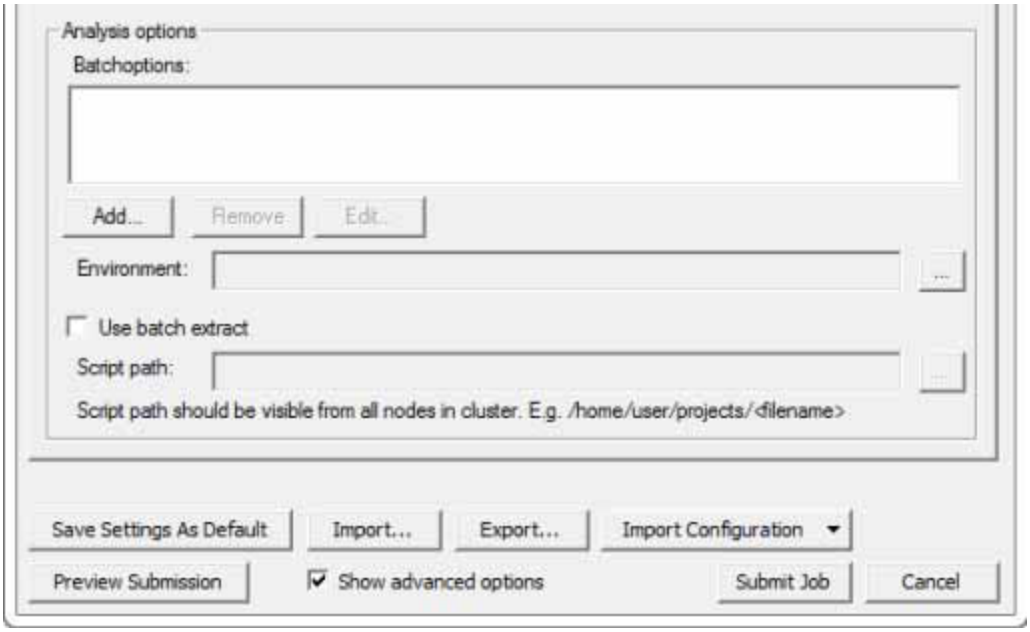
This dialog box provides access to all -batchoption commands. The drop-down menu lets you select specific categories, and you can choose to display only frequently used commands. You can edit and remove any batch options you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

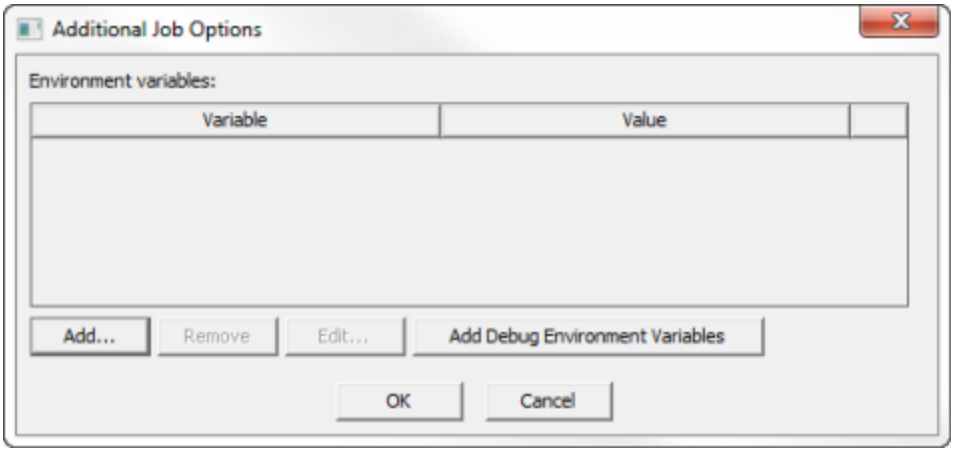
Select a Registry Key in order to show the current Value for the type. The lower field explains the meaning of the Type Value.

Any batchoptions for which you select **Add** will be visible in the **Submit Job** dialog box.

If you have the **Show advanced options** box checked in the **Submit Job** dialog box, the Environment field and the Use batch extract fields display.

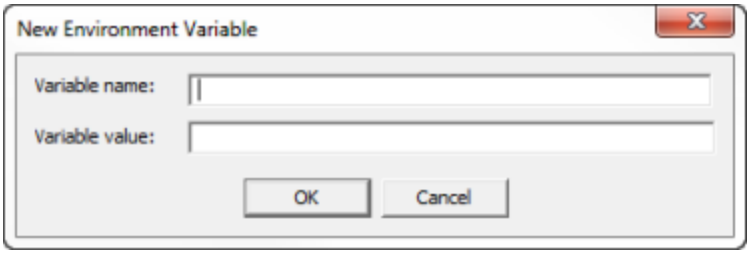


The **Environment** field lets you specify any environment variables. Click the ellipsis button [...] to display the **Additional Job Options** dialog box.



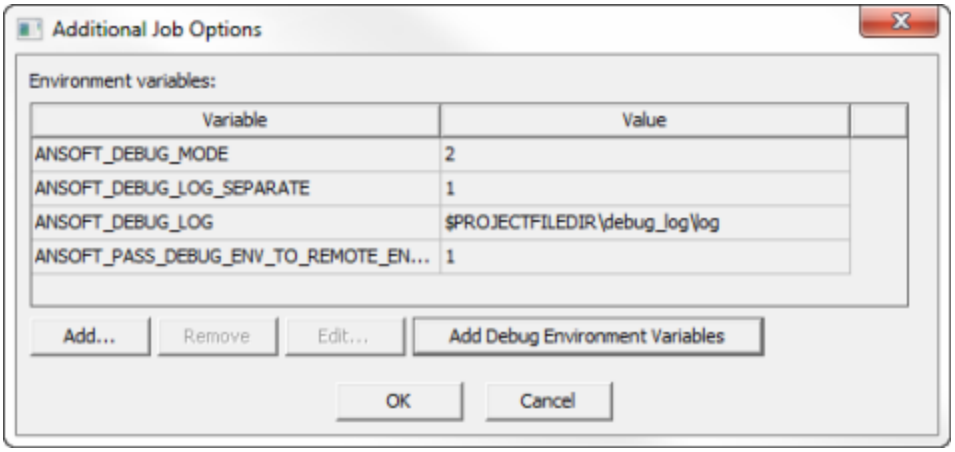
Click the **Add...** button to open the **New Environment Variable** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you can provide a Variable name and Variable value. Click **OK** to display the variable in the **Additional Job Options** dialog box.

Select a variable to enable the **Remove** and **Edit...** buttons. You can also click **Add Debug Environment Variables**.



> **Note:**
>
> For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:
>
> ANSYSEM_SKIP_NODE_ONLINE_CHECK
>
> Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted

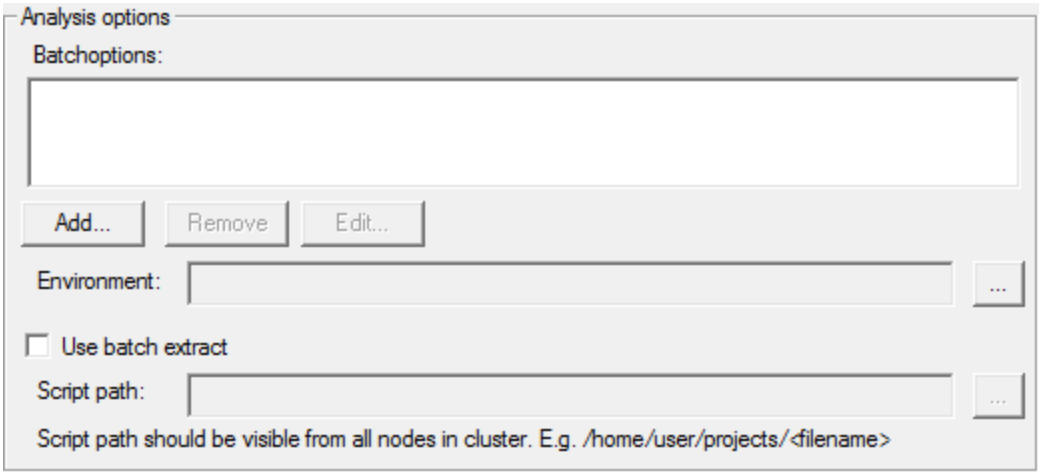
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (if not set, Ansys Electronics Desktop requires nodes to be online for submission).

If you have enabled **Show Advanced Option**s, any variables that you add will be displayed in the **Environment** field of the **Submit Job** dialog box.

Use Batch Extract for Windows HPC

Enabling **Show Advanced Options** for Windows HPC also displays the **Use batch extract** check box and script path field.

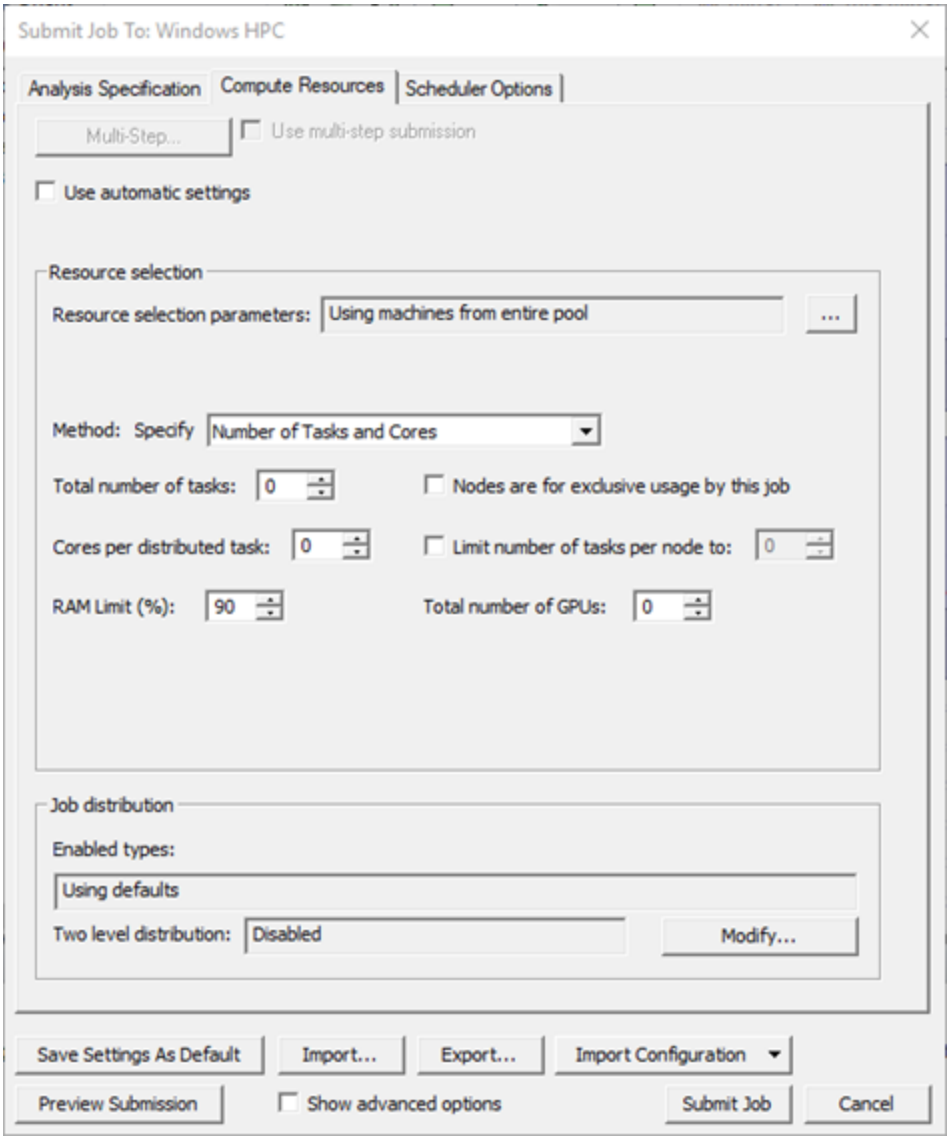


See Running Ansys Electronics Desktop from a Command line for an explanation of the solve information available through batch extract.

The **Preview Submission** button opens a window that shows the text commands that will be sent to the scheduler.

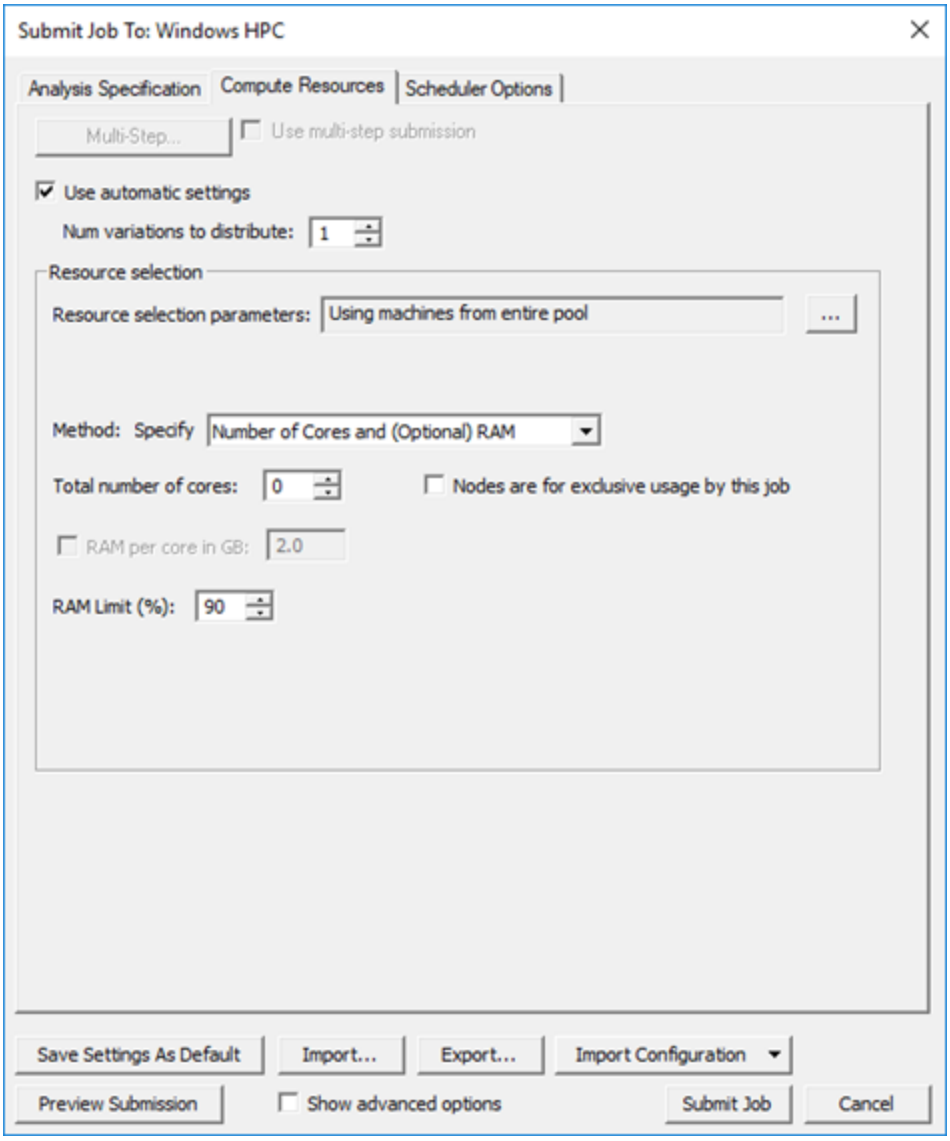
The following figure shows the **Compute Resources** tab of the **Submit Job To** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Ansys Electronics Desktop configurations, the **Submit Job To** dialog box includes a **Use automatic settings** check box that simplifies the **Compute Resources** tab.

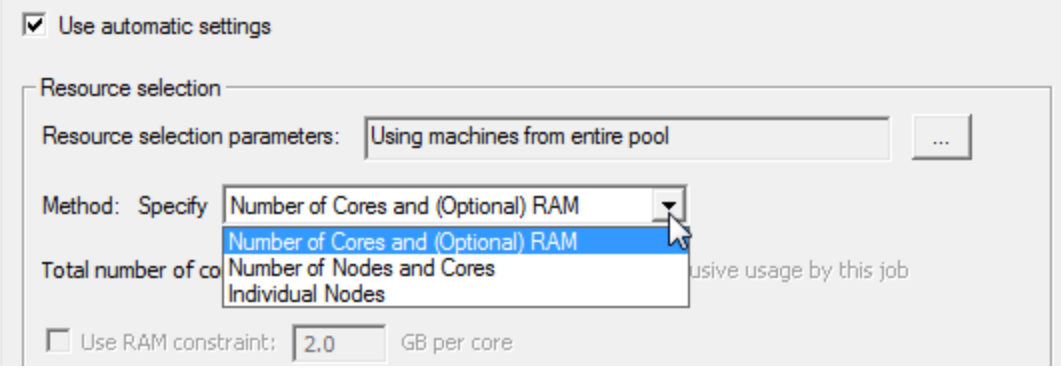
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution** field is removed. When using automatic settings, you can specify **Resource selection** parameters. The ellipsis button [...] opens the Compute Resource Selection Parameters dialog box. If you do not specify any parameters, the default is "Using machines from entire pool."

The Method field of the **Submit Job To** dialog box has a drop-down menu with two or three selections, depending on whether you select **Use automatic settings**.

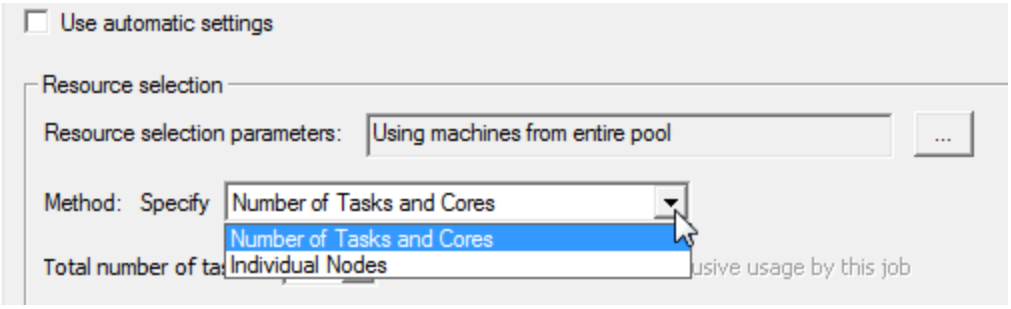
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

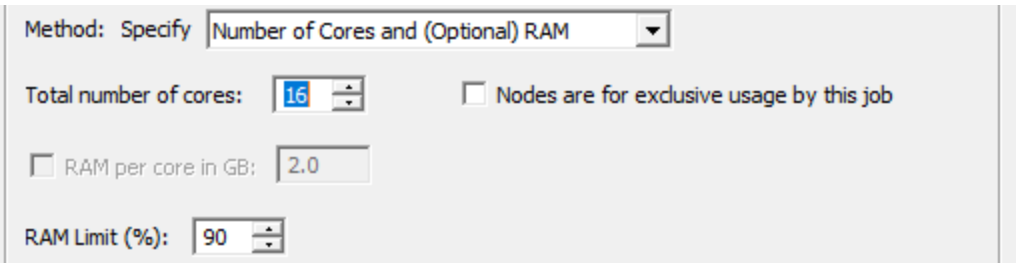
If you select Use automatic settings, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains.

If you uncheck or cannot access Use automatic settings, these two Methods are listed:



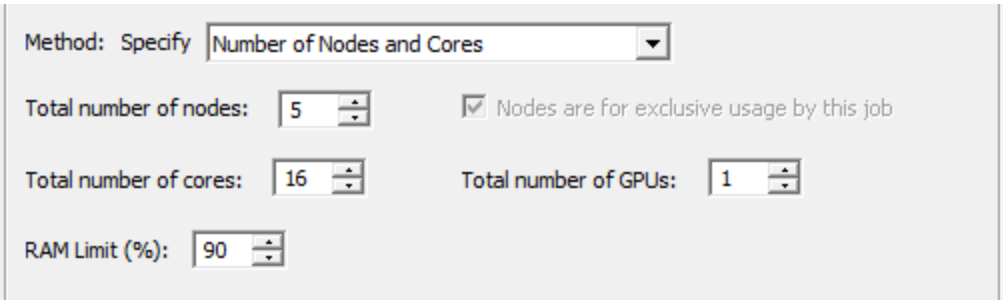
For Windows, there is a 64 core limit for a single machine. Each Method selection changes the available options listed:

- Specify Number of Cores and (Optional) RAM

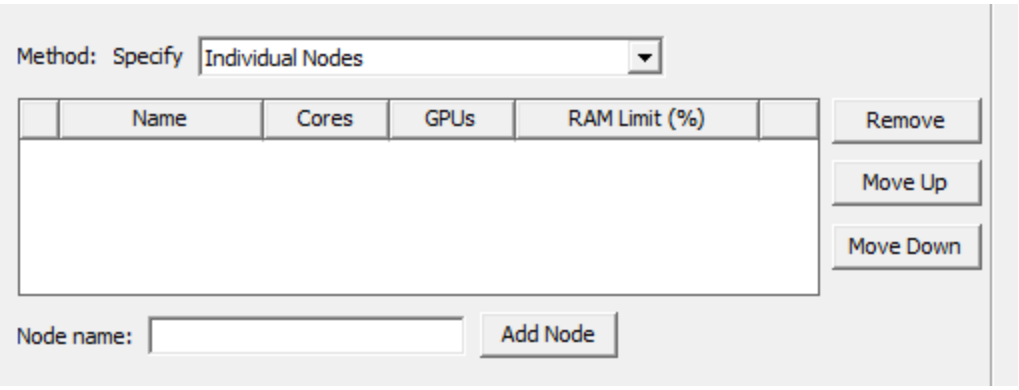


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Nodes and Cores

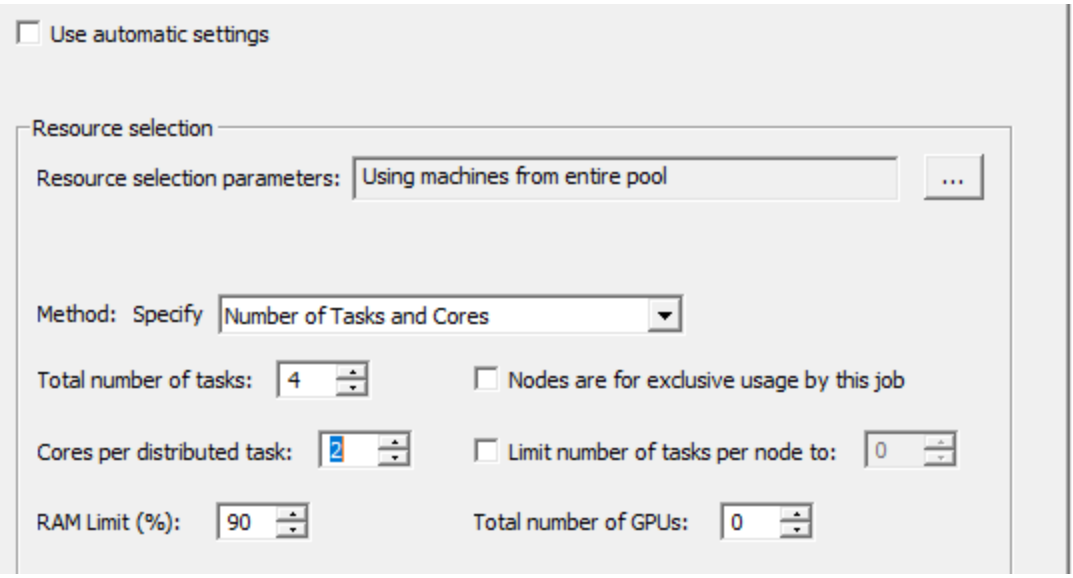


- Individual Nodes



- Number of Tasks and Cores ("Use automatic settings" is unchecked for this option. Checking "Use automatic settings" means that you do not have to specify tasks or core parameters):

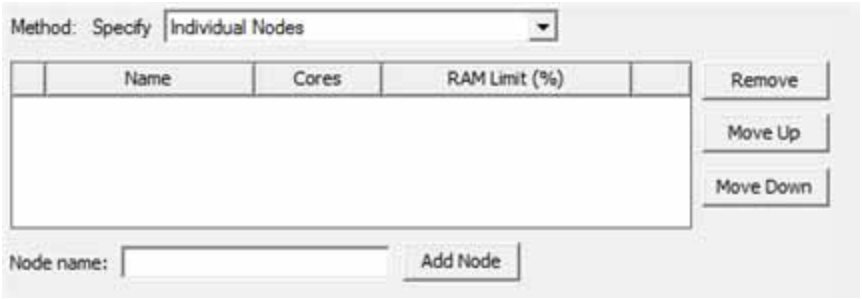
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual Node List

For Windows HPC jobs, you may either specify a node list, or specify job parallelization parameters, but not both.

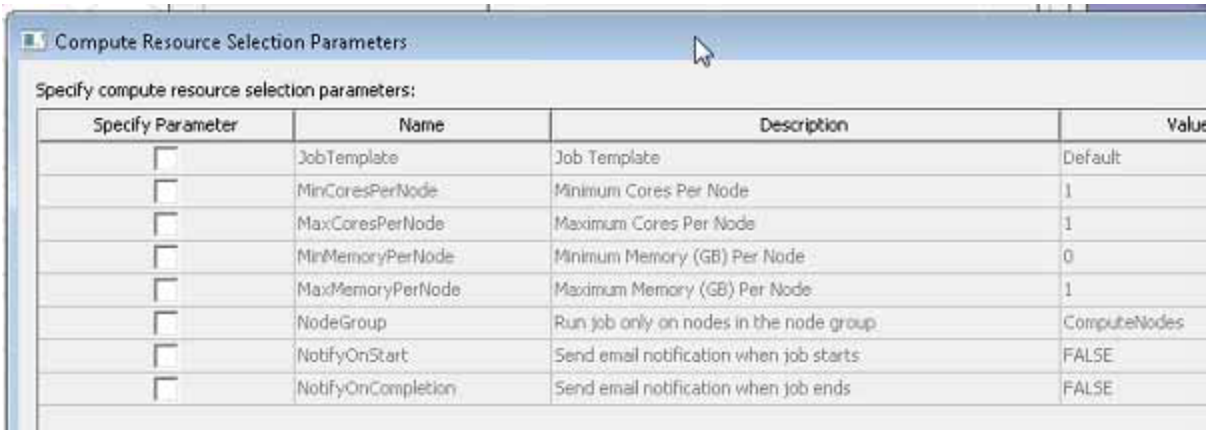
If you select the Individual Nodes Method, you may specify a node list, and the Job parallelization controls are disabled. In this case, the node list should only include cluster nodes that are valid for the job. For each node, you enter the node name and add the node. In the table, you can specify the number of cores and the RAM limit as a percentage. You can use the Remove, Move Up and Move Down buttons to edit and order the list.



Compute Resource Selection Dialog

By default, you can draw from the entire pool. You can also click the ellipsis button [...] to open a **Compute Resource Selection** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resource selection parameters for Windows HPC jobs are:

- JobTemplate: Job Template - The JobTemplate may limit the job parameters or specify defaults values for job parameters
- MinCoresPerNode: Minimum Cores Per Node
- MaxCoresPerNode: Maximum Cores Per Node
- MinMemoryPerNode: Minimum Memory (GB) Per Node
- MaxMemoryPerNode: Maximum Memory (GB) Per Node
- NodeGroup: Run job only on nodes in the node group
- NotifyOnStart: If True, send email notification when job starts. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)
- NotifyOnCompletion: If True, send email notification when job ends. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)

Job Parallelization

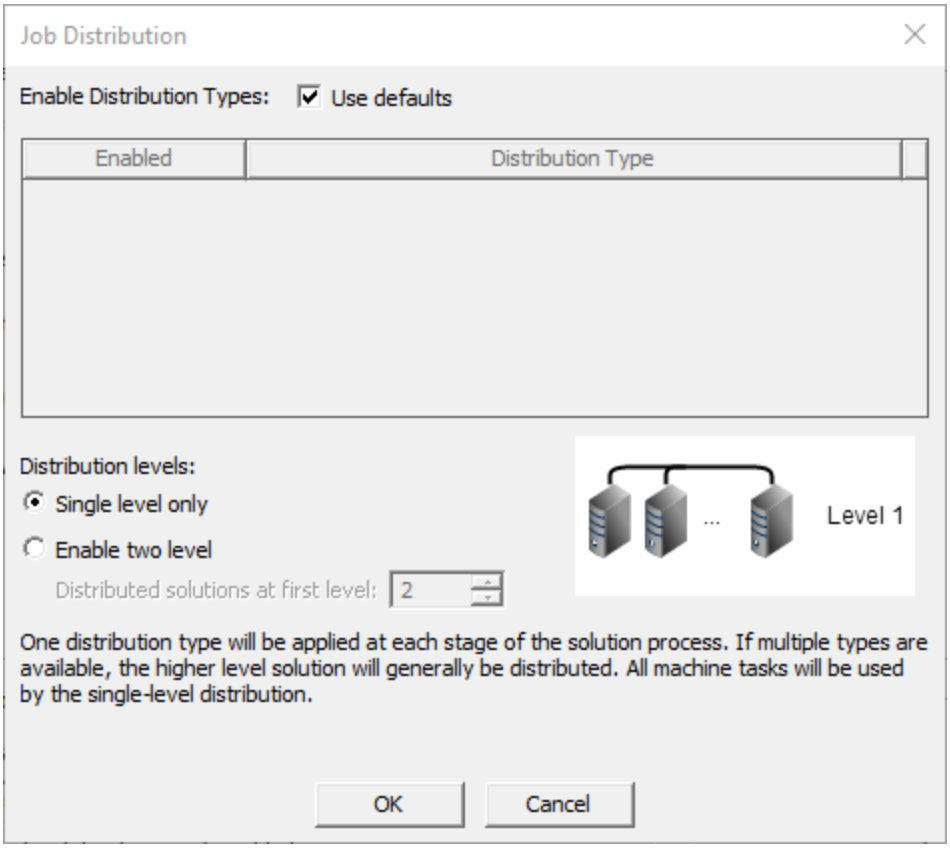
For Windows HPC jobs, you may either specify a node list, or specify the job parallelization parameters, but not both. The Job parallelization fields let you specify:

- Total number of tasks: The number of nodes requested for the job is the total number of tasks divided by limit on the number of tasks per node, rounded up if it is not an integer.
- Cores per distributed task. This determines the amount of multiprocessing per task.
- Whether nodes are for exclusive usage by this job
- Whether to limit the number of tasks per node to a value. If the "Limit number of tasks per node" check box is not checked, then the job is submitted with a job unit type of "Core".

Job Distribution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

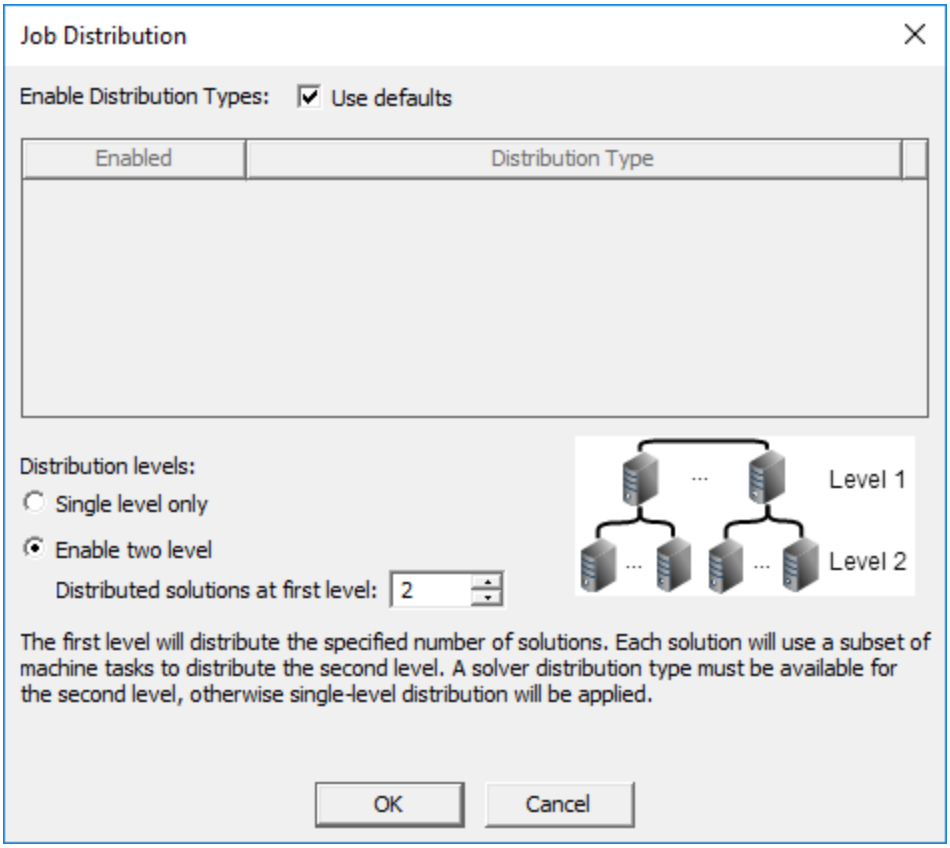
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the **Job Distribution** dialog box and select the **Enable two level** option if applicable and desired.



> **Note:**
>
> Two-level distribution is not currently supported for Mechanical designs. Keep the default **Single level only** option for *Mechanical – Modal* and *Mechanical – Thermal* solutions.

- Second level distribution operates within DSO. If available and enabled you can specify the number distributed solutions for level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

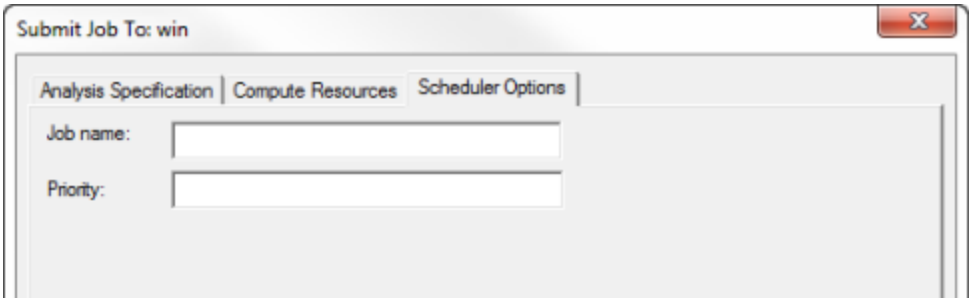


In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

Scheduler Options

The **Scheduler Options** tab provides for specifying the job name and/or the job priority. While the **Show advanced options** check box enables the display of Job submission options, no job submission options should be specified for Windows HPC.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Preview Submission

The Preview Submission button opens a window that shows a text description of the job to be submitted and the task used to start the product on one of the nodes.

The JOB PARAMETERS section contains information on parameter that apply to the job as a whole.

- The "Job resource parameters" section indicates whether the job has exclusive use of nodes, the job unit type, and the minimum and maximum number of units requested for the job, node group, and email notifications.
- The "Job attributes" section displays the job name and job priority.
- The "User Specified Compute Resource Attributes" displays the Resource selection settings.

The TASK PARAMETERS section contains information on parameters that apply to the Desktop task, which is the main task of the job.

- The "Desktop task resource parameters" section indicates the job unit type (which is the same as in the JOB PARAMETERS), and the minimum and maximum number of units requested for the Desktop task.
- The "Command Line section" displays the desktop task command line, including all arguments.
- The "Environment variables" section displays the environment variables that are set for the Desktop task; the same environment variables will also apply to all other tasks of the job.
- The "Working directory" section indicates the working directory in which the Desktop task will run.

Monitor Job

If you have checked the Monitor Job option on the **Submit Job To** dialog box, **AnalysisSpecification** tab, you can invoke the **Monitor Job** window by clicking **Tools** > **Job Management** > **Monitor Jobs...** This dialog box may also be brought up by checking the **Begin**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

monitoring this job now check box when a job is successfully submitting using the job submission dialog box. For more details, see: Monitor Jobs Window.

Windows® HPC Job Templates

The job templates are managed by the Windows HPC cluster administrator. Every cluster has at least one job template, the "Default" job template. Every job has an associated job template. If no job template is specified, then the "Default" job template is used. The job template controls two related aspects of the job submission process. When a job is submitted, there are a number of job parameters which may be specified. Each parameter has a set of valid values. For example, the Priority parameter has five valid values, Highest, AboveNormal, Normal, BelowNormal, and Lowest. The job template controls the default value of each parameter; this is the value that the parameter has if it is not specifically overridden by the submitter. For example, in the Default job template, the default value of the Priority parameter is Normal. The job template may also limit the allowed values of each parameter to a subset of the valid values. For example, a job template for privileged users could allow all five Priority values, which a job template for unprivileged users could limit the allowed Priority values to Normal, BelowNormal and Lowest.

Each job template is a Windows object with access controlled by an ACL (access control list). Instead of the usual "Read" or "Read & Execute" permissions, there is a "Submit Job" permission which corresponds to the right to submit a job with this job template. The cluster administrator may create job templates to limit or control access to cluster resources. For example, a job template with limited allowed job run times, or access to a limited set of compute nodes could be created by the cluster administrator. Specific users or user groups could be forced to use this limited job template by omitting access to the other job templates or by adding a deny access entry for the specified user or group to the other job templates. See the *Microsoft HPC Pack 2019: Job Templates* white paper from Microsoft for additional details:

https://learn.microsoft.com/en-us/powershell/high-performance-computing/job-templates?view=hpc19-ps

Job templates may also be created to allow users to run jobs with limited knowledge of the appropriate job parameters. The cluster administrator creates a job template which has reasonable default values for the type of job to be run, and informs users which job template to use for each type of job. The template could also limit some parameters to only the subset of all values that are useful for the type of job associated with the template.

Selecting Computation Resource Units (Job Unit Type)

The Job Unit Type is the smallest unit of processing resources used to schedule the job. This is one of the most important job properties. There are three options for the Job Unit Type: cores, nodes or sockets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Cores**: Jobs are scheduled in units of cores, which may be also described as a CPU cores, logical processors, or CPUs. This is the smallest unit of granularity available. This selection allows the scheduler to start multiple tasks on a processor, if the total number of cores needed by the tasks is less than or equal to the number of cores on the processor. This selection may also allow the scheduler to distribute more of the computational load to processors with more cores than to processors with fewer cores. For Windows, there is a 64 core limit for a single machine.
- **Nodes**: Jobs are scheduled in units of nodes, hosts or machines. This is the coarsest level of granularity that may be selected. When this option is selected, only one task will run on any give node at any given time. This is useful in cases where it is not desirable to run multiple tasks on a single host. For example, if each task is multi-threaded, running multiple tasks on the same node may not be needed to fully utilize the computing resources on the node. This may also be preferred if the tasks are memory intensive, and multiple tasks would be competing for the limited memory resources.
- **Sockets**: A socket (which may also be called a NUMA node) is a collection of cores sharing a direct connection to memory. A socket will contain at least one core, and it may contain several cores. The socket concept may not necessarily correspond to a physical socket. Scheduling at the socket level may be useful in cases in which each task requires extensive use of the memory bus, and scheduling multiple tasks on the same socket would result in excessive bus contention.

Windows® HPC Non-Exclusive Jobs

Windows HPC allows three choices for the Cluster AffinityType setting:

- No Jobs: Affinity is not set for any jobs
- Non Exclusive Jobs (default): Affinity is not set for exclusive jobs
- All Jobs: Affinity is set for all jobs

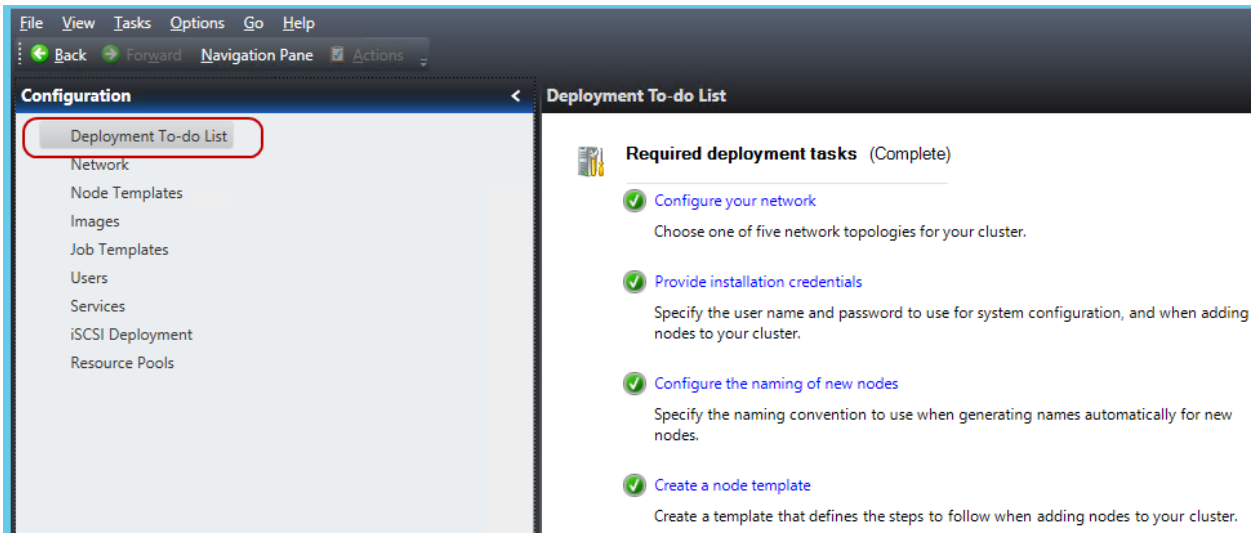
If the AffinityType is set to “No Jobs,” then we can allow non-exclusive jobs in many cases, but non-exclusive jobs will be disallowed if you specify an explicit host list (because all resources on all specified hosts are used), if you specify nodes and cores (to ensure that the job has exclusive access to the specified number of nodes), or if you specify multi-threading (to prevent a single task from being split across hosts). If the AffinityType is set to “Non Exclusive Jobs,” then exclusive jobs will be disallowed for all cases, the same as the restrictions for the previous release.

The previous job submission dialog always has the “exclusive” check box disabled for Windows HPC jobs, so that “exclusive” jobs cannot be submitted to a Windows HPC cluster. The “exclusive” check box is now enabled for Windows HPC jobs. This allows you to submit exclusive jobs to a Windows HPC cluster. If this check box is checked, but the submission parameters and Windows HPC cluster AffinityType do not allow submission of an exclusive job, then an error message is displayed when you submit or preview the job.

Setting the Windows HPC Cluster Affinity Type

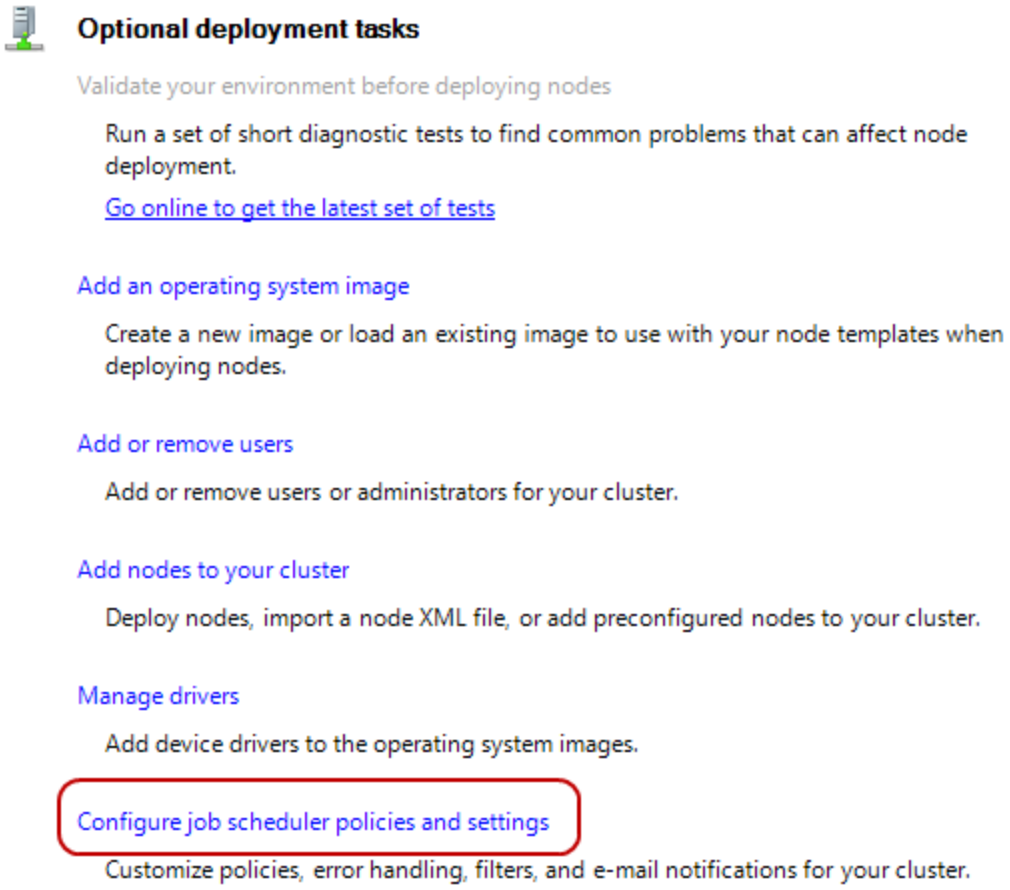
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Windows HPC Cluster administrator is able to modify the Affinity Type setting for the cluster. The administrator should start the Windows HPC Cluster Manager. The first step is to select the Configuration button on the lower left pane, then select the Deployment To-do List in the upper left pane. Then, the upper right pane displays the Deployment To-do List, as shown below.



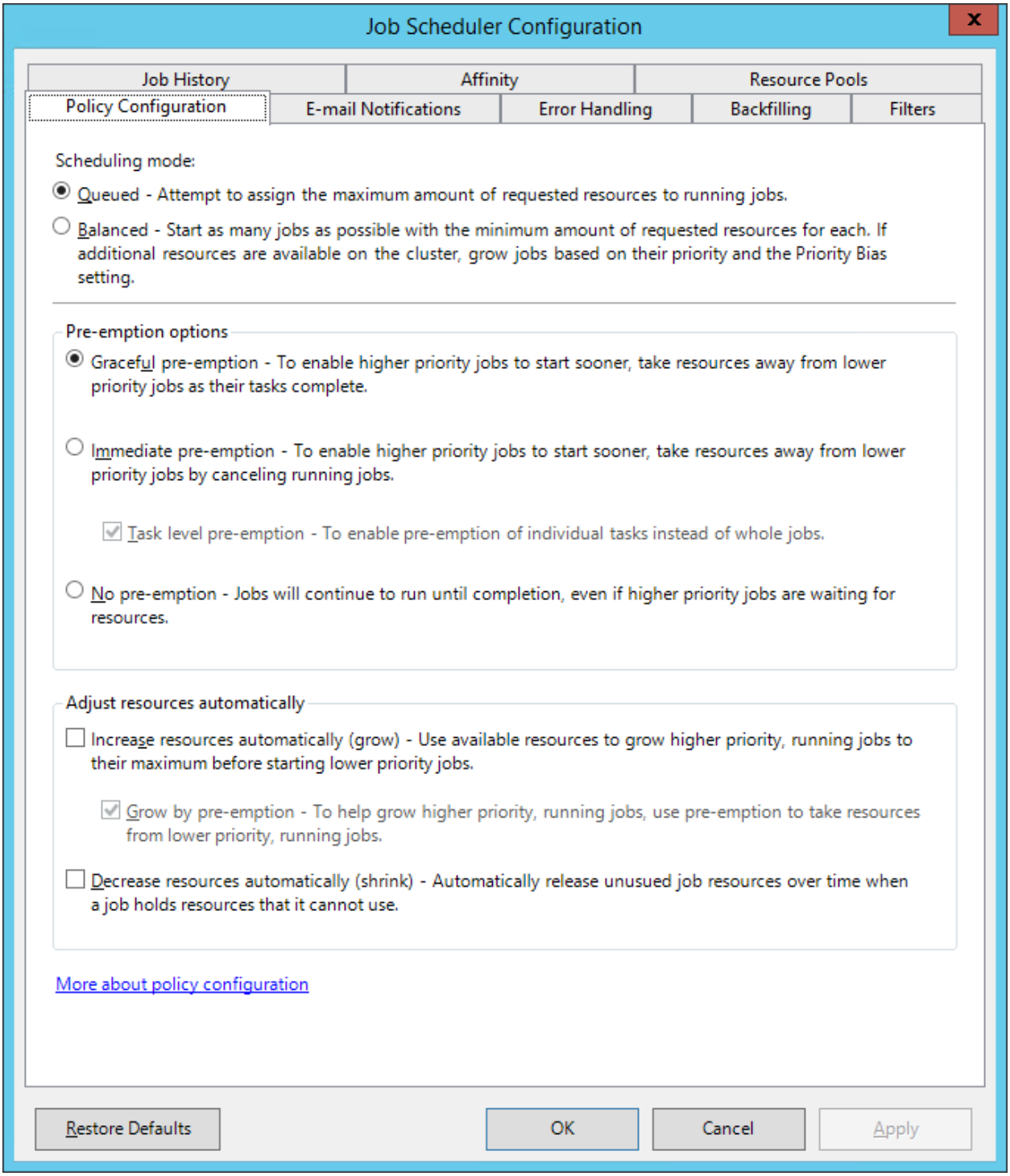
Next, the "Configure job scheduler policies and settings" should be selected. This is under the heading "Optional deployment tasks."

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



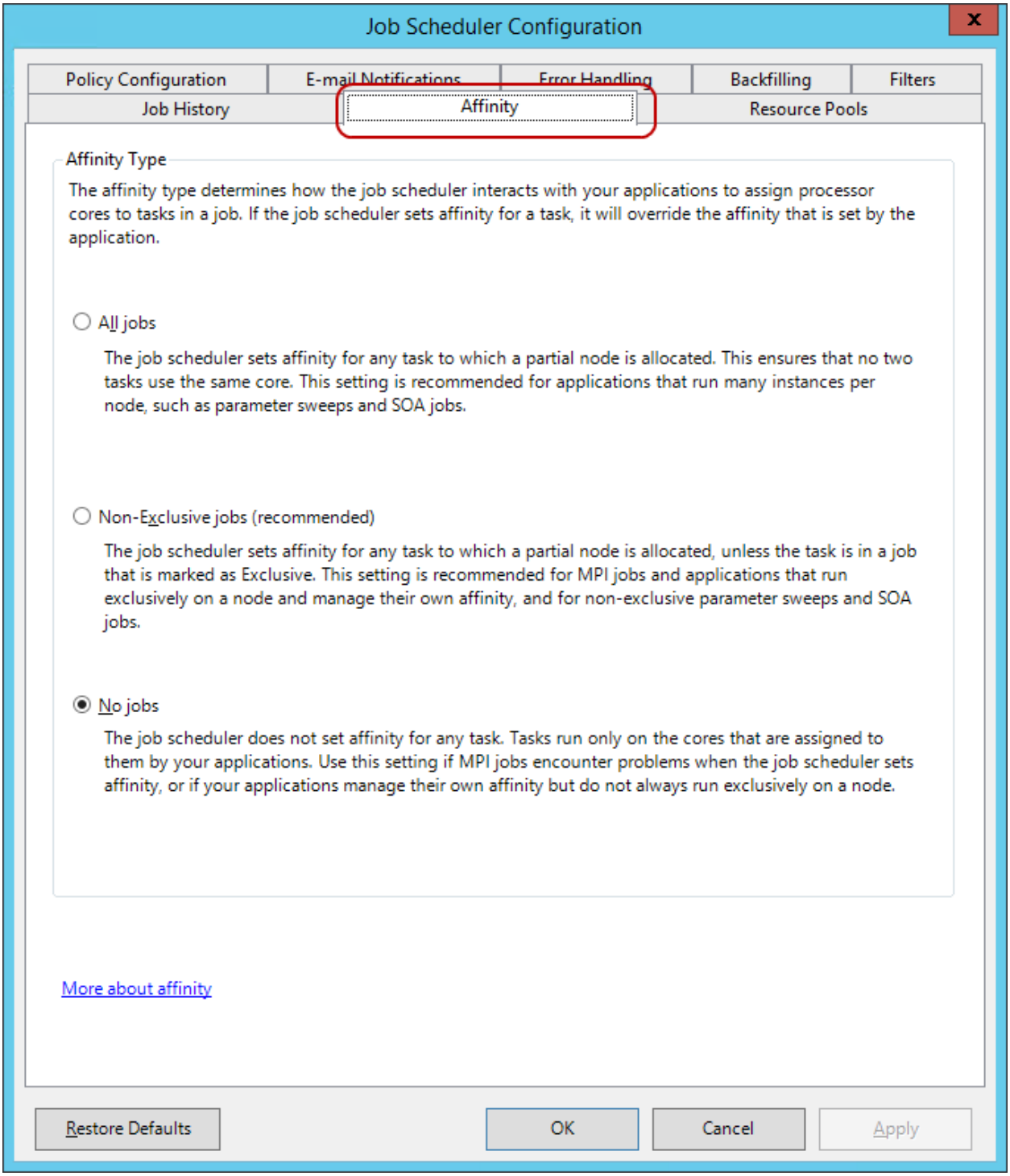
The *Job Scheduler Configuration* dialog box pops up, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Next, select the "Affinity" tab should be selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The cluster administrator may now select a radio button to select the desired Affinity Type:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs
- Non-Exclusive jobs
- No jobs

In order to submit non-exclusive jobs, the "No jobs" option should be selected. The "Non-Exclusive jobs" option may be selected, but this will not allow non-exclusive Ansys Electromagnetics jobs to be submitted to the cluster. The "All jobs" option is not supported for Ansys Electromagnetics jobs. The OK or Apply buttons may be used to apply the selected option to the cluster.

Examples with "Submissions with Nodes are for exclusive usage by this job" Unchecked and Checked

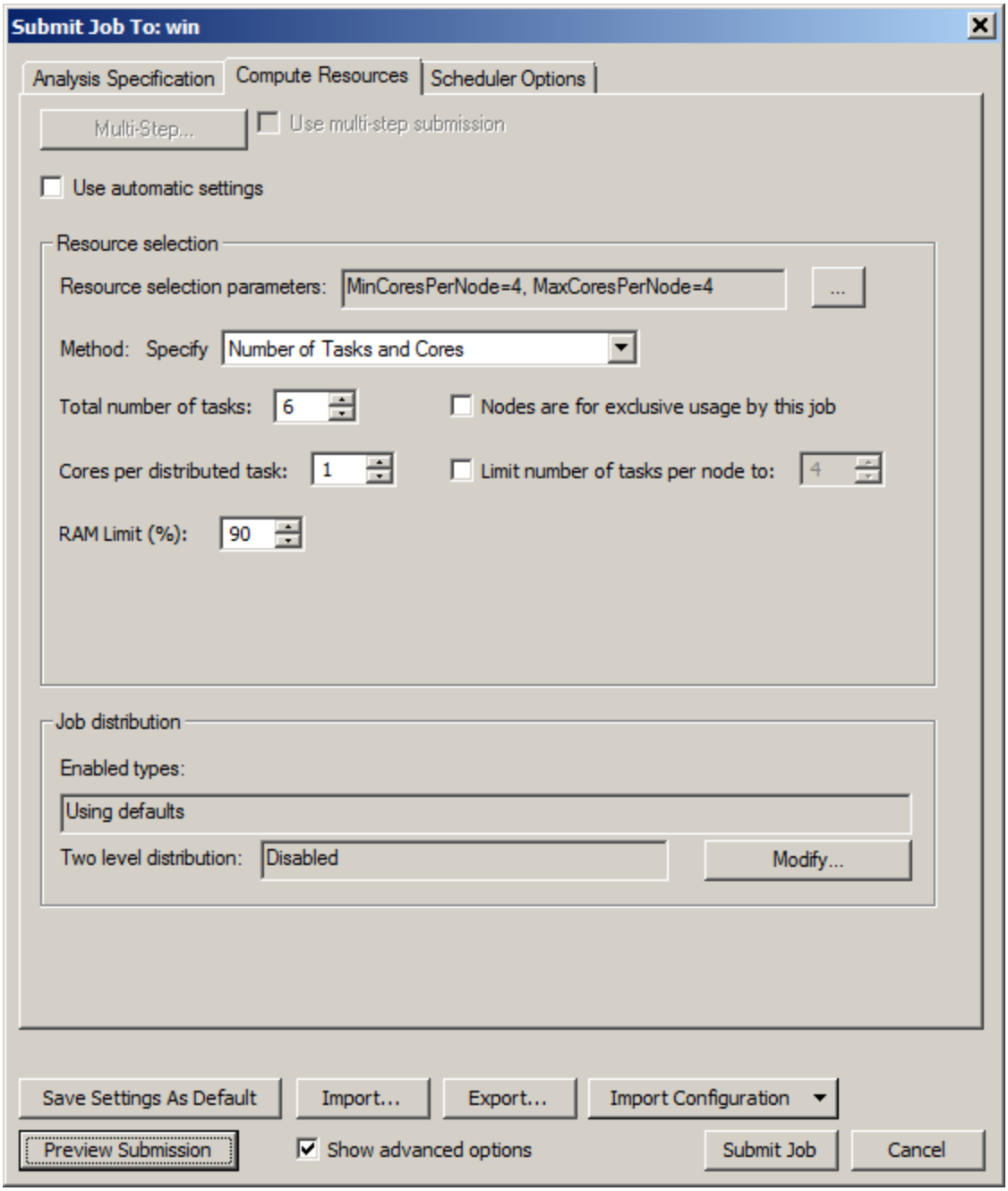
Most of the following screen shots are for the same project file submitted to a Windows HPC cluster with the AffinityType set to "No Jobs". The scheduler resource selection parameters are the same for all of these screen shots:

- MinCoresPerNode=4
- MaxCoresPerNode=4

The first set of screen shots show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is not checked, and the resource selection method is "Specify Number of Tasks and Cores". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. If it is unchecked, then the job is submitted with the exclusive parameter for the job set to false.

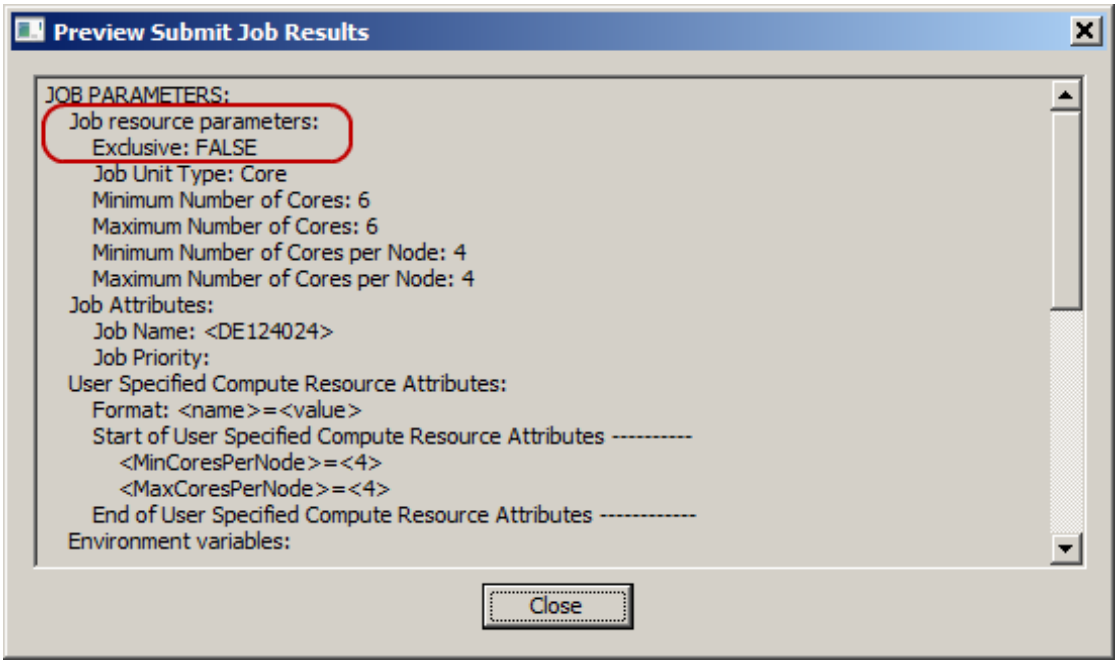
The following screen shot shows this configuration with a Submit Job with Task and Cores specified and nodes as non-exclusive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows Preview for the job submission indicating that the exclusive parameter for the job is false.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

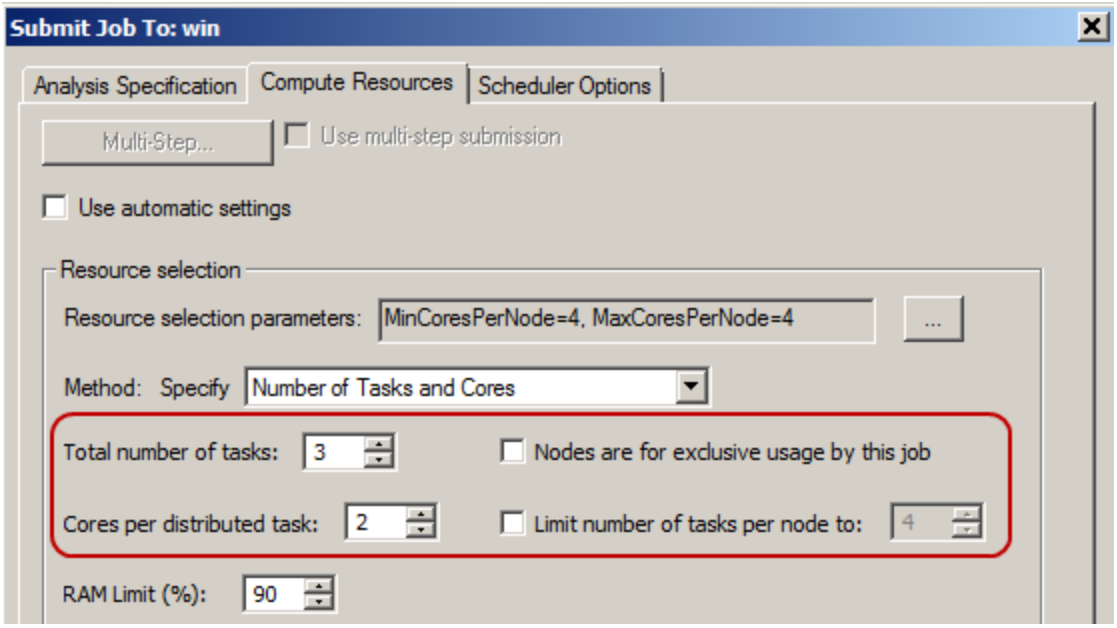


Jobs may only be submitted using this resource selection method with the exclusive parameter set to false if all three of the following conditions are met:

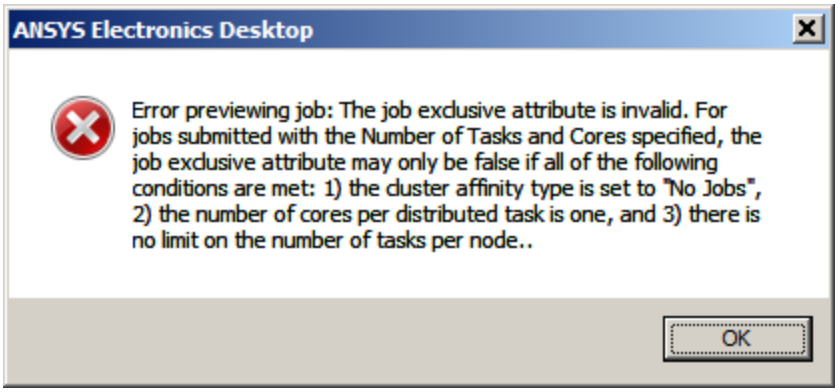
1. The cluster AffinityType is "No Jobs",
2. There is no multiprocessing, and
3. There is no limit on the number of tasks per node.

If any of these requirements are not met, then the job is not submitted or previewed, and an error message appears. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



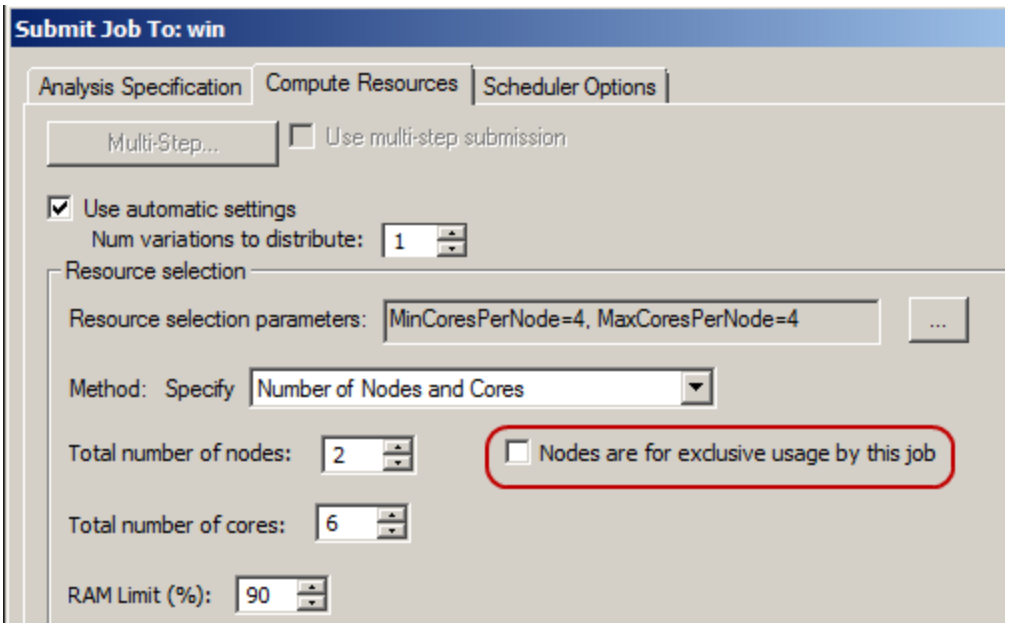
The following figure and screen shot ErrorMsgTasksAndCoresNonExMP.png shows the error message box shown to the user.



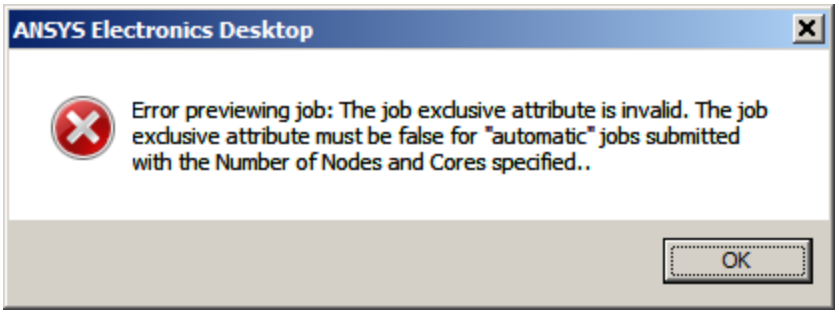
The next example shows submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Nodes and Cores". If you select *Use automatic settings* with *Num variations to distribute* set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set Num variations to distribute to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically. The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

behavior is the same as previous releases. If it is unchecked, then the job is not submitted or previewed because only exclusive jobs are allowed when using this resource selection method, and an error message box is shown to the user in this case. The following figure shows this configuration:

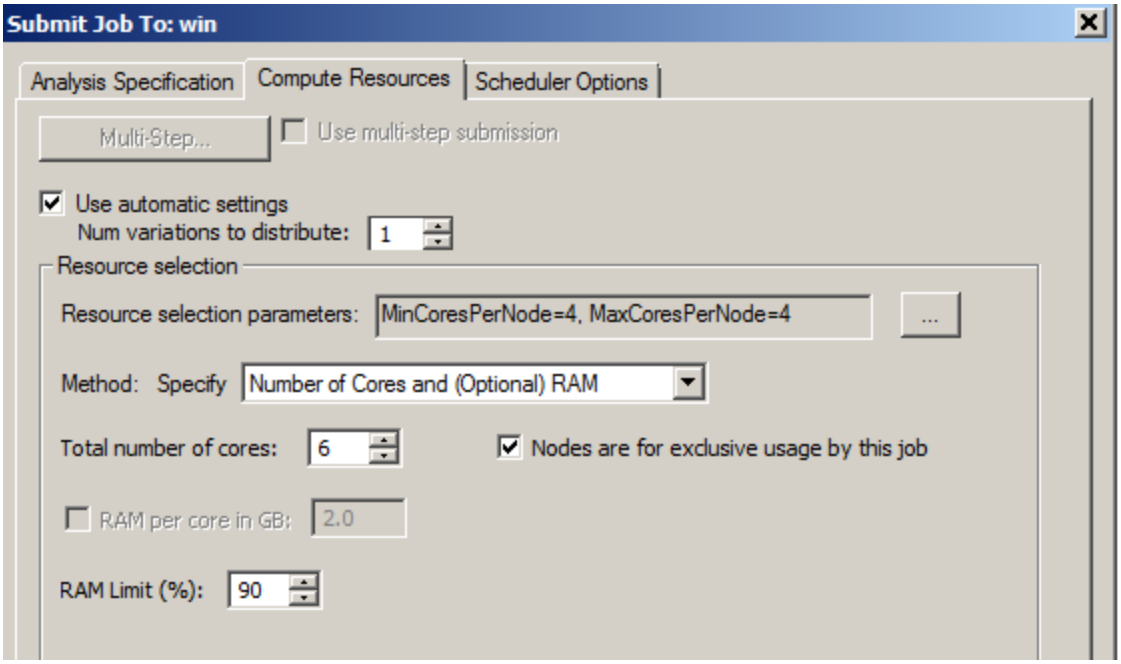


The following figure shows the error message.

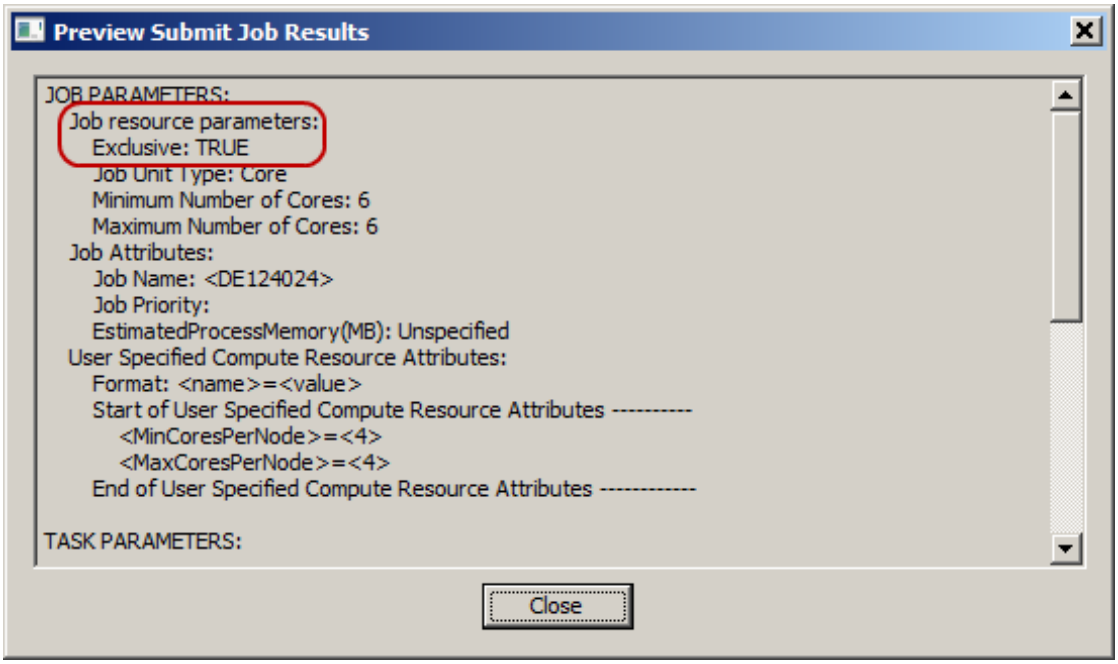


The next example show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Cores and (Optional) RAM". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and you may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

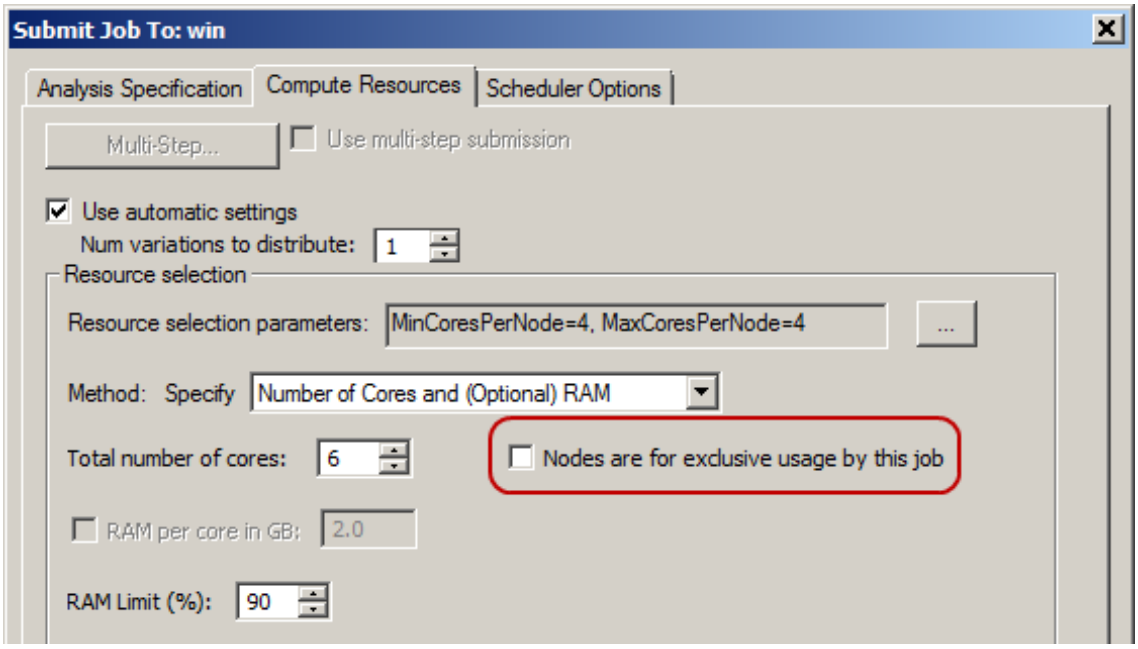


The following figure shows the Job Preview for this case, indicating that the exclusive parameter for the job is true.

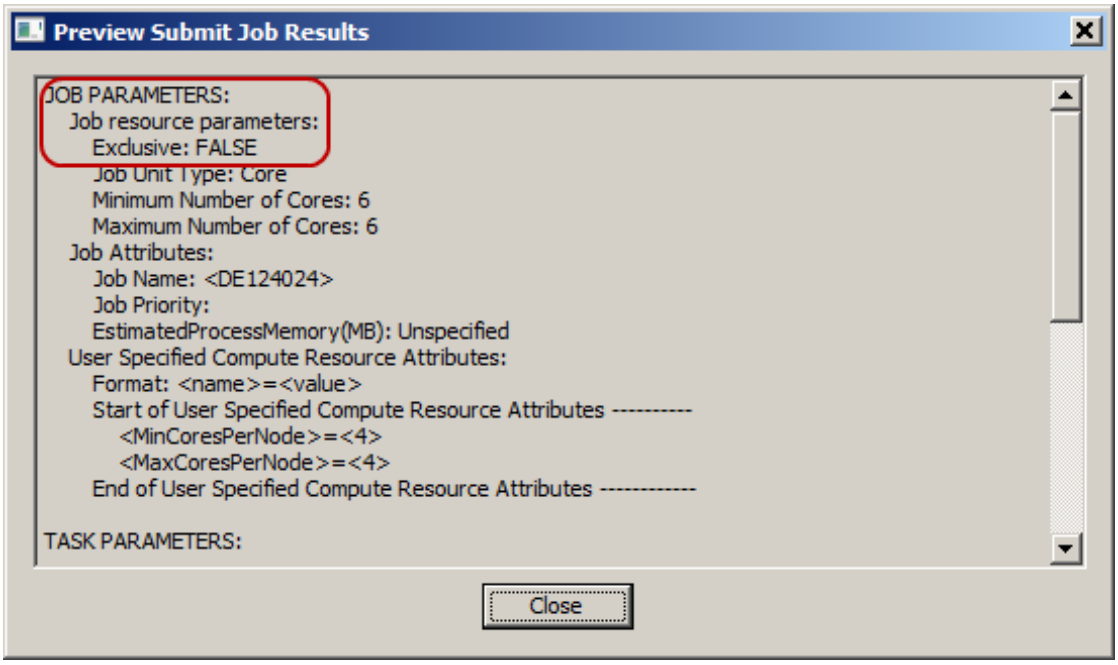


If the exclusive check box is unchecked, then the job is submitted with the exclusive parameter set to false. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



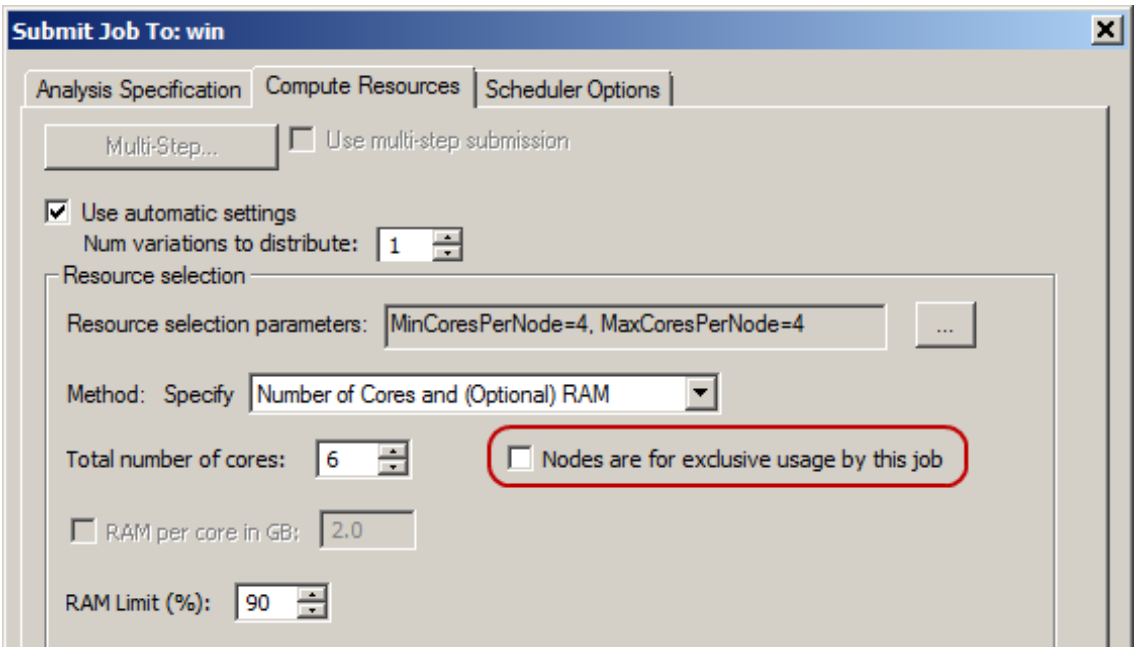
The following figurescreen shot PreviewCoresAndRAMNonEx.png shows the Job Preview for this case, indicating that the exclusive parameter for the job is false.



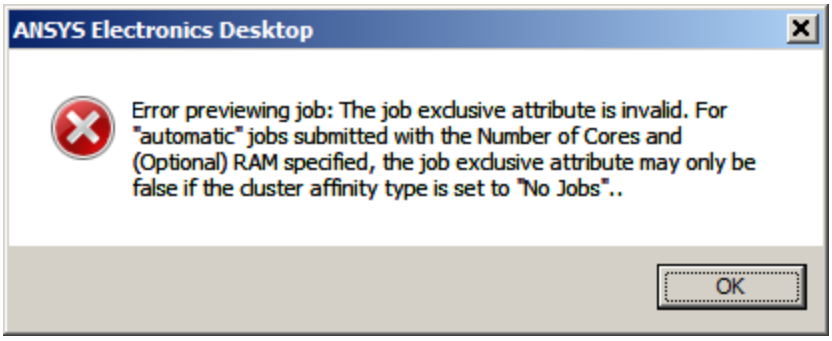
The last example resembles the previous one, except that the Windows HPC cluster AffinityType is set to "Non Exclusive Jobs" in this case. The job is submitted with the "Use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatic settings" check box in the checked state, and the resource selection method set to "Specify Number of Cores and (Optional) RAM". Because the AffinityType is not "No Jobs", only exclusive jobs may be submitted to this cluster. If the exclusive check box is not checked, then the job is not submitted or previewed, and an error message box is displayed to the user. The following figure shows this configuration.



The following figure shows the error message.



Windows® HPC Job Credentials

Normally, a user will be prompted for the credentials used to submit a job. One way to simplify this process is to use the "cluscfg setcreds" command to set the user's credentials in the credentials cache. If this is done, then no password needs to be supplied for a job submitted for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the specified user. Here is a cluscfg command that may be used to set the user credentials in the credentials cache:

```
cluscfg setcreds /password:* /scheduler:cluster_name
/user:domain\user_name
```

Here:

- cluster_name = the name of the cluster (hostname of the head node)
- domain = optional domain name; if omitted, the following \ should also be omitted
- user = user name

When this form of the command is used, the user is prompted for the password and also asked if the password should be remembered (cached).

See the following web page for more information on the cluscfg setcreds command:

http://technet.microsoft.com/en-us/library/cc947669(WS.10).aspx

Integration with Platform's Load Sharing Facility (LSF)

The Load Sharing Facility (LSF) scheduler is only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using LSF commands (qsub, etc.)

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

> **Note:**
>
> If a temp directory is setup by the LSF cluster administrator, analysis engines use this temp directory, overriding the setting in the Ansys EM product.

LSF Job Management

You can use Ansys Electronics Desktop to submit batch jobs to LSF and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select LSF as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to LSF.

   LSF-specific Settings:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the following:
  - **Queue** – a drop-down menu lets you select Normal, chkpn_rerun_queue, idle, license, night, normal_allow_excl, owners, priorityu, or short.
  - **MinCoresPerNode** – the minimum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus>=N]
  - **MaxCoresPerNode** – the maximum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus<=N]
  - **MinMemoryPerNode** – the minimum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem>=M]
  - **MaxMemoryPerNode** – the maximum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem<=M]

  If you do not specify parameters, the scheduler does so.

- For LSF, the only non-automatic method of **Resource Selection** is **Number of Tasks and Cores**. You can specify the number of tasks, whether they are for exclusive use by the job, cores per distributed task, and a limit number of tasks per node.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

LSF Job Submission Guidelines

Before submitting an LSF job, ensure the following are true:

- The project is available in a shared drive that is accessible to all machines in the cluster.
- The project is available using the same path on all machines of cluster.
- There is sufficient space in the project directory and temp directories.
- There is sufficient memory per engine.
- The number of compute resources (Distributed Analysis machines and Multi Processing cores) will achieve the desired scale factor and effective resource utilization.

Integration of Ansys EM Products with LSF

With LSF you do not need to set 'Distributed Analysis Machines' or 'Remote Machine' options. Instead, you submit a job to LSF, requesting the appropriate resources for the job (number of processors, memory per processor, etc.).

For example:

```
bsub -n 1 ansysedt.exe -Batchsolve -ng -local -machinelist num=1
OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
bsub -n 4 ansysedt.exe -Batchsolve -ng -Distributed -machinelist
num=4 OptimTee.adsn
```

The job is queued by LSF until the requested resources are available. Upon resource availability, LSF starts Electronics Desktop with the specified command line on one of the allocated machines. During analysis, Electronics Desktop dynamically obtains the allocated 'Distributed Analysis Machines' from LSF. Electronics Desktop interfaces with LSF to launch engines on remote machines without going through Ansys RSM.

Installing Ansys EM Tools on LSF Cluster

The LSF scheduler is supported on Linux only.

lsf.conf should contain this line:

```
LSF_UNIT_FOR_LIMITS=MB
```

The administrator should have this line in the lsb.params file to ensure that memory reservations are per-slot (per-core):

```
RESOURCE_RESERVE_PER_SLOT=Y
```

Jobs may be submitted in any of the following ways:

- Using LSF commands (bsub, etc.)
- Using the generic scheduler GUI in local mode
- Using the generic scheduler GUI in service mode

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

Setup:

1. Install on a single node, on a shared drive.
2. Setup 'temp directory' to a path that is same on all nodes (for example, /tmp).
3. Ensure that the product is available using the same path on all nodes.

Permissions:

- All users of the cluster should have read/write permissions to temp directory.
- All users should have read/execute permissions in the installation directory.
- Turn OFF any firewall between cluster nodes.

When a desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary; installation is sufficient. Select the scheduler through the Electronics Desktop GUI. Ensure that scheduler commands are available in the path before you launch Electronics Desktop.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

There is no need to install RSM unless you are using the scheduler GUI on a post-processing node that is different from the job submission node. In this case, RSM must be configured with the scheduler type and path.

A post-processing node is a node in the cluster that can run Electronics Desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Per-slot Resource Reservation

Set the cluster for per-slot resource allocation if the automatic cores and RAM resource selection method is to be used. You can check the cluster to see if per-slot resource allocation is configured by using the "bparams -a" command. Search the output for "RESOURCE_ RESERVE_PER_SLOT" to determine the setting (either "Y" or "N"). If set to "N" then consult the LSF administration guide on how to change this to "Y".

Scenario 1: The post-processing node and job-submission node roles are served by distinct machines.

In this case, perform the following configuration:

The job-submission node should be configured to run RSM service, which serves as a proxy to scheduler. The RSM Service should be running as 'root' in order to facilitate jobs running using the credentials of the job's owner. A **configuration file in the RSM installation folder should be edited** to specify information regarding the scheduler that manages jobs on this cluster. A block labeled 'Scheduler' must be included within the 'AnsoftCOMDaemon' block. This block contains two string entries:

- SchedulerName: this contains the unique part of the scheduler proxy library name
- ConfigString: this contains a scheduler specific configuration string

The case of the SchedulerName string is significant on Linux because Linux file names are case sensitive. In Ansys Electromagnetics Suite 2025 R1, possible scheduler names are: lsf and sge. The ConfigString entry is a scheduler specific configuration string, described below.

In addition, the AnsoftRSMService must be started with appropriate environment variables set. Generally, the environment variables must be set the same as they would be set for using the scheduler via command lines.

LSF Details

For the LSF scheduler proxy library, the ConfigString entry in the ansoftrsmservice.cfg configuration file is ignored. It may be empty or omitted entirely.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The AnsoftRSMService must be started with the environment set as it would be set for submitting jobs to the LSF cluster.

- For Linux, the cshrc.lsf or the profile.lsf file may be sourced to set up the environment, depending on the shell.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='lsf'
'ConfigString'=''
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Scenario 2: The post-processing node and job-submission node roles are served by the same machine.

The **Select Scheduler...** command (as described in the Job Management User Interface for LSF section) is used to gather details about the scheduler. In this case, the Desktop process should be started in an environment suitable for submitting jobs to the scheduler. See below for details.

The environment should be configured so that the following LSF environment variables are set appropriately for the LSF cluster in use: LSF_BINDIR, LSF_SERVERDIR, LSF_LIBDIR, and LSF_ENVDIR. In addition, the following LSF commands should be found in the LSF_BINDIR directory: "bsub", "bjobs", "bkill", "lsid", "lsrun", "lshosts", "bmgroup", "bparams" and "bqueues".

Using the bsub Command to Submit Batch Jobs

The LSF bsub command may be used to submit jobs. The typical command format is:

`bsub` *`bsub_args ansys_exe ansys_args`*

where:

- *bsub_args* are the options of the bsub command,
- *ansysEM_exe* is the pathname of the Ansys Electromagnetics desktop executable to launch, and
- *ansys_args* are the arguments to the Ansys Electromagnetics desktop executable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bsub Arguments

The **bsub** command has a large number of options that may be used to control the submission process. Only a few options that are often used with Ansys Electromagnetics jobs are listed here. The following options may be used to submit serial or parallel LSF jobs.

-n*min_proc*, *max_proc* or **-n***min_proc*

Submits a parallel job, specifying the number of processors (or slots) required for the job. Here, *min_proc* is the minimum number of processors, and *max_proc* is the maximum number of processors. If no maximum is specified, then exactly *min_proc* processors are requested. If PARALLEL_SCHED_BY_SLOT=Y in lsb.params, this option specifies the number of slots required to run the job, not the number of processors. If the **-n** command line option is not specified, then the job is submitted as a serial batch job.

-R "span[ptile=n]"

There are many ways to use the -R "res_req" option to the bsub command. We only cover -R "span[ptile=n]" here, because this option is very useful for Ansys Electromagnetics jobs. When this option is specified, the LSF scheduler will allocate n processors (or slots) on each host to this job, even if more processors are available on the host.

-x

All hosts running this job operate in exclusive execution mode. The job will only run on a host having no other jobs running on that host. No other batch jobs will be started on a host while this job is running on that host.

See LSF documentation for a complete list of options for the **bsub** command.

> **Important:**
>
> If the Ansys EM tool executable pathname (*ansys_exe*) or any of the arguments of the Ansys tool command (*anssys_args*) contain characters which are interpreted by the command shell, these special characters must be properly quoted to ensure that the correct command is launched by LSF. A similar problem may occur if any of the *ansoft_args* require single quote, double quote or space characters. Note that the Ansys Electronics Desktop command is processed by the shell twice: once when the **bsub** command is processed, and again when the job starts. See: Example Command Lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example LSF bsub Command Lines (Linux Only)

> **Note:**
>
> The following examples use HFSS as the product, but similar command lines will work for all Ansys EM products.

Serial Job

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -n 1 option indicates that this job runs on one core.

Serial Job Requiring a Minimum of 4GB

```
bsub -n 1 -R "select[mem>4000]"
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -R "select[mem>4000]" option indicates that this needs a minimum of 4 GB memory.

Multi-processing Job using 4 Cores

```
bsub -n 4 -R "span[ptile=4]"
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -BatchSolve
-batchoptions -machinelist num=4 ~/projects/OptimTee.aedt"
```

- The -R "span[ptile=4]" option indicates that the four cores need to be on the same machine.
- The -batchoptions option indicates that HFSS should use four cores for multi-processing.
- The entire hfss command is in double quotes, and the double quotes enclosing the -batchoptions value are escaped. Each of these double quotes is replaced by the sequence "\"".

Distributed Processing Job using 4 Engines

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -
BatchSolve -Distributed ~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The -n 4 option indicates that the four cores are needed for the job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option can have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing

```
bsub -n 4 -R "span[ptile=2]" ~/projects/OptimTee.csh
```

Shell Script (~/projects/OptimTee.csh):

If a command is included in the **bsub** command line, the entire command will be processed by the command shell two times. The command is processed when the **bsub** command is processed by the shell and is processed again when the command is started by the scheduler. This example shows how to use a shell script so that the command line will be processed only once. The command is placed in the shell script, and then the shell script pathname is placed in the **bsub** command line. Then, the command is only processed by the command processor when the job is started. When using this approach, the shell script should be accessible from all of the cluster hosts.

```
#!/bin/csh
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -BatchSolve
-Distributed -machinelist num=2 -batchoptions
~/projects/OptimTee.aedt
```

- The **-n 4** option indicates that the four cores are needed for the job.
- The **-R "span[ptile=2]"** option indicates that the cores must be allocated in groups of two cores on the same machine.
- The **-machinelist num=2** option indicates that this is a DSO job and that a total of two engines will be started.
- The **hfss** command is placed in the shell script (~/projects/OptimTee.csh). In the **bsub** command line, the **hfss** command is replaced by the shell script pathname.

Monitoring LSF Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under LSF.

Ansys Electronics Desktop -monitor Command Line Option

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -monitor command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

LSF bpeek Command

The LSF bpeek command may be used to monitor job progress. The command **pbeek [-f]** *job_id* displays the standard output and standard error produced by the job with id *job_id* from the job start to the current time (the time when the command is executed). This command is only valid for jobs that have not yet finished. When used with the **-f** option on Linux, the output of the job is displayed using the command tail **-f**, so that ongoing progress may be monitored.

In order to display messages to standard output and standard error, specify the **-monitor** command line option on the Ansys EM tool command line. Then, these messages can be seen using the LSF bpeek command.

Terminating LSF Batch Jobs

To cancel or terminate an Ansys EM LSF batch job, we recommend using the job monitoring UI to terminate jobs cleanly, rather than using the bkill commands. Using this approach will allow the batch job to shut down in an orderly fashion.

Using the LSF **bkill** command without the **-s SIGTERM** option or simply terminating the job processes may cause some of the following problems:

- Some engine processes are not shut down and continue to run.
- LSF job is not fully removed.
- Project .lock file is not removed.
- MainWin core service processes (watchdog, mwrpcss and/or regss) are not stopped.

Some of these may interfere with submission of additional LSF batch jobs. For example, it may be necessary to manually remove the project lock file to submit another batch job for the same project. MainWin core service processes may also interfere with starting subsequent Ansoft batch jobs. Normally, these processes should timeout and end 15 seconds after the Ansys Electromagnetics product shuts down. Any MainWin core service processes (watchdog, mwrpcss and/or regss) that continue to run for more than 15 seconds after the product has stopped may be hung. The hung processes may need to be manually killed, after ensuring that these processes are associated with an Ansys EM job that has finished or terminated.

Stop a job cleanly - ensures that the results obtained until now are preserved:

```
bkill -s TERM <jobid>.
```

Stop an job abruptly - results are most likely lost. You have to manually remove the project lock file:

```
bkill <jobid>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSF Known Issues and Workarounds

The following are known issues. Workarounds are noted when available:

- Desktop or remote machine cannot have multiple IP addresses. This is unsupported.
- Core dump files may appear when a job has finished running. Results are still computed correctly. Workaround: Limit size of core dumps to 0 through the following job submit option: `bsub -C 0 -n <number-of-cores> -q <queue-name>`
- Firewall should be turned off on all machines in the cluster.
- Sometimes LSF ends a job (for example, a job may be preempted due to a high priority job). This may result in the presence of a .lock file in the project directory. You must manually delete the .lock file before continuing with further analysis.
- When an LSF job is ended, MainWin services (watchdog, regss, and mwrcpss) could keep running. The result is that later jobs cannot start on the machine. The fix is to end these processes before starting a new job.
- Analysis fails abruptly when running out of resources (cpu/memory/disk). Ensure sufficient resources are provided.

LSF Troubleshooting

The following are general troubleshooting steps:

1. Ensure the LSF `lsrun` command is enabled.
2. Look for user errors.

   For example:

   - Are the executable path and project path correct and complete?
   - Are there sufficient resources (CPU/Memory/Disk) allocated to the job?
   - Is the project available on the execution host?
   - Does the job submitter have read/writer permissions on the project directory and read/execute permissions on the installation directory?
   - Is the project locked?
3. Determine whether this is a standalone product issue.
   - Run Electronics Desktop on the machine outside of the scheduler and see if it opens and analyzes.
4. Examine outputs and logs.
   - Output of the LSF batch job. Obtain this using LSF commands: `"bacct -l <jobid>"`
   - Batch log (typically <projectname>.log, located in the project directory.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enable additional debug logs using the steps below.

   In the job submission window, set the following environment variables:

   - ANSOFT_DEBUG_MODE = 1
   - ANSOFT_DEBUG_LOG = <*path to directory accessible by all machines in the cluster*>
   - ANSOFT_DEBUG_LOG_SEPARATE = 1
   - ANSOFT_LSF_LOG = <*path to a specific .log file in the directory set under ANSOFT_DEBUG_LOG*>
6. For each pair of machines between which remote analysis fails, run `ping remote-machine` and note the output.
7. For each machine in the network, dump network interfaces (for example, run `ifconfig -a`) and note the output.

Integration with PBS (Portable Batch System)

The PBSPro and PBS/Torque schedulers are only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using PBS commands (qsub, etc.) or the PBS gui (xpbs)

For information about supported schedulers, visit the Platform Support page. Then locate and click the following link:

Ansys 2025 R1 – Job Schedulers and Queuing Systems Support

PBS Job Management

You can use Ansys Electronics Desktop to submit batch jobs to PBS and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select PBS as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to PBS.

   PBS-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify either the **Queue** or **QueueAtServer** parameter.

     For **Queue**, you may select a queue for the job from the list of queues configured for the default server. Only queues that are enabled and that do not have the from_ route_only attribute set to true are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **QueueAtServer**, you may specify a queue at the default server or at another server by entering text into this field in one of three formats:

- queue_name
- @server_name
- queue_name@server_name

The queue_name format submits with the name queue_name. The other formats submit with the name server_name. The *destination* value of the -q *destination* option on the qsub command line is the user-specified string. *This string will not be validated by the scheduler proxy library*.

Either **Queue** or **QueueAtServer** may be specified, but both may not be specified. If neither **Queue** nor **QueueAtServer** is specified, the job is submitted to the default queue at the default server.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Non Standard Installations for PBS

PBSPro

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBSPro configuration file. The default pathname of the PBSPro configuration file is /etc/pbs.conf. The environment variable PBS_CONF may be used to specify a different pathname for the PBSPro configuration file.

PBS/Torque

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBS/Torque server file. The name of the PBS/Torque server file is server_name, and it is installed in the TORQUEHOME directory. By default, TORQUEHOME is /var/spool/torque. To specify a different TORQUEHOME directory, the environment variable ANSOFT_TORQUEHOME should be set to the pathname of the desired directory.

PBS Limitations

General Limitations

- There is no support for GPUs when submitting jobs via the GUI.
- Support for PBSPro and PBS/Torque is only available on Linux; Windows is not supported.
- Staging of input or output files is not supported for jobs submitted using the GUI.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs submitted via the GUI are independent jobs. Neither job dependencies nor job arrays are supported.
- If the user specified server is not the default server, then there is no check for sufficient resources before submitting the job.
- If the user specified server is not the default server, then the limit on the number of tasks per node is ignored for both PBSPro and PBS/Torque. For PBS/Torque, only one task will be allocated for each node. For PBSPro, the scheduler may allocate any number of tasks to a node, provided that the node has sufficient cores for all of the tasks.
- For jobs submitted to a routing queue, the check for sufficient nodes and cores only verifies that there are sufficient nodes and cores associated with the server.
- The queue attributes “resources_max” and “resources_min” are not checked when determining whether there are adequate resources to run the job.

PBSPro Limitations

- The PATH in the submission user’s default environment must include the directory containing the PBSPro commands.
- Failover is not supported.
- HPC Basic Profile Jobs are not supported.
- Globus vnodes are not supported. When checking for sufficient nodes and cores for the job, only nodes of type PBS are considered.
- For jobs submitted to an execution queue, the only vnode attribute used to determine if a vnode is available to the job is the “queue” attribute.

PBS/Torque Limitations

- When checking for sufficient nodes and cores for the job, only nodes of type cluster are considered.
- The “exclusive” check box has no effect for PBS/Torque.
- For PBS/Torque, even if a job is submitted to an execution queue at the default server, there is no check for sufficient nodes and cores available to the queue. All of the server’s execution nodes are assumed to be available for the job.
- For PBS/Torque, there are significant limitations when submitting a job in which the number of tasks and number of cores per task are specified. Unlike PBSPro, there is no capability to specify that the cores should be allocated in “chunks”. Instead, the submission command includes the number of groups of nodes and the number of processors per node (ppn) for each node in the group. To determine the size of each group and the ppn setting for each group, the server nodes are examined from largest number of cores to smallest. This may not be optimal because some of these nodes may not be usable by the queue specified for the job, or because the nodes with the largest number of cores may be busy. Similar issues could occur for PBSPro, but they should be less likely, because only the nodes usable by the queue are considered.

Submitting Jobs Via Ansoftrsmservice on a Different Host

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before starting the ansoftrsmsevice as a daemon on a job submission host, the 'Scheduler' section of the ansoftrsmservice.cfg must be specified.

This section contains two settings, 'SchedulerName', which must be set the string 'pbs', and 'ConfigString', which must be set to the pathname of the directory containing the PBSPro or PBS/Torque commands.

Here is an example ansoftrsmservice.cfg file, showing the format of this file and an example 'ConfigString' setting:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='pbs'
'ConfigString'='/share/pbs/default/bin'
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Submitting Ansys EM PBS Batch Jobs

The PBS qsub command may be used to submit Ansys EM batch jobs. The typical command format is:

qsub *qsub_args script*

where:

- *qsub_args* are the options of the **qsub** command,
- *script* is the pathname of the job script.

The job script is a shell script containing the Ansys batch command or commands to be run. If a batch command line contains any characters that are special to the shell running the script, then these special characters should be quoted, as needed. The job script may also contain PBS directives on lines before the first executable line of the script. Any **qsub** options on the command line will take precedence over the PBS directives in the job script.

When a PBS batch job is started, the job script runs as the job user in a new shell. In this shell environment, the path must include the directory containing the PBS commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You should ensure that the PATH variable set in the shell startup script (e.g., .cshrc, .profile, .bashrc) includes the directory containing the PBS commands. For example:

```
export PATH=/opt/pbs/default/bin:$PATH
```

If the PATH variable is not set correctly, the job runs only locally, the batch log file shows the list of allocated hosts as empty, and the error file shows an error (sh: qstat: command not found.)

Further PBS directives need to be on top of the job script file. This is discussed in the PBS documentation.

Serial PBS Batch Jobs

In the PBS documentation, serial batch jobs are also called single-node jobs. In general, any job submitted without specifying the -l nodes=value command line argument, will run as a serial or single-node job.

See Monitoring PBS Batch Jobs for options that can facilitate monitoring of Ansys Electromagnetics batch jobs.

Parallel PBS Batch Jobs

In the PBS documentation, parallel batch jobs are also called multi-node jobs. When an Ansys Electromagnetics batch job is run as an PBS parallel job, the PBS scheduler will select the hosts for the distributed analysis job based on the qsub command line arguments, the PBS resource directives from the job script, and the status of the hosts when the job is run. The desktop process will be started on one of these hosts. The desktop process will obtain the list of hosts allocated to the job from the PBS scheduler, and start analysis processes on these hosts, as needed, using the PBS scheduler facilities. To run a PBS parallel job, the job must be submitted with a -l nodes=value **qsub** command line argument or with a -l nodes=value PBS directive in the job script.

Monitoring PBS Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under PBS.

PBS qstat Command

The PBS **qstat** command may be used to display information on jobs and queues. In this section, several qstat command line options that may be used to monitor job progress are described.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **qstat -a** command displays information about all jobs in the system.

The **qstat -r** command displays information about all running jobs in the system.

The **qstat -s** command resembles the **qstat -r** command; the only difference is that a comment from the scheduler or batch administrator is also shown for each job.

The **qstat -au** *userid* command displays information about all jobs owned by user userid.

The **qstat -f** *jobid* command displays all available information about the job with id *jobid*.

See the PBS manual pages for more information.

Ansys EM -monitor Command Line Option for PBS

The Ansys EM **-monitor** command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

The PBS scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o** [*hostname*:]*pathname* and the **-e** [*hostname*:]*pathname* command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname for that stream.

The **qsub -j join** option controls whether the standard error stream for the job will be merged with the standard output stream for the job. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard output file or stream. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard error file or stream. A join value of n indicates that the standard output and standard error streams will not be joined. If the **qsub -j join** option is not specified, then the standard error and standard output streams will not be joined.

A user can monitor the progress of a job by checking the standard output file for progress, info and warning messages, and checking the standard error file for error and fatal messages.

qsub Arguments

The PBS **qsub** command has a large number of options for control of the submission process. In this section, we review the -l nodes=*value* command line option with Ansoft parallel batch jobs.

This option or directive has the following format:

`-l nodes=`*`node_spec`* `[+`*`node_spec...`*`][#`*`suffix`*`]`

where *node_spec* is one of the following

`nodename[:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Host name of the specified node, followed by optional **ppn** or **cpp** specifiers.

`[`*`N`*`][:`*`property`*`[:`*`property...`*`]][:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional number of nodes, followed by optional node properties, followed by optional **ppn** or **cpp** specifiers. If the number *N* is omitted, then the default value of 1 host is used.

Here, the optional **ppn** or **cpp** specifiers *pc_spec* are of form:

`ppn=X`

Number of processes (tasks) per node. Default is 1 if not specified.

`cpp=Y`

Number of CPUs (threads) per process. Default is 1 if not specified.

The optional global suffix, #suffix, which applies to all hosts has one of the following values:

`#excl`

This suffix requests exclusive access to the allocated nodes.

`#shared`

This suffix requests shared access to the allocated nodes.

The total number of requested processes is determined by adding up the product of the number of nodes and the number of processes per node for each *node_spec*. In general, this should match the number of distributed engines specified in the Ansys Electromagnetics desktop -Machinelist num=*num_distributed_engines* command line option.

The number of CPUs per process (**cpp**) specified in the PBS **qsub** command line or in the PBS directives in the script file should generally match the number of processors per engine specified in the Desktop **-batchoptions** value.

See the PBS documentation for a complete list of options for the **bsub** command, and further information on running multi-node jobs.

Example PBS qsub Command Lines

All of the following examples show how to submit Linux hfss jobs on PBS, but similar command lines and job scripts will work for all Ansys EM products. Most of the following examples are PBS "Single-node jobs." The last example is a PBS "multi-node jobs"; this example demonstrates how to specify the allocation of threads, tasks and nodes to a job.

Serial job:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Serial job that needs a minimum of 4GB memory and two hours of real (wallclock) time:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l walltime=2:00:00
#PBS -l mem=4gb
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```

Multi-processing job using 4 cores:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -batchoptions
-machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -batchoptions option indicates that Ansys Electronics Desktop should use four cores for multi-processing.

Distributed processing job using 4 engines on a single host:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -Distributed -
machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option may now have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed processing and multi-processing job using 8 cores on two nodes, running 4 engines (two per node) with 2 cores for multi-processing:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l nodes=2:ppn=2:cpp=2#excl
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -Distributed
-machinelist num=4 -batchoptions ~/projects/OptimTee.aedt
```

- The PBS directive **#PBS -l nodes=2:ppn=1:cpp=2#shared** indicates that two nodes are requested [2], two processes (engines) run on each node [ppn=2], and each process will use two cores [cpp=2]. The hosts allocated to this job may not be used for any other jobs while this job is running [#excl].
- The **-machinelist num=4** option indicates that this is a DSO job and that a total of four engines will be started. This option is required for all batch jobs.

Integrating Ansys EM Tools with Third-Party Schedulers

This document indicates how to create a dynamically linked library to allow integration of Ansys EM tools with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, it is then automatically loaded and used by Ansys EM tools.

- Introduction
- Common Requirements for Running Jobs
- Using a Shared Library (Linux) or a DLL (Microsoft Windows)
- Scheduler Proxy Interfaces
- Using an IronPython Program for Scheduler Integration

Introduction

Ansys EM Software Tools may be run as serial or parallel jobs on a cluster under control of a scheduler. Serial jobs are run using a single analysis engine at any one time on a single host. If the tool performs multiple analyses (for a frequency sweep or a parametric analysis, for example), the analyses are performed one after the other. Parallel jobs are run using multiple analysis engines running in parallel on the same host or on separate hosts. For parts of the analysis (such as meshing), the parallel job may use only a single analysis engine on a single host. Other parts of the analysis (such as a frequency sweep, parametric analysis or DDM) may be distributed to multiple analysis engines running in parallel.

- Serial Jobs
- Parallel Jobs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Serial Jobs

When an Ansys EM batch analysis runs as a serial job, the analysis engines run on the same host as the desktop process. The desktop process does not need to interact with the scheduler to get the names of hosts allocated to the job or to start processes on other hosts.

Parallel Jobs

For a parallel job, the desktop process starts multiple analysis engines that run in parallel. These engines may be started on the host where the desktop process is running, or on other hosts allocated to the job. The desktop process interacts with the scheduler to obtain information on the hosts that are allocated to the job, and to start engines on the local host or on other hosts allocated to the job. This document provides information on how to facilitate this interaction between the desktop process and the scheduler controlling the cluster.

For some popular job schedulers in a standard configuration, Ansys EM provides an "out of the box" integrated solution that will work with the scheduler. In this case, the Ansys EM installation includes code that will determine if the analysis is running as a scheduler job and communicate with the scheduler when needed. For other schedulers, the code to obtain information about the hosts allocated to a job and to distribute portions of the job to hosts assigned to the job is not provided in the installation. In order to facilitate using Ansys EM Software Tools with other schedulers, the user may provide a way for Ansys EM Tools to interact with the scheduler. Currently, two general approaches are available to users.

In the first approach, the user creates a shared library (on Linux) or a dynamically linked library (on Microsoft Windows) to provide communication between the Ansys EM Tool and the scheduler. This library is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EM Tool interacts with the library to get information from the scheduler, and to start additional processes on specified hosts. Each such library implements the same set of extern "C" functions needed to mediate the interactions between the Ansys EM Tool and the scheduler.

In the second approach, the user creates an IronPython program to provide communication between the Ansys EM Tool and the scheduler. This program is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EMs Tool uses the IronPython program to get information from the scheduler, and to start additional processes on specified hosts. Each python script contains a class implementing a specified interface, which contains functions needed to mediate the interactions between the Ansys EM Tool and the scheduler. The details of the interface are described below. The IronPython interface is equivalent to the extern "C" functions used in the first approach.

Common Requirements for Running Jobs

The following requirements must be met for serial and parallel jobs to run successfully. They apply whether using "out of the box" scheduler integration or scheduler integration using a library

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or using an IronPython program. *Host requirements* apply to all hosts that may be allocated to an Ansys EM serial or parallel batch job.

Installation Requirements

The Ansys EM installation directory must be accessible from all cluster hosts using the same path. One way to achieve this is to place the Ansys EM installation on a shared drive that is accessible to the cluster hosts using the same pathname. On Windows, this may require the use of UNC names to refer to the installation directory. Another option is to install the Ansys EM tool locally on each cluster host using the same local directory path.

Project File and Directory Requirements

The directory containing the project file must also be available from all cluster hosts using the same path. The project file and the containing directory must be readable and writable by the user account used to run the job. The controlling process for a distributed job is called the Desktop process, and it reads from and writes to the project file and other files in the same directory and its subdirectories. Although only the Desktop process reads from and writes to this directory, the Desktop process may be started on any of the hosts allocated to the job, so all hosts should have access to this directory using the same path.

For cloud submission such as Cloud Direct, Ansys HPC Platform Services, and Ansys Cloud Burst Compute there are no requirements for shared drives or shared directories (installation or project).

Using a Shared Library (Linux) or a DLL (Microsoft Windows)

This section describes how to create a dynamically linked library to allow integration of Ansys Electromagnetics Suite 2025 R1 with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys Electromagnetics Suite 2025 R1.

Installation Details

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansoft installation directory. For example, if the Ansys EM installation directory is "C:\Program Files\v251\AnsysEM", then the scheduler proxy library must be installed in directory "C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers.

The scheduler proxy library base name must match "libprefix_scheduler" on Windows and "liblibprefix_scheduler" on Linux. The extension must be a valid extension for a dynamically loaded library on the platform where it is used. The scheduler proxy library name prefix libprefix shall be unique, so it does not conflict with other scheduler proxy libraries in the same directory. To avoid confusion, the scheduler proxy library name should be all lower case on OSs where file names are case sensitive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Build Information for Scheduler Proxy Library

This section contains the recommended compiler and linker settings for building a scheduler proxy library.

- 64-Bit Microsoft Windows
- Linux

64-Bit Microsoft Windows

The proxy library should be compiled and linked as a 64-bit DLL, using the following recommended compiler and linker options:

Compiler Options

- Use of MFC: Use Standard Windows Libraries
- Character Set: Use Multi-Byte Character Set [/D "_MBCS"]
- Runtime Library: Multi-threaded DLL [/MD]
- Calling Convention: __cdecl [/Gd (default)]

Linker Options:

- Create a DLL [/DLL]
- 64-bit code [MACHINE:X64]

Linux

The proxy library should be compiled and linked as shared library (*.so) file. The following compiler and linker options are recommended when building using gcc/g++:

Compiler Options

- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Linker Options:

- Create a shared object file: [-shared]
- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Implementation Details for Custom Scheduler Integration

Function Name Prefix

Each exported function will have a scheduler specific function name prefix. The function name prefix will be the same as the library name prefix, except that it is converted to upper case. For

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

example, if the library name prefix is "lsf", then the function name prefix is "LSF". In the examples below, we use FN_PREFIX to denote the function name prefix.

The scheduler proxy library must provide implementations of the following extern "C" functions:

- IsProductLaunchedInYourEnvironment
- GetTempDirectory
- GetMachineListAvailableForDistribution
- GetMessageStringToRegisterForSigTerm
- LaunchProcess
- GetUseRsmForEngineLaunch
- GetThisJobID
- GetSchedulerDisplayName

IsProductLaunchedInYourEnvironment

Purpose

Determine if the program is running in the context of the scheduler for which this library was written.

Signature

```
extern "C" bool FN_PREFIX_IsProductLaunchedInYourEnvironment();
```

Arguments

None.

Return Value

Returns true if the current process is running as a job of the scheduler. Otherwise, false is returned.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetTempDirectory

Purpose

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
extern "C" bool FN_PREFIX_GetTempDirectory(char * buffer,
unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the temporary directory path name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the pathname of the temporary directory, including the terminal null byte, then the pathname is copied to the buffer and true is returned. If the buffer length is insufficient for the pathname of the temporary directory, then the buffer is unchanged, and false is returned.

> **Note:**
>
> To get the pathname of the temporary directory, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the pathname. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

GetMachineListAvailableForDistribution

Purpose

Get the list of hosts allocated to the current job. A host will appear in the list multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host appears in the list is equal to the number of processors or cores of the host that are allocated to the current job. The list is a text string containing a space separated list of hostnames.

Signature

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
extern "C" bool FN_PREFIX_GetMachineListAvailableForDistribution
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the list of machines available for distribution or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the lists of hosts, including the terminal null byte, then the list is copied to the buffer and true is returned. If the buffer length is insufficient for the list of hosts, then the buffer is unchanged, and false is returned.

Note:

- To get the list of hosts for distribution, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the list. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The hostnames in the list provided by this function shall be used in calls to LaunchProcess(). These host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

GetMessageStringToRegisterForSigTerm

Purpose

Obsolete. The string copied to the buffer should be an empty string.

Signature

```
extern "C" bool FN_PREFIX_GetMessageStringToRegisterForSigTerm
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the string or NULL.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string, including the terminal null byte, then the string is copied to the buffer and true is returned. If the buffer length is insufficient for the string, then the buffer is unchanged, and false is returned.

> **Note:**
>
> To get the string, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the string. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

LaunchProcess

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names in the list provided by the GetMachineListAvailableForDistribution function. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
extern "C" int FN_PREFIX_LaunchProcess(const char* hostName,
const char* exePathName, const char* arg1, const char* arg2);
```

Arguments

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

arg2: The second argument of the analysis engine command line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain newlines, tabs, spaces or other characters that are interpreted by the command processor, such as single quote (') or double quote (") characters, or dollar signs ($). Newlines or tabs may be replaced by spaces, if the newline or tab characters cannot be easily handled. If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.

GetUseRsmForEngineLaunch

Purpose

This function is optional. If this feature is not needed, then the function need not be implemented. Most schedulers should not need this feature.

For some schedulers, it may be desirable for the Ansoft RSM service to launch the engine processes instead of using the scheduler proxy library. For example, if the scheduler proxy library is limited to launching one process per host, then the scheduler proxy library may be used to launch one Ansoft RSM service executable per host, and the Ansoft RSM executable will launch all of the engine processes.

If the Ansoft RSM service should be used to launch engine processes for this scheduler, then this function shall be implemented and it shall return true.

If the Ansoft RSM service should not be used to launch engine processes for this scheduler, then this function is not required. If it is implemented, it should return false. If it is not implemented, it will be treated the same as if it was implemented and returns false.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Signature

```
extern "C" bool FN_PREFIX_GetUseRsmForEngineLaunch(void)
```

Arguments

None.

Return Value

Returns true if the Ansoft RSM service should be used to launch engine processes for this scheduler. Returns false if the Ansoft RSM service should not be used to launch engine processes for this scheduler.

> **Note:**
>
> This function is optional. If not implemented, then it is treated the same as if it was implemented and returns false.

GetThisJobID

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
extern "C" bool FN_PREFIX_GetThisJobID(char * buffer, unsigned
int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the Job ID or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string identifying the current job, including the terminal null byte, then the job ID is copied to the buffer and true is returned. If the buffer length is insufficient for the job ID, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the job ID, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the job ID. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- For many schedulers, the job ID may be obtained from the value of an environment variable.

GetSchedulerDisplayName

Purpose

Get a string identifying the scheduler associated with the current scheduler proxy library. This string is displayed to the end user to identify the scheduler.

Signature

```
extern "C" bool FN_PREFIX_GetSchedulerDisplayName(char * buffer,
unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the scheduler display name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the scheduler display name, including the terminal null byte, then the scheduler display name is copied to the buffer and true is returned. If the buffer length is insufficient for the scheduler display name, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the scheduler display name, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the scheduler display name. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The scheduler display name is generally a fixed string.

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" below
- "Distributed Analysis and Multi-Processing in the Same Job" on the next page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v251\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Scheduler Proxy Interfaces

Scheduler proxy supports following new graphical interface functions. The scheduler specific prefix of each function is not shown in this listing.

void Initialize(const std::string& config):

Initialize the proxy library for scheduler interaction. The **config** argument contains scheduler specific initialization information.

int CheckEnvironment(std::string& msg):

Check the environment in which the proxy library is running.

- Returns 0 (success) if the environment is appropriate for submitting jobs to the scheduler.
- Returns a non-zero error code if the environment is incorrect. If a non-zero error code is returned, an error message to display to the user is written to the msg argument.

int GetSchedulerInfo(std::string& msg, std::string& schedulerName, std::string& schedulerDescription, std::string& schedulerVersion):

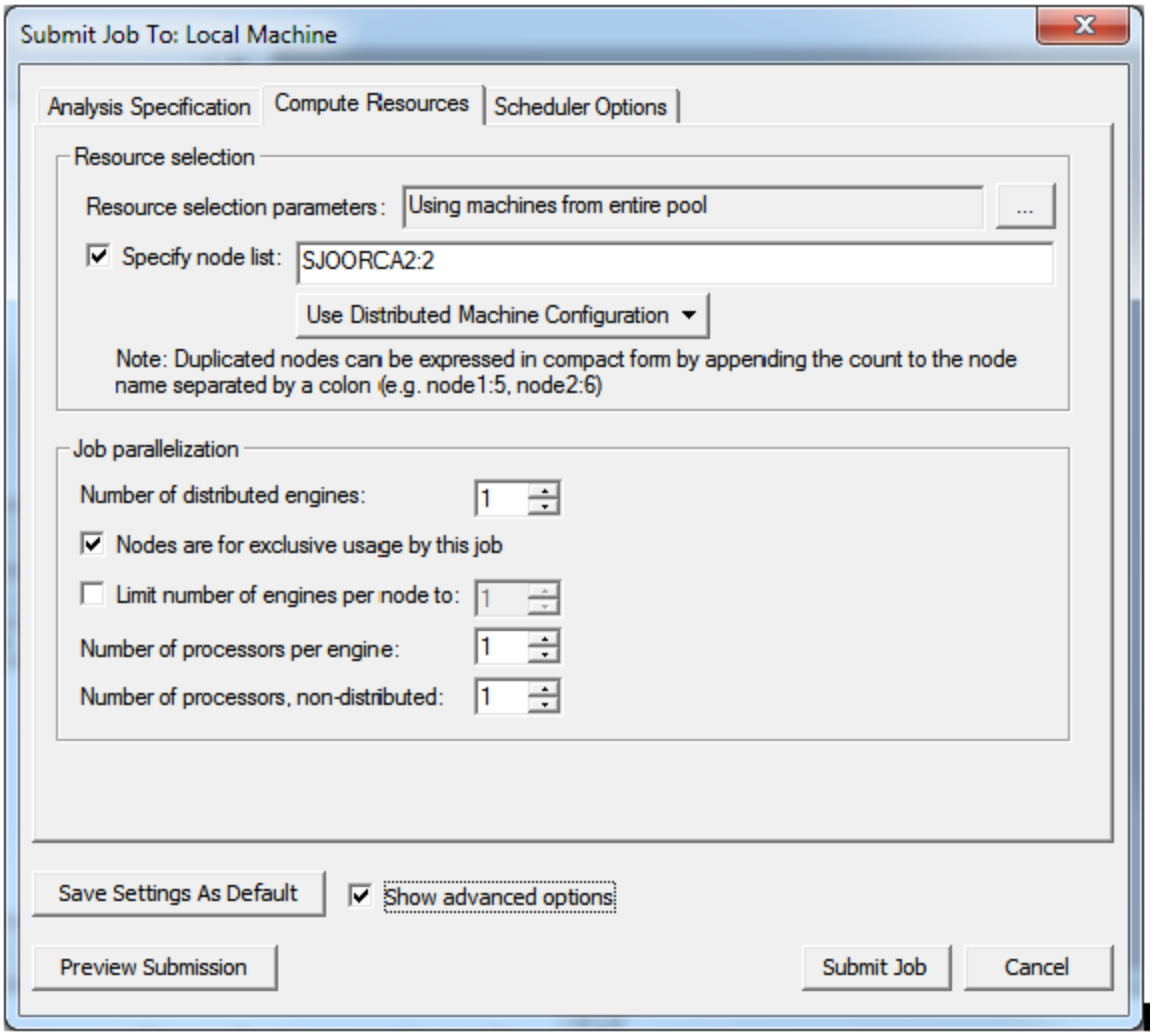
This function returns some basic information about the scheduler with which the scheduler proxy library interacts.

- On success, 0 is returned, and the scheduler name, scheduler description, and scheduler version are written to the **schedulerName**, **schedulerDescription** and **schedulerVersion** arguments.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

int GetComputeResourceAttributes(std::string& msg, AttributeDefinitionsStruct& attributeDefs):

The *Compute Resource* tab or the *Submit Job To* dialog box allows the user to specify scheduler specific resources. This function returns the information used to create and populate the *Compute Resource* tab.



Each line in the dialog box is defined by a single attribute definition in the **attributeDefs** argument. An attribute definition defines the name and description of an attribute, as well as information about the allowed values and the default value. In general, only the most commonly specified job attributes are included in the **attributeDefs** argument.

- On success, 0 is returned, and the attribute definitions are written to the **attributeDefs** argument.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.
- If the scheduler proxy library does not support any attributes using this approach, the **attributeDefs** argument will contain no attribute definitions, and 0 will be returned.

int AbortJob(std::string& msg, const std::string& jobID, bool force, const SubmissionUserStruct& submissionUser):

This function requests the scheduler to abort a job identified by the **jobID** argument. If the force argument is true, then errors should be ignored (the exact behavior is scheduler specific). The **submissionUser** argument contains information about the client user (the user running the Desktop process). The request to abort the job should run in the context of this user. If no user is specified, then the request to abort the job runs as the user of the process or thread running the function.

- If the request is successfully submitted, then 0 is returned.
- If there is an error, then a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

int SubmitUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the resources allocated to the job are uniformly distributed across the nodes allocated to the job. All other arguments are input arguments, and they are described below:

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

The **jobParallelization** argument contains information on how the job should be parallelized. It contains the following integral parameters:

- the total number of distributed engines,
- the number of cores to allocate for each distributed engine,
- the maximum number of engines to allocate to a single node (optional), and
- the number of cores to allocated for the non-distributed portion of the analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It also contains a boolean parameter indicating whether nodes used for this job should be exclusively allocated to this job.

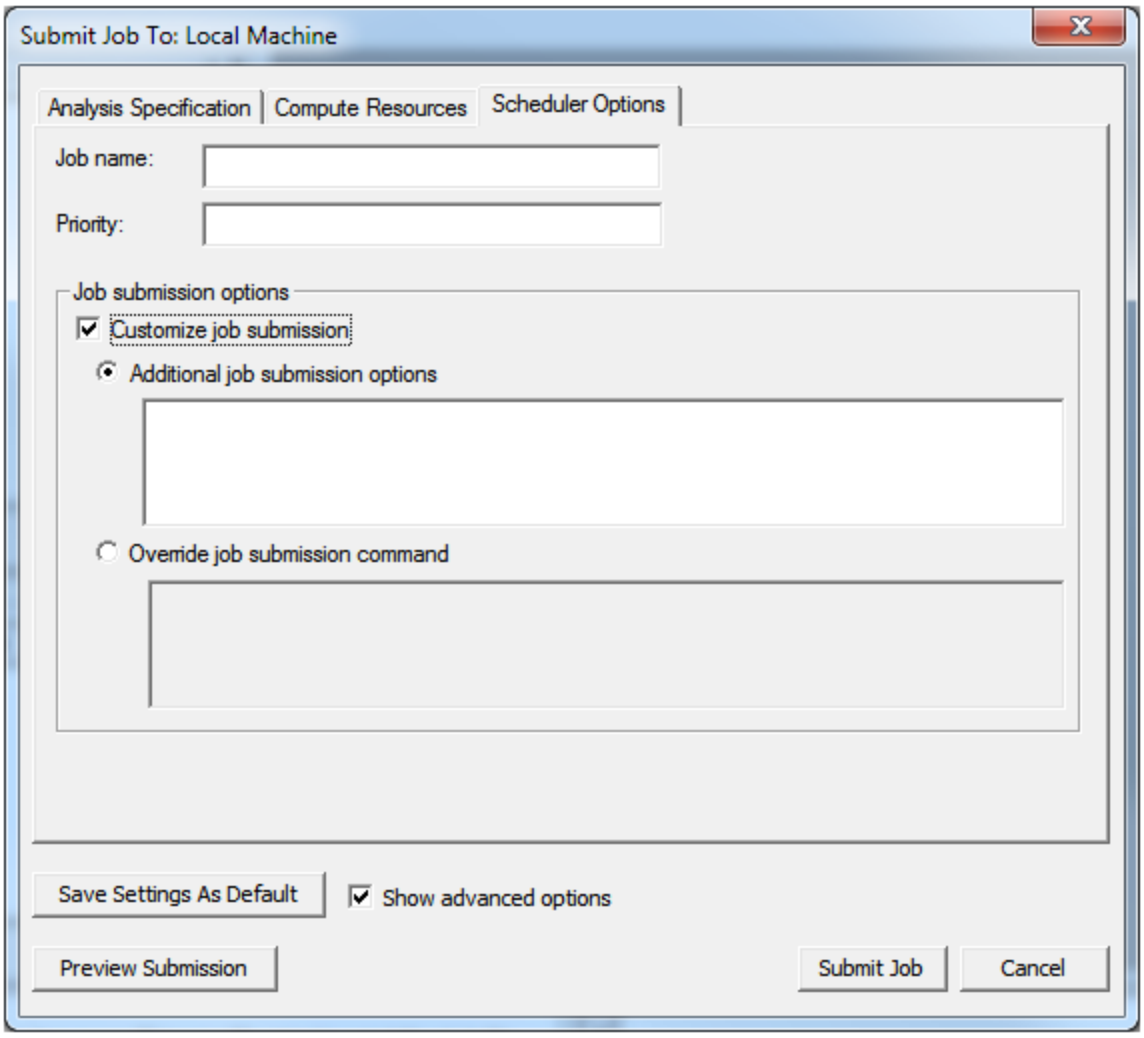
The **computeResources** argument is a reference to an object of type **UniformComputeResourcesStruct**. This **struct** contains zero or more resource attribute settings for the job. Each resource attribute setting consists or a resource name and a resource value. The resource name is the name of one of the resources defined in the **AttributeDefinitionsStruct** filled in by the **GetComputeResourceAttributes()** function. The resource attribute value is the value specified for the resource attribute by the user using the *Compute Resource* tab of the *Submit Job To* dialog box. If no resource attributes are specified by the user in this dialog box, then the **computeResources** argument will contain no resource attribute settings.

The **jobOptions** argument contains the environment variable settings for the job.

The **jobAttributes** argument contains job submission attributes which are not necessarily related to the compute resources allocated to the job. The job name and the requested job priority are included in this data structure.

The *Scheduler Options* tab of the *Submit Job To* dialog box allows the user to either specify additional job submission options or to specify all submission options, replacing the settings from the other *Submit Job To* dialog box controls.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The user-specified submission options are included in this data structure, as well as a boolean setting indicating whether the user specified options are in addition to the automatically generated options, or whether they replace the automatically generated submission options.

The **submissionUser** argument contains information about the client user (the user running the Desktop process). The job is submitted to the scheduler to run as this user.

The **jobParametersCB** argument is a pointer to an object that implements the **IJobParameters** interface. This interface allows the scheduler proxy library to get additional information about the job. Specifically, the **GetWorkingDirectory()** interface function returns the working directory to be used for the job.

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

int SubmitNonUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const NonUniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the nodes to use and the number of engines to run on each node are specified by the user. All other arguments are input arguments, as for the **SubmitUniformJob()** function. These input arguments are the same as for the **SubmitUniformJob()** function, except that the **computeResources** argument is a reference to a **NonUniformComputeResourcesStruct**, as described below:

The **computeResources** argument is a reference to an object of type **NonUniformComputeResourcesStruct**. This object contains a vector of pairs, where each pair consists of the name of a node in the cluster, and the number of engines to run on the node.

int PreviewUniformJob(std::string& msg, std::string& preview, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser, const IJobParameters* jobParametersCB):

This function is similar to the **SubmitUniformJob()** function, but instead of submitting the job, text representing how the job will be submitted is written to the preview argument. Typically the preview text includes the job submission command and the contents of the job script created for the job. For some schedulers, this content may not be meaningful, so the text returned could be different.

- On success, 0 is returned, and the job preview text is written to the preview argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

The other arguments are input arguments with the same meaning as for the **SubmitUniformJob()** function. The **submissionUser** argument is ignored for this function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Testing Scheduler Integration

One way to test these functions is to run the analysis for an Ansys EM product in batch mode. When running in batch mode, a batch log file is created in the same directory as the project file. The batch log file has the same base name is the project file, with an extension of ".log". For example, if the project file is TestProject123.aedt, then the batch file is TestProject123.log. The batch log file contains useful information about the analysis run.

See the product specific help for details on running the product in batch mode, and for the command line options to use for distributed analysis.

- Testing IsProductLaunchedInYourEnvironment
- Testing GetSchedulerDisplayName and GetThisJobID
- Testing GetTempDirectory
- Testing GetMachineListAvailableForDistribution
- Testing LaunchProcess
- Testing GetUseRsmForEngineLaunch

Testing IsProductLaunchedInYourEnvironment

This function should be tested first. If the Ansys EM application is not able to load and run this function, or if it returns false, then none of the other functions will be called. If the batch analysis is running in a scheduler environment, and this function returns true, then there will be an "info" message near the beginning of the batch log indicating that the analysis is running as a scheduler job. This message will include the scheduler display name returned by the function GetSchedulerDisplayName, and it will also include the job ID returned by the function GetThisJobID. If the batch analysis is not running in a scheduler environment, then none of the messages will include a scheduler display name or job ID.

If this message does not appear when running in a scheduler environment, ensure that the scheduler proxy library is named correctly, that it is built correctly, that it is installed in the correct directory, and that the function name prefix is the same is the library prefix converted to upper case.

Testing GetSchedulerDisplayName and GetThisJobID

As described above, when running a batch job in a scheduler environment, the scheduler display name and the job ID will appear in an "info" message near the beginning of the batch log. The values returned by these functions are copied to this message verbatim, so they can be directly compared to the expected values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Testing GetTempDirectory

Many schedulers create a temporary directory for each job and delete the directory after the job finishes. One way to verify that this function is working correctly is to determine the pathname that the scheduler uses for the temporary directory and to monitor the contents of the temp directory as the job is running. If the analysis engines write files to this directory as the job runs, then this function is working.

Testing GetMachineListAvailableForDistribution

This function is used for distributed analysis. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The list in the info message can be directly compared to the expected list of machines.

To verify that the machine list is constructed correctly for a variety of cases, it may be necessary to test several jobs with different resource requirements and verify that the machine list is correct in each case. For example, one may run batch analyses with the following resource requirements:

- One processor on one host
- Several processors on one host
- One processor on each of several hosts
- Several processors on each of several hosts

Testing LaunchProcess

This function is used to launch analysis engines in the case where the analysis is distributed across multiple hosts. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The batch log may also contain info messages when portions of the analysis distributed to different machines start or finish. These messages usually include the name of the host when the analysis ran or will run. One can verify that the analysis is actually running on the expected host or hosts using the Linux ps command or the Windows Task Manager.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In general, one analysis engine is started for each occurrence of each host in the list of machines available for distribution. For example, if the list of hosts is "hostA hostA hostA hostB hostB", then a total of 5 engines would be started, three on hostA and two on hostB. In some cases, an additional engine is started to perform the portion of the analysis which is not distributed; if this is the case, the non-distributed engine is idle during the portion of the analysis which is distributed. If this occurs in the case where the list of hosts is "hostA hostA hostA hostB hostB", then a total of 6 engines would be started, but at most 5 engines would be active at any given time. When each analysis engine is running, it may start additional child processes to do a portion of the analysis, but these are not counted as additional analysis engines because the parent of the sub-engine is inactive (waiting for the sub-engine results) when the sub-engine is active.

Testing should be sufficient to demonstrate that the scheduler proxy library can start multiple engine processes on the desktop host, and can also start multiple engine processes on other hosts.

Testing GetUseRsmForEngineLaunch

In most cases, this function will not be implemented or tested. If this function is implemented and returns true, then the Ansys Electromagnetics desktop application will not start the analysis engines using the LaunchProcess function directly. Instead, the Ansys Electromagnetics desktop application will start one AnsoftRSMService process on each host using the LaunchProcess function, and the engine processes will be started by these AnsoftRSMService processes. One may check for these processes using the Linux ps command or the Windows Task Manager. One AnsoftRSMService process should run on each host. These processes will be named ansoftrsmservice.exe or AnsoftRSMService.exe. These processes will be started on each host before any analysis engine is started on the host, and will remain running until the job is complete.

Troubleshooting Custom Scheduler Integration

- None of the Proxy Functions are Called
- Troubleshooting IsProductLaunchedInYourEnvironment Function
- Troubleshooting GetSchedulerDisplayName
- Troubleshooting GetThisJobID
- Troubleshooting GetTempDirectory
- Troubleshooting GetMachineListAvailableForDistribution
- Troubleshooting LaunchProcess
- Troubleshooting GetUseRsmForEngineLaunch

None of the Proxy Functions are Called

There are several problems which could result in none of the proxy functions being called.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansys Electronics Desktop installation directory.

The scheduler proxy library name must match "*_scheduler.dll" on Windows and "lib*_ scheduler.so" on Linux. If the library name does not match this format, then the library will not be loaded. In addition, the function name prefix must be the same as the library name prefix converted to upper case. For example, if the library name prefix is "abc", then the function name prefix is "ABC". In this example, the library name is "abc_scheduler.dll" on Windows, and "libabc_scheduler.so" on Linux. In this example, the full name of the IsProductLaunchedInYourEnvironment function is ABC_IsProductLaunchedInYourEnvironment on Windows and Linux, and it must have extern "C" linkage.

Verify that the compile and link flags follow the guidelines in the section "Build Information", above. Incorrect compile or link flags may prevent the library from being loaded by the Ansys Electromagnetics product.

If there is a problem with calling the IsProductLaunchedInYourEnvironment function, then none of the other functions will be called. The other functions are only called if the IsProductLaunchedInYourEnvironment function is successfully called and returns true.

Troubleshooting IsProductLaunchedInYourEnvironment Function

Verify that the conditions specified in the section "None of the Proxy Functions are Called" are met.

Verify that this function returns true when called in an environment running under the scheduler, and that it returns false when called in an environment not running under the scheduler.

Troubleshooting GetSchedulerDisplayName

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the scheduler display name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the display name, including the terminal null byte, then the display name is copied to the buffer and true is returned.

Troubleshooting GetThisJobID

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verify that the job ID is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the job ID, including the terminal null byte, then the job ID is copied to the buffer and true is returned.

Troubleshooting GetTempDirectory

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the temporary directory name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the temporary directory pathname, including the terminal null byte, then the temporary directory pathname is copied to the buffer and true is returned.

Troubleshooting GetMachineListAvailableForDistribution

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the list of hosts is a valid ASCII string containing a space separated list of host names. A host name will appear in the list a number of times equal to the number of processors or cores available to the job on that host.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the list of hosts, including the terminal null byte, then the list of hosts is copied to the buffer and true is returned.

Troubleshooting LaunchProcess

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The hostName argument is a host name from the list returned by the GetMachineListAvailableForDistribution function. Verify that the LaunchProcess function can accept host names in the format returned by the GetMachineListAvailableForDistribution function.

The exePathName argument is the pathname of the analysis engine executable to be started. This pathname may contain spaces or other characters special to the shell. Ensure that the LaunchProcess function is able to handle such cases.

The arg1 and arg2 arguments may contain newlines, tabs, single quotes, spaces, dollar signs, and other characters which may be special to the shell. Ensure that the LaunchProcess function is able to handle such cases. If needed, the newline characters may be replaced by other whitespace characters. One or both of these arguments could also be an empty string; verify that the empty string is correctly passed to the engine process command line.

If a scheduler command is used to launch the engine process on a remote machine, the engine command line may be processed by the shell twice, once when the scheduler command is processed by the shell, and again when the engine command is processed by the shell. In such cases, the quoting of characters special to the shell will need to be take these two passes through the shell into account. In some implementations, it may be necessary or convenient to use different approaches for launching engine processes on the local machine and on remote machines; if this is done, verify that the approach used to determine whether the hostName argument represents the local machine is correct.

Troubleshooting GetUseRsmForEngineLaunch

In most cases, this function will not be implemented. If it is implemented, then follow the suggestions below.

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

If the RSM should be used for launching engines, verify that this function returns true. Otherwise, verify that this function returns false.

Using an IronPython Program for Integration with a Scheduler

This section describes how to create an IronPython program for integration with a scheduler. Each such program is used for a single specific scheduler environment. If the program is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys EM tools.

Installation Details

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The IronPython program must be installed in the schedulers subdirectory of the Ansys EM installation directory. For example, if the installation directory is "C:\Program Files\ANSYS Inc\v251\AnsysEM", then the IronPython program must be installed in directory C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers".

The program file extension must be ".py". Select the program name so that it does not conflict with other IronPython programs in the same directory. If the Operating System or file system treat file names in a case sensitive manner, the file extension ".py" must be lower case.

Python Programming Notes

The scheduler program will be run in the IronPython environment both on Microsoft Windows and on Linux. There are some differences between IronPython and CPython. The version of IronPython in use is 2.7.0.40

Implementation Details

The program must contain the following:

Import the ISchedulerPluginExtension interface as follows:

```
from Ansys.Ansoft.SchedulerPluginDotNet import
ISchedulerPluginExtension
```

Define a class which implements the ISchedulerPluginExtension interface. In this document, this class is named SamplePluginExtension, but any class name may be used. The class member functions are described in the next section. The class definition will look similar to the following:

```
class SamplePluginExtension(ISchedulerPluginExtension):

def GetName(self):
return "SamplePluginExtension"

def GetDescription(self):
return "Example python script plugin extension"
. . .
```

Include the following line in the program so that the class that you have defined, SamplePluginExtension, is loaded by the infrastructure:

```
ExtensionRegistrar.RegisterPluginExtension(SamplePluginExtension())
```

The infrastructure will make the EntensionRegistrar object available in the environment where the program is loaded.

Each of the functions to be implemented in the SamplePluginExtension class is described below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetName
- GetDescription
- IsProductLaunchedInYourEnvironment
- GetSchedulerDisplayName
- GetThisJobID
- GetUseRsmForEngineLaunch
- GetTempDirectory
- GetMessageStringToRegisterForSigTerm
- GetMachineListAvailableForDistribution
- LaunchProcess

GetName [IronPython]

Purpose

Return a short string containing the name of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetName(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the name of the plugin extension.

> **Note:**
>
> The plugin extension name is generally a fixed string.

GetDescription [IronPython]

Purpose

Return a string containing the description of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetDescription(self)
```

Arguments (excluding self)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

None.

Return Value

Returns a string containing the description of the plugin extension.

> **Note:**
>
> The plugin extension description is generally a fixed string.

IsProductLaunchedInYourEnvironment [IronPython]

Purpose

Determine if the program is running in the context of the scheduler for which this program was written.

Signature

```
IsProductLaunchedInYourEnvironment(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if the current process is running as a job of the scheduler. Otherwise, False is returned.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetSchedulerDisplayName [IronPython]

Purpose

Get a string identifying the scheduler associated with the current plugin extension. This string is used to identify the scheduler.

Signature

```
GetSchedulerDisplayName(self)
```

Arguments (excluding self)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

None.

Return Value

Returns a string containing the description of the scheduler for which this plugin extension was written.

> **Note:**
>
> The scheduler display name is generally a fixed string.

GetThisJobID [IronPython]

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
GetThisJobID(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the Job ID for the current job.

> **Note:**
>
> For many schedulers, the job ID may be obtained from the value of an environment variable.

GetUseRsmForEngineLaunch [IronPython]

Purpose

For some schedulers, it may be desirable for the AnsoftRSM program to launch the engine processes instead of using the scheduler plugin extension directly. For example, if the plugin extension is limited to launching one process per host, then the plugin extension may be used to launch one AnsoftRSM executable per host, and the AnsoftRSM executable will launch all of the engine processes.

If AnsoftRSM should be used to launch engine processes for this scheduler, then this function shall return True.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If AnsoftRSM should not be used to launch engine processes for this scheduler, then this function shall return False.

Signature

```
GetUseRsmForEngineLaunch(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if AnsoftRSM should be used to launch engine processes for this scheduler. Returns False if the plugin extension should be used to directly launch engine processes for this scheduler.

> **Note:**
>
> If this function returns True, then the plugin extension will directly launch only one process on each host.

GetTempDirectory [IronPython]

Purpose

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
GetTempDirectory(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the pathname of the temporary directory provided by the scheduler for the current job. Returns an empty string if no temporary directory is provided by the scheduler for the current job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If the return value is an empty string, then the temporary directory specified on the command line or in the registry will be used.

GetMessageStringToRegisterForSigTerm [IronPython]

Purpose

Obsolete. This function should return an empty string.

Signature

```
GetMessageStringToRegisterForSigTerm(self)
```

Arguments (excluding self)

None.

Return Value

Returns an empty string.

> **Note:**
>
> This function should always return an empty string.

GetMachineListAvailableForDistribution [IronPython]

Purpose

Get the names of the hosts allocated to the current job. A host name will appear in the output string multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host name appears in the string is equal to the number of processors or cores of the host that are allocated to the current job. The output is a text string containing the host names separated by space characters.

Signature

```
GetMachineListAvailableForDistribution(self)
```

Arguments (excluding self)

None.

Return Value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A string containing the names of the hosts allocated to the job, separated by space characterss. The number of times the host appears in the string is equal to the number of processors or cores of the host that are allocated to the current job.

> **Note:**
>
> The hostnames in the string provided by this function shall be used in calls to LaunchProcess(). The host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

LaunchProcess [IronPython]

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names provided by the GetMachineListAvailableForDistribution function. See the GetMachineListAvailableForDistribution section above. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
LaunchProcess(self, hostName, exePathName, arg1, arg2)
```

Arguments (excluding self)

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

arg2: The second argument of the analysis engine command line.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain spaces or other characters that are interpreted by the command processor, such as backslash (\), single quote (') or double quote (") characters, or dollar signs ($). If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, and that command is processed by a command shell, then the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.
- The command line arguments arg1 and arg2 may be empty strings. These arguments must be preserved, even if they are empty strings. In some versions of the IronPython subprocess module, empty argument strings are discarded, resulting in an incorrect number of command line arguments. A workaround for this issue is to replace an empty string argument by a string consisting of a single space character.

Ansys Cloud Direct Support for HPC Job Management

Ansys Electromagnetics Desktop supports the Ansys Cloud Direct service that allows quick and easy access to cloud-based HPC resources. The service is accessible through both the Simulation Ribbon and **Tools** > **Job Management**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> For this release, you can submit HFSS, HFSS 3D Layout, Icepak, Maxwell 3D, Maxwell 2D, Mechanical, Q3D Extractor, and 2D Extractor jobs, monitor their progress, and download and post process the results. Other solvers are either in Beta or are not yet supported for Ansys Cloud Direct.

Uploads to and downloads from Ansys Cloud Direct use archives to improve speed. The Project browser shows both .aedt and .aedtz (archive) files. If you submit an unarchived project, an archive will be automatically created for upload. When the job completes, the results are packaged into an archive. When the download is initiated, the repackaged archive file is transferred to the download folder along with the other job-related files, such as the log.

For single setups, you can use multi-step submission. This separates the resource specification and reporting as appropriate for each stage of the solve.

Prerequisites for Using Ansys Cloud Direct with Ansys Electronics Desktop

- You will need an "Ansys Cloud Compute Essentials" subscription associated with your Ansys Account. The Ansys Account is an extension of the Ansys Discovery account that includes those products and services that require sign-in, such as the AppStore, Cloud Compute and others. Contact your account manager or service representative to obtain a subscription invitation.
- You must select Ansys Cloud as your Job Management Scheduler, and, if necessary, log in to the Ansys Cloud Service using the Ansys Discovery Account and associated password.

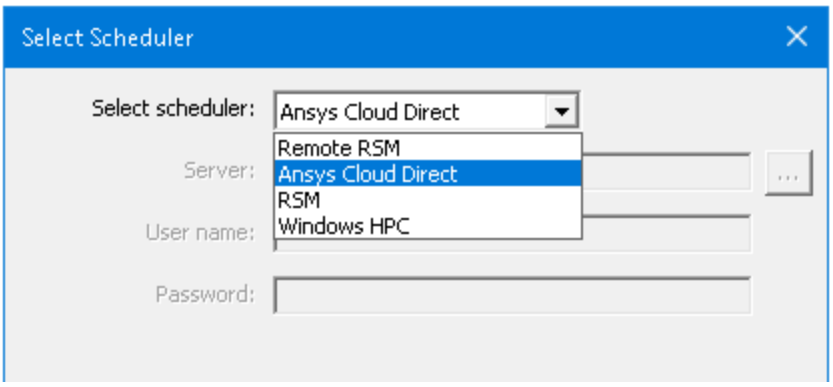
Job Management UI for Ansys Cloud Direct

You can use Ansys Electronics Desktop to submit batch jobs to Ansys Cloud Direct and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select Ansys Cloud Direct as the scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Log in** to launch a login window for Ansys Cloud Direct.

   After a successful login, you may not need to log in again for several days if you do not log out.

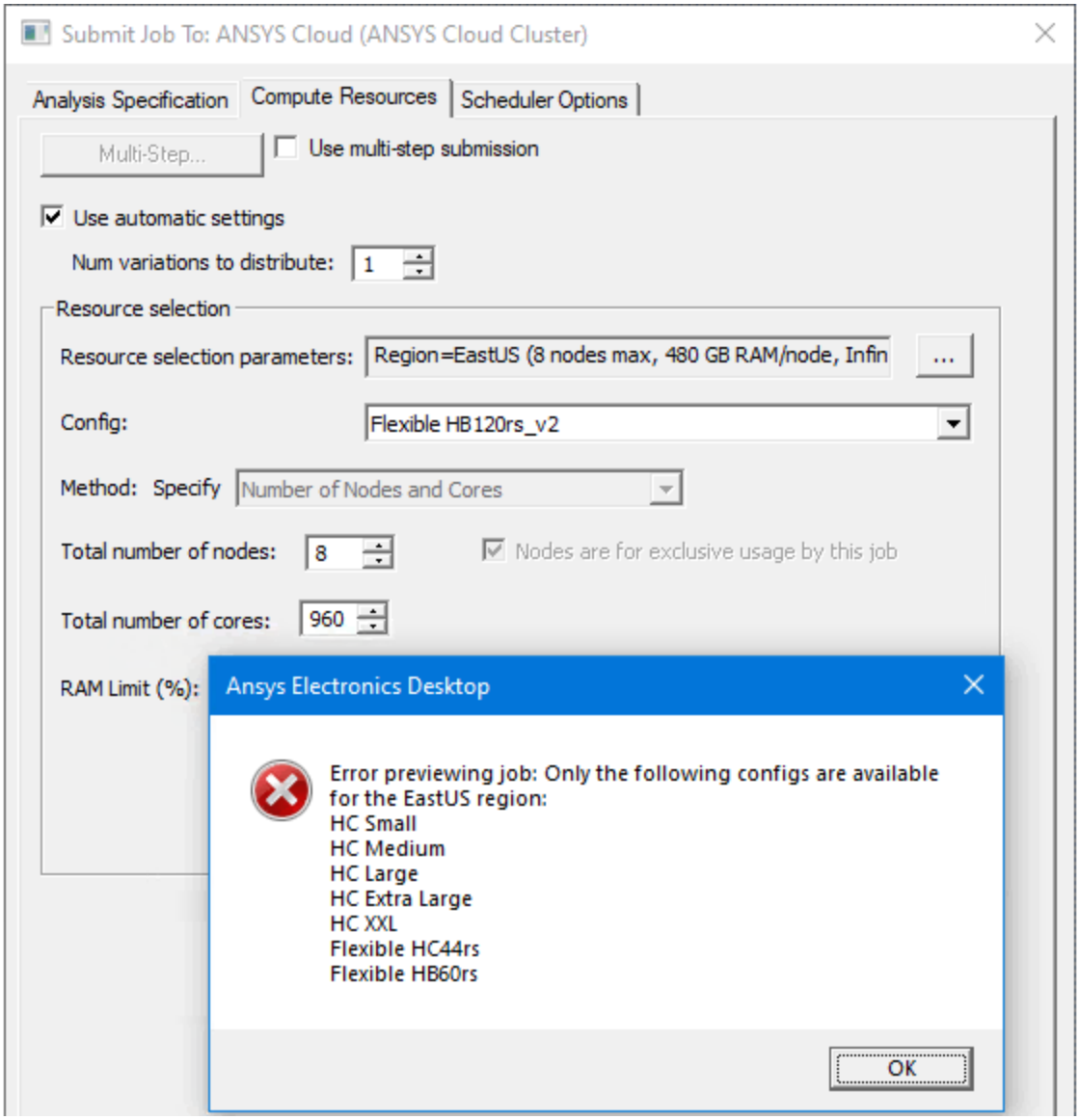
   If you wish to log out, click **Log out** from the **Select Scheduler** window.

3. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to Ansys Cloud Direct.

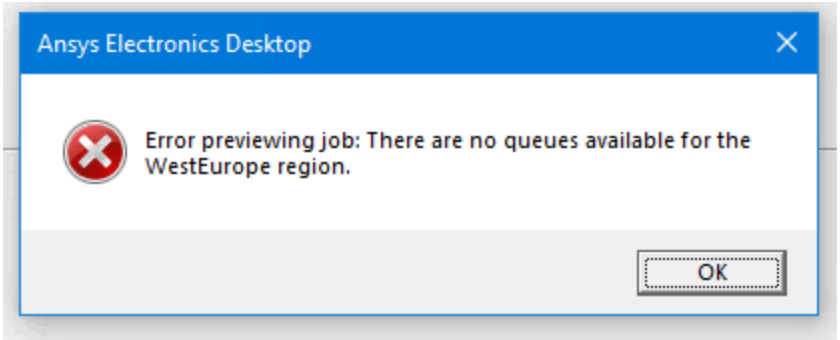
   Ansys Cloud Direct-specific Settings:

   - On the **Compute Resources** tab, **Resource Selection Parameters** will allow you to select a **Region** (for example, EastUS, JapanEast, WestEurope). The **Config** drop-down menu does not display configurations that are not available in any of the selectable regions. That is, if the regions are EastUS, WestEurope, and SouthwestUS, and none of these regions have the smallhc-based queues, then the "HC Small" configuration does not appear. If you select a resource that is not available in the specific region you select, an error messages shows the available choices for that region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

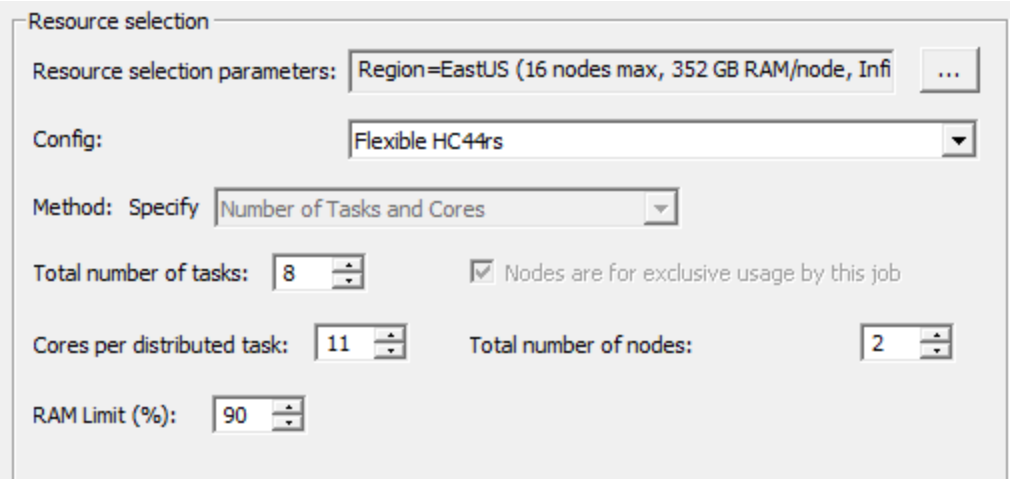


If no queues are available, an error message provides that message:



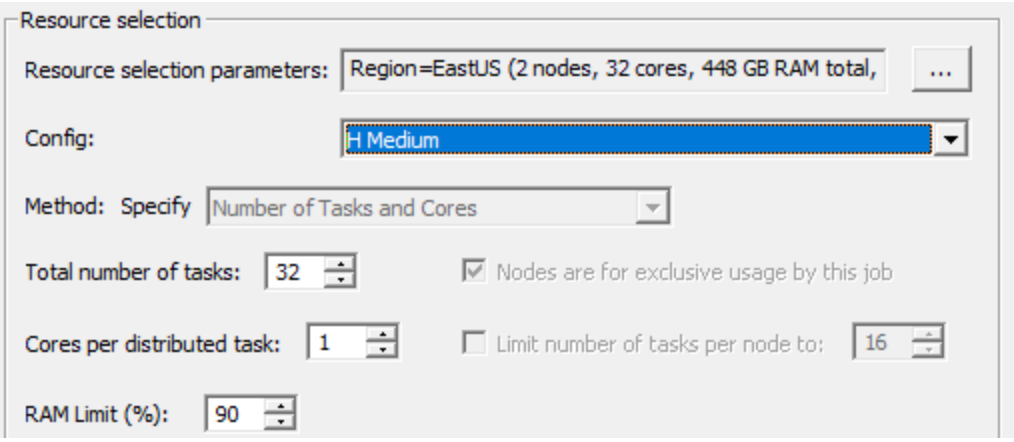
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Compute Resources** tab, select a **Config** size and style (Flexible or fixed) based on the following:
  - **Small:** 16 cores, 1 node, 224 GB RAM total
  - **Medium:** 32 cores, 2 nodes, 448 GB RAM total, Infiniband
  - **Large:** 128 cores, 8 nodes, 1792 GB RAM total, Infiniband
  - **Extra Large:** 256 cores, 16 nodes, 3584 GB RAM total, Infiniband
  - **XXL:** Even larger.
  - **Flexible**: Flexible queues permit the following resource selection methods: Automatic Number of Nodes and Cores, Manual Number of Tasks and Cores. If you select a Flexible queue for manual Number of Tasks and Cores, you will see a new item in **Resource Selection** for "Total number of nodes" replacing the "Limit number of tasks per node to" option that appears for fixed queues.



In comparison, if you selected a fixed configuration, the **Resource selection** panel appears like this:

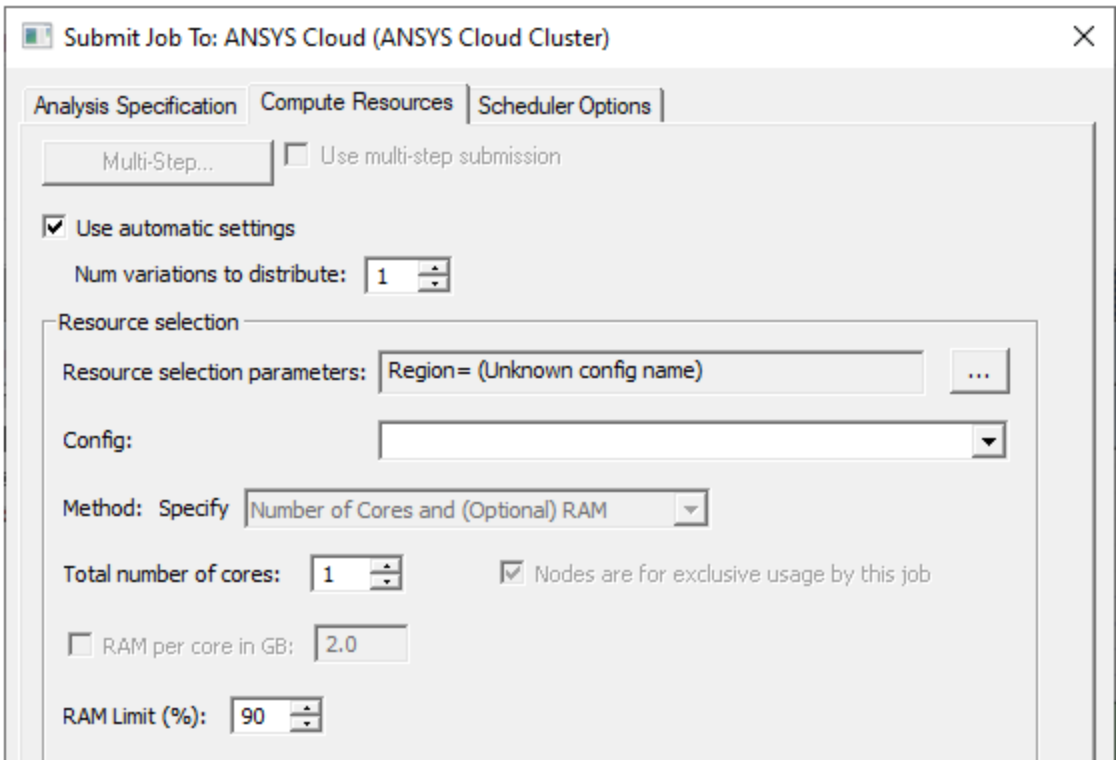
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



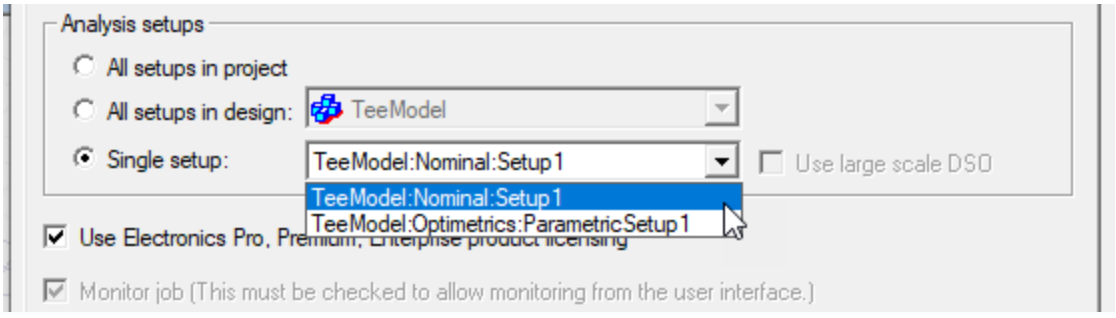
> **Important:**
>
> Attempts to use more for submission will result in error messages.

- On the **Compute Resources** tab, if you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

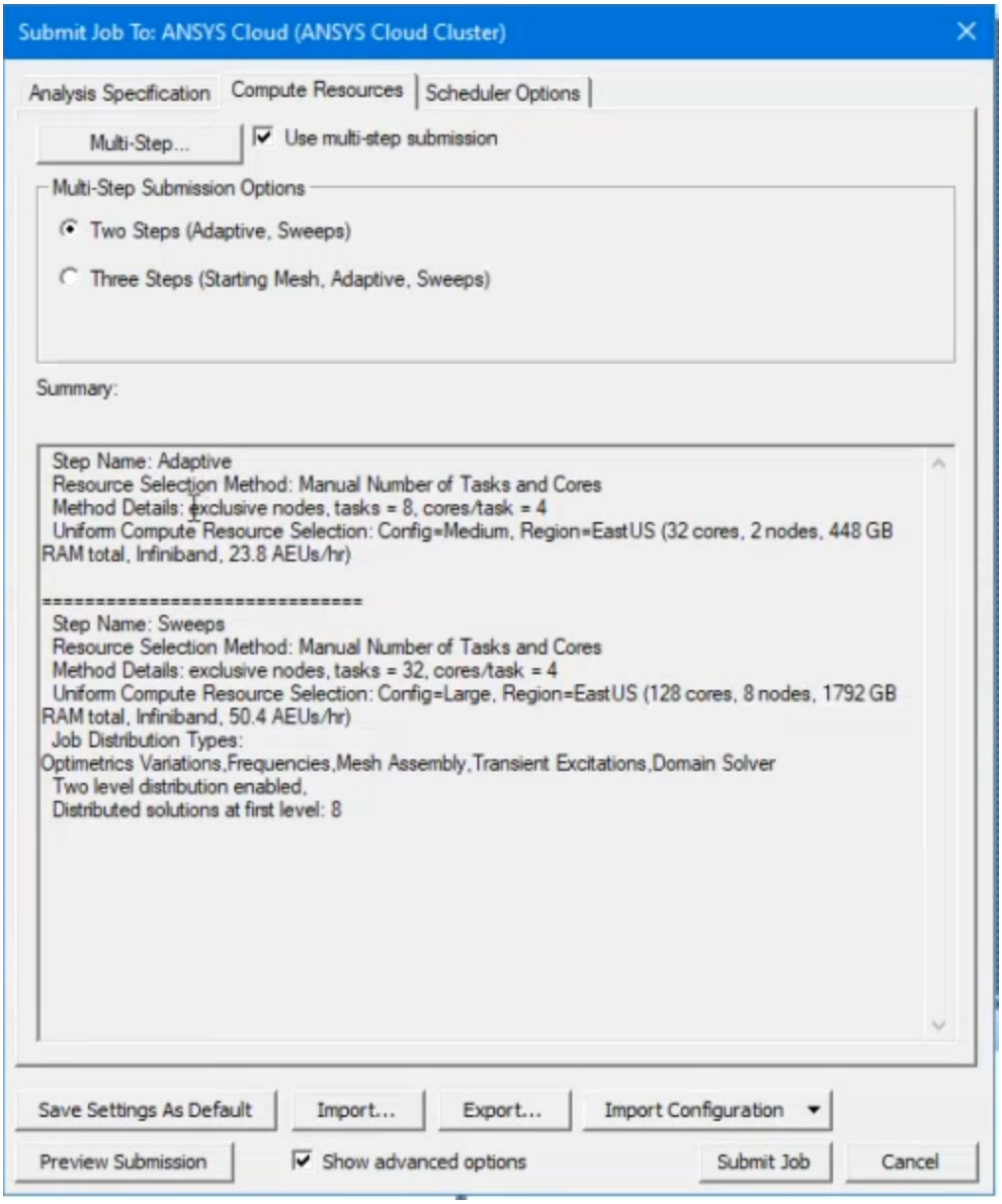


- In response to a set of minimal constraints, Ansys Cloud Direct scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines with two processors per engine and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Click **Preview Submission** to see the number of resources assigned, and that the scheduler-generated code includes an MPI specification.
- To use **Multi-Step Submission** on Ansys Cloud Direct, you must specify a single setup on the **Analysis Specification** tab.



This enables **Multi-Step** options on the **Compute Resources** tab.

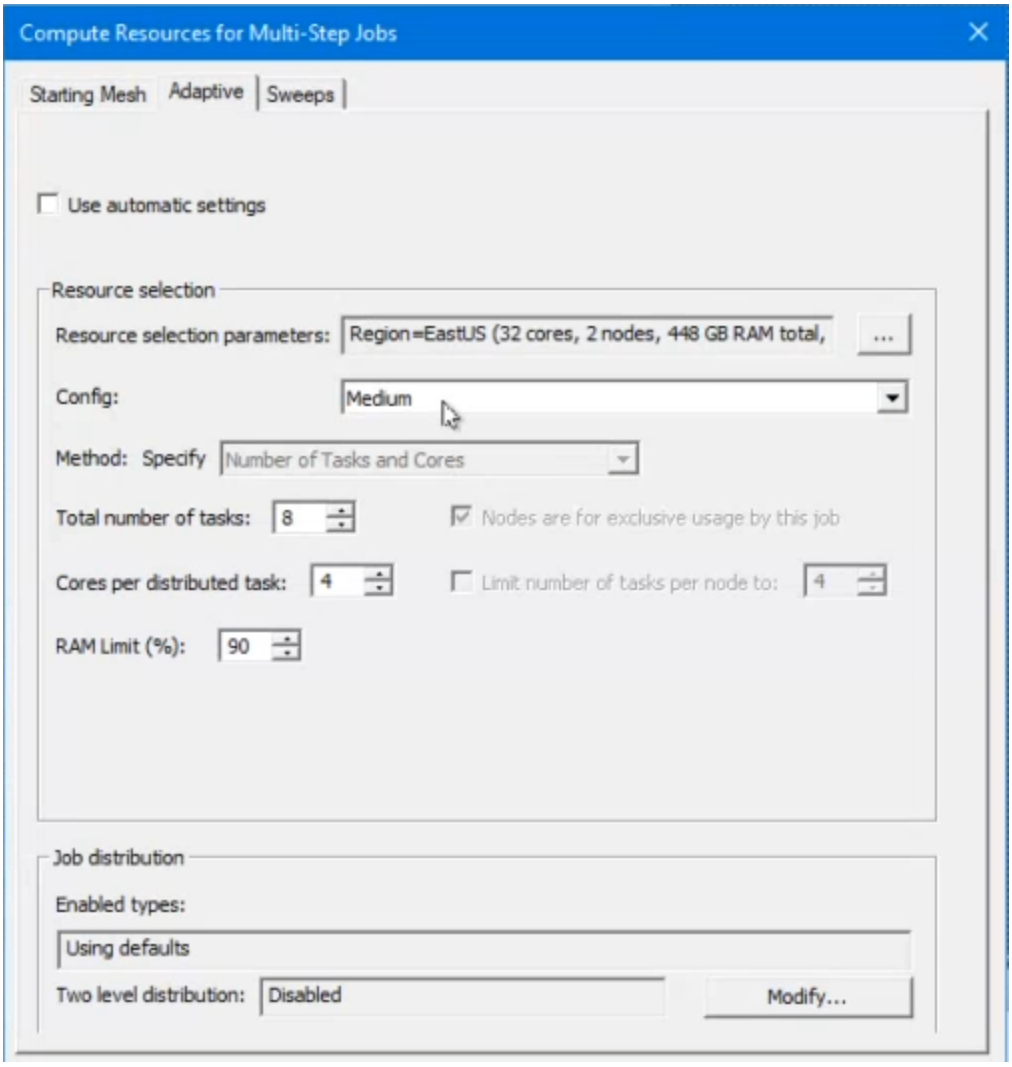
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click the **Multi-Step** button to open the **Compute Resources for Multi-Step Jobs** dialog box. It has tabs for **Starting Mesh**, **Adaptive**, and **Sweeps**.

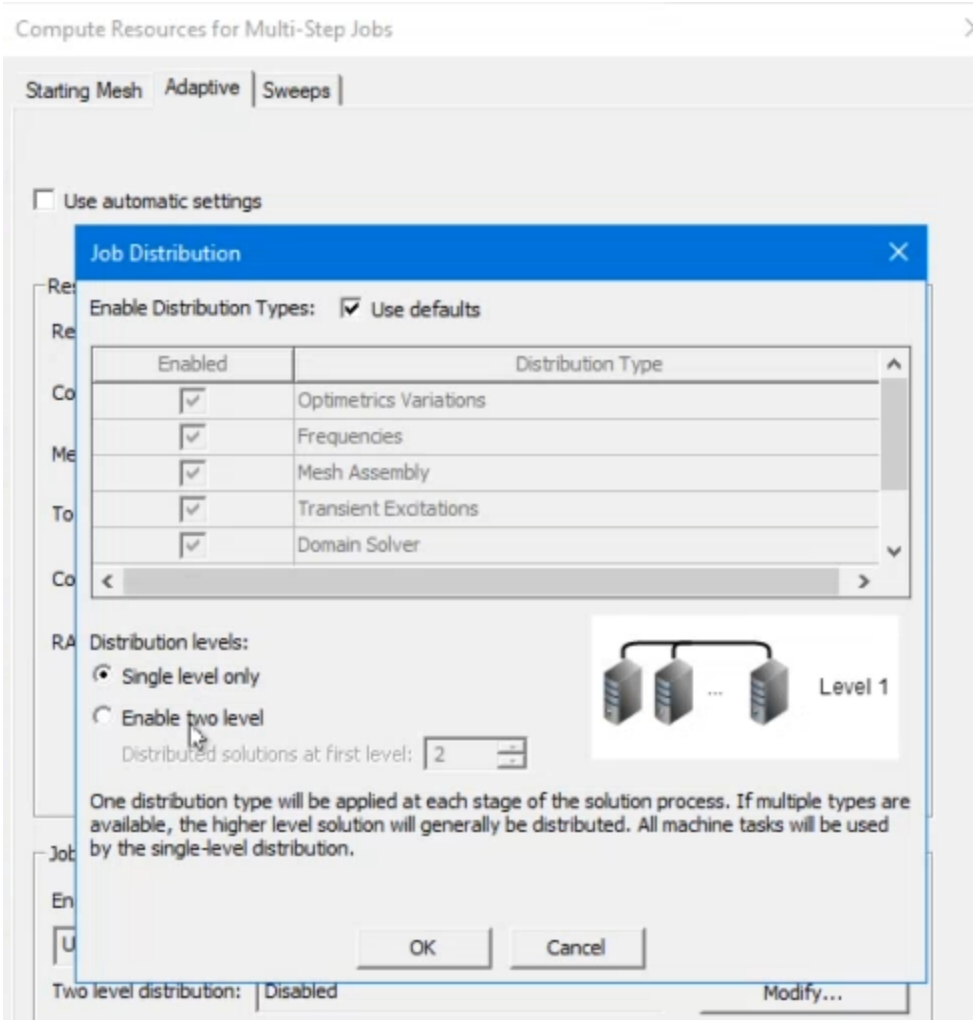
On the **Adaptive** tab, you can **Use automatic settings**, or specify **Resource Selection**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



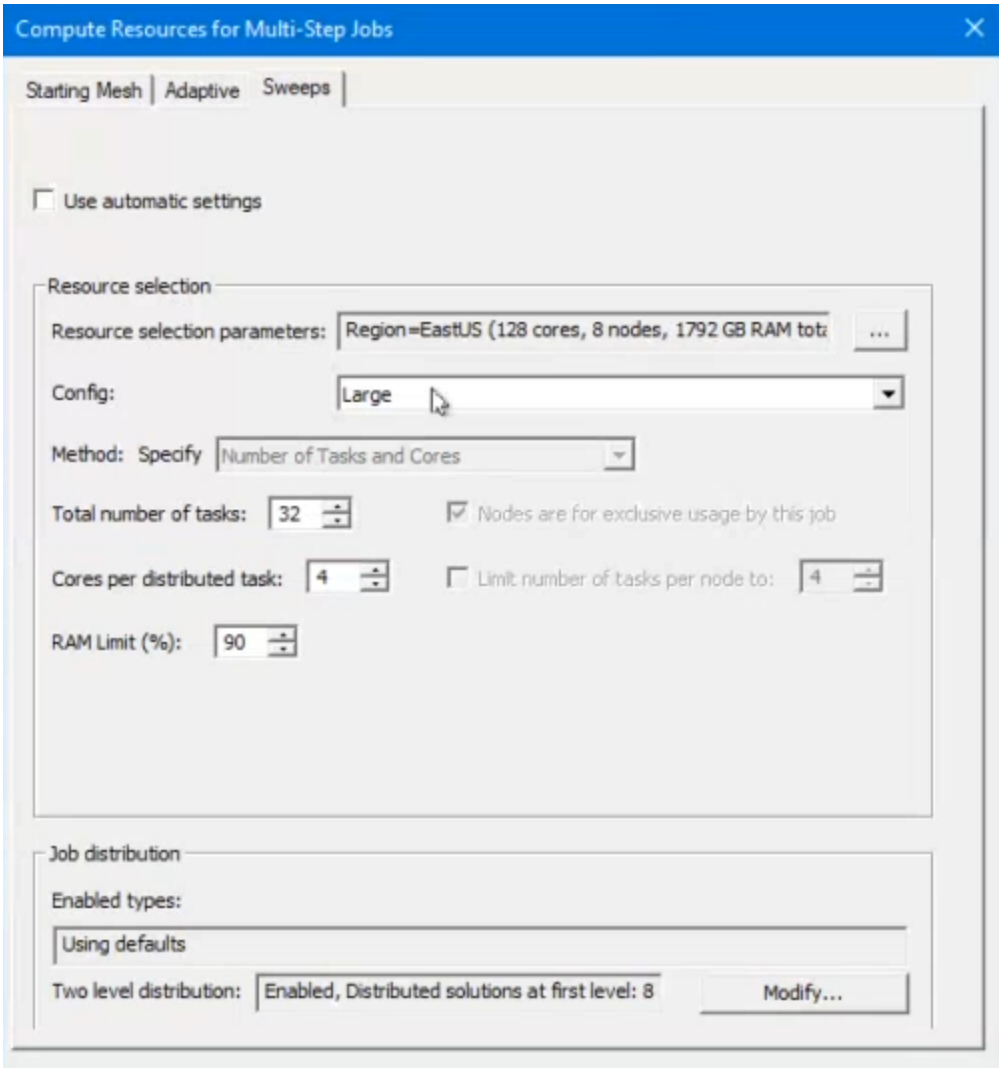
Click the **Modify...** button on the **Adaptive** tab to open the **Job Distribution** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Sweeps** tab lets you specify resources or **Use automatic settings**.

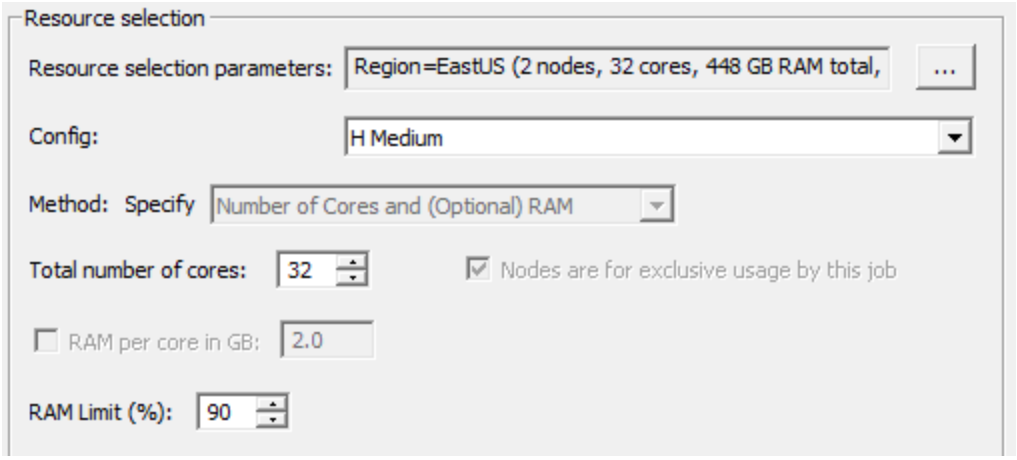
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



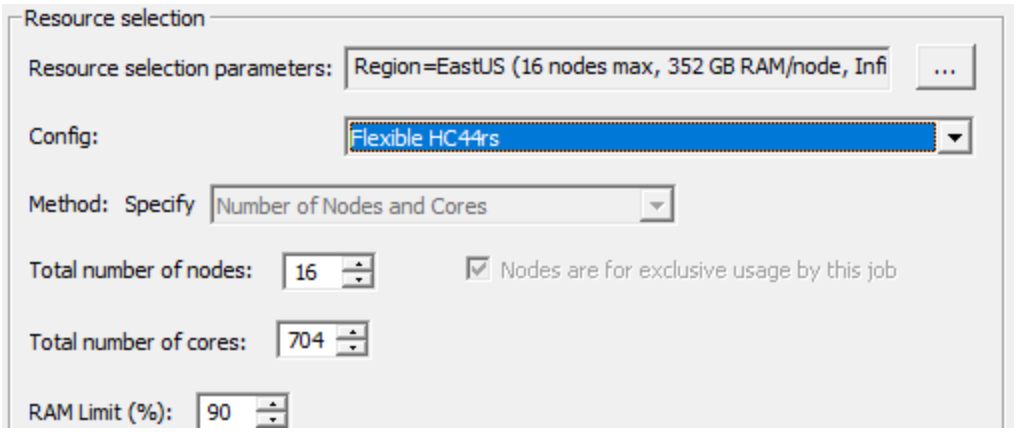
The **Sweeps** tab includes an independent **Modify...** button to specify **Job Distribution**.

Select automatic settings, or specify different resources for each step. When a job is finished, that is, on the last step, reports update and traces are extracted into CSV files. If select fixed queue for automatic sections, the **Resource selection** panel only allows cores and has RAM with RAM disabled:

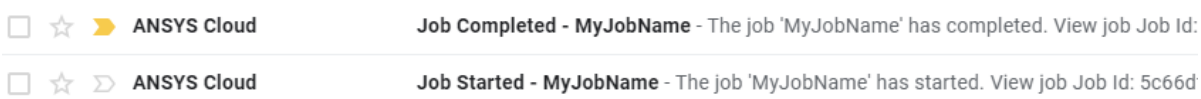
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you select flexible queue for automatic settings, The **Resource selection** panel only allows nodes and cores for a flexible queue:

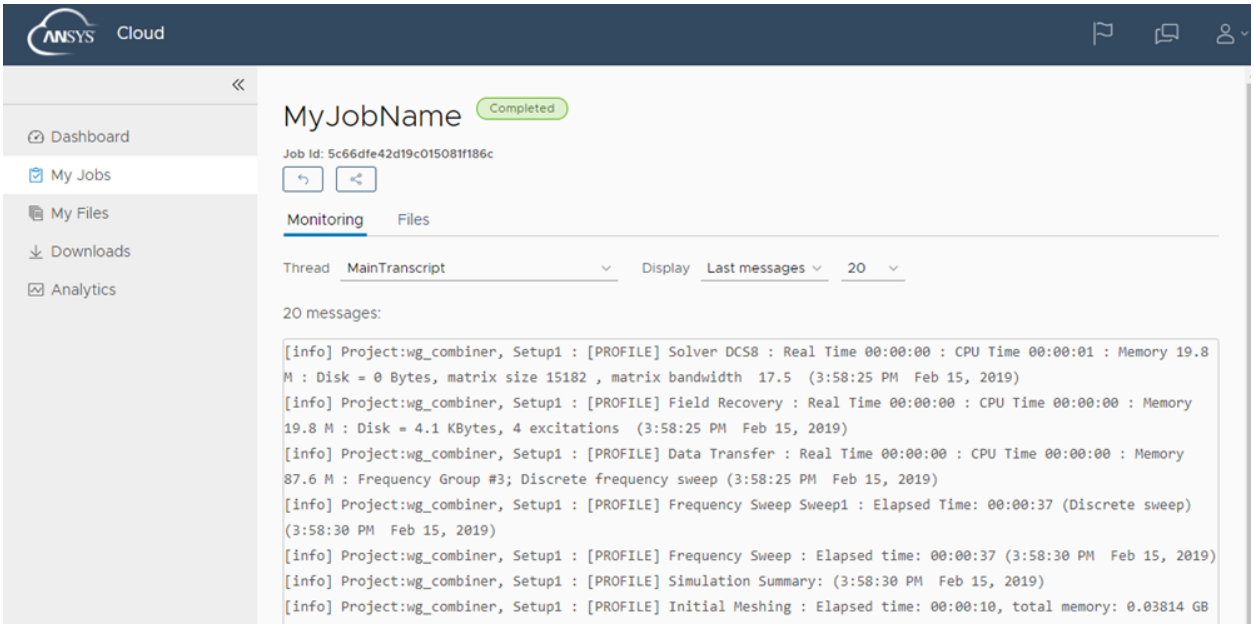


4. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.
5. You will receive an email when the job has started, and another when it has finished.



6. Use the link in the email (or the **Portal** button on the job monitoring window) to launch the Ansys Cloud Portal and view detailed results:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variable Settings for Ansys Cloud Direct

When submitting jobs to Ansys Cloud Direct from Electronics Desktop, certain environment variables and batchoption settings must be made to ensure proper operation on the cloud. These settings are done automatically through the job submission window.

These environment variable settings will override any attempts from the user to set the same environment variables:

- ANS_NODEPCHECK=1
- ANSYSEM_FIX_REVERSE_LOOKUP=1
- ANSYSEM_DESKTOP_SUBNET_FROM_COMENGINE_ADDR=1

The environment variable ANSYSEM_ENV_VARS_TO_PASS contains a semicolon-separated list of filters for environment variables to be passed from ansysedt.exe to COM engines. You are allowed to set this environment variable. After being prepared for the job it will have the following filters set at minimum (with the user-requested filters following these):

```
ANSOFT_*;ANS_*;ANSYSEM_*;ANSYSLMD_LICENSE_*;I_MPI*
```

Troubleshooting and Debugging Environment Variables

To keep downloaded files that help with troubleshooting, enable the environment variable ANSYS_EM_PRESERVE_DOWNLOAD_FILES.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Debug logs (and other files generated in the working directory) from remote nodes are automatically collected.

Option Settings for Ansys Cloud Direct

The following option is automatically added for Ansys Cloud Direct jobs:

- -autoextract
- For single step jobs it will include "-autoextract reports".
- For multi-step jobs, the last step will include "-autoextract reports" but the first steps will only include "-autoextract" (no reports). Reports are only extracted on the last step.

See: Running Electronics Desktop from a Command Line.

Automatic Batchoption Settings

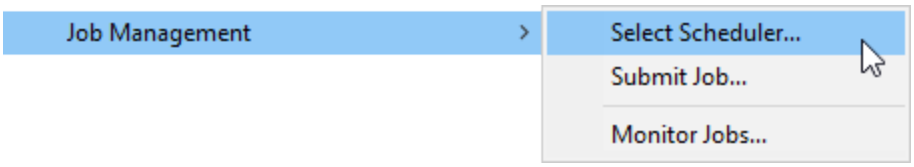
The license type batch option ("HPCLicenseType") is forced to be "Pack" because Cloud is configured to only work with pack licensing. The following MPI batchoptions are automatically set as "Intel" because Ansys Cloud Direct is configured to only work with Intel MPI.

- HFSS 3D Layout Design/MPIVendor
- HFSS/MPIVendor
- Icepak/MPIVendor
- Maxwell 2D/MPIVendor
- Maxwell 3D/MPIVendor
- Q3D Extractor/MPIVendor

Selecting a Scheduler

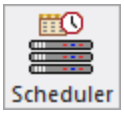
Access the **Select Scheduler** window one of three ways:

- Click **Tools** > **Job Management** > **Select Scheduler...**



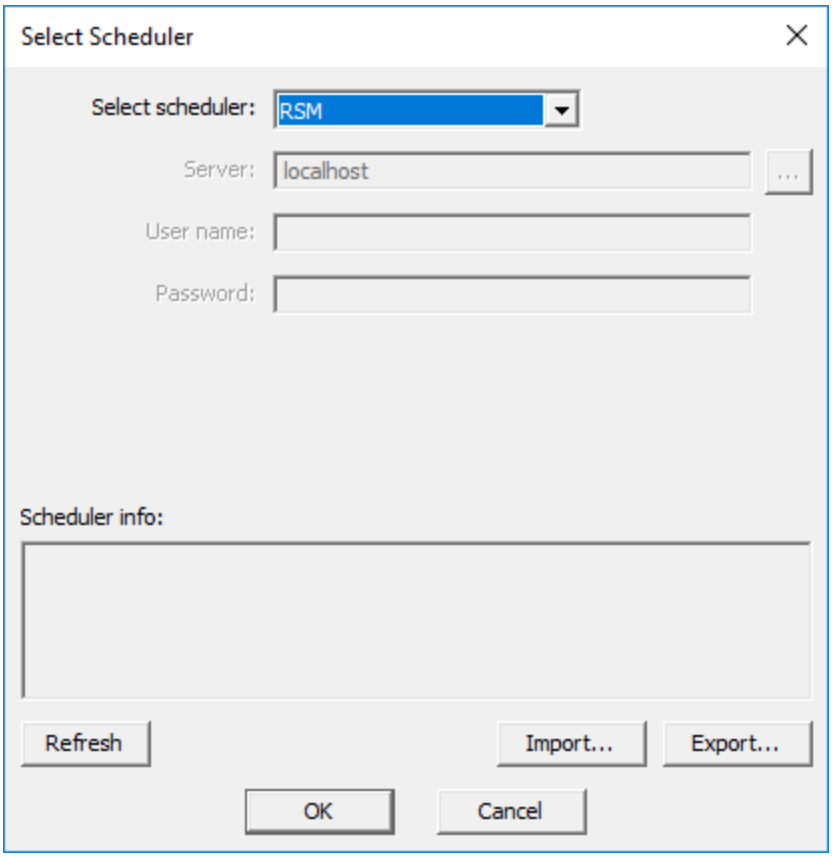
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **Simulation** tab and click the **Scheduler** icon.



- From a command window, use the -showselectscheduler command.

The **Select Scheduler** window appears:



From the **Select Scheduler** window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Use the **Select scheduler** drop-down menu to select a scheduler.

   > **Note:**
   >
   > If you select a scheduler that is unsupported in your environment, you will receive a warning message.
   >
   > See HPC Integration for a list of currently supported schedulers.

2. If applicable for the scheduler type, enter server and user information. If Ansys Cloud Direct or Ansys Cloud Burst Compute is selected then the “Login” button will open the default browser and prompt for login.

   Information about the selected scheduler appears in the **Scheduler info** field.

3. Click **OK** to complete your selection.

Submitting a Job

Ansys Electromagnetics Desktop supports its own Remote Simulation Management (RSM) and Ansys Cloud Direct along with other High Performance Computing (HPC) software management programs (See: HPC Integration). The **Simulation** tab of the ribbon includes icons for setting HPC Options, creating and selecting analysis configuration, selecting the scheduler, submitting jobs, and monitoring jobs.

There are two ways that the GUI may be used to submit jobs. The first (and most common) mode requires that the Desktop (UI) process run on a host which is also a submission host for the job scheduler. This mode is called local mode or working mode.

The second mode is only supported on Linux in the Ansys Electromagnetics Suite. In the second mode, an administrator configures the RSM Service to act as an interface to the job scheduler, and starts the RSM Service on a submission host for the cluster. The user runs the Desktop (UI) process on another host (which may be called the post-processing host). To submit a job, the user specifies the host where the RSM Service is running, and the Desktop process connects to the RSM Service over the network to submit the job. In this mode, some configuration is required, and the RSM Service typically must run as a privileged user (for example, root), so that it can launch processes as any user. This mode is useful for cases in which the submission hosts are not able to run graphical processes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:

ANSYSEM_SKIP_NODE_ONLINE_CHECK

Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

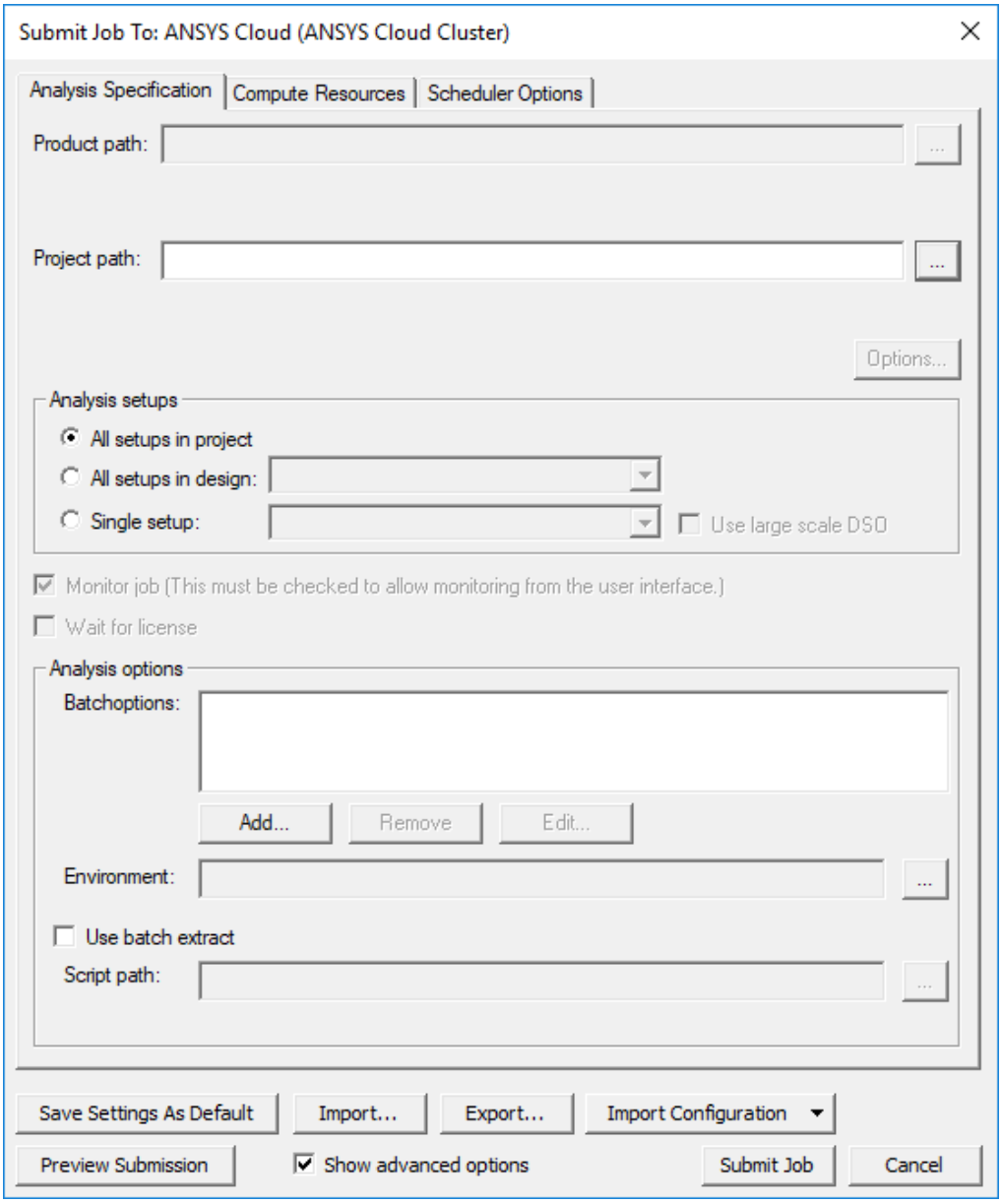
Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (i.e. if not set, Ansys Electronics Desktop requires nodes to be online for submission).

To submit a job:

1. Prepare your design.
2. Open the **Submit Job To** window one of the following ways:
   - On the **Simulation** tab, click the **Submit** icon.
   - Select **Tools** > **Job Management** > **Submit Job**.
   - Select **Mechanical** > **Submit Job**.
   - Right-click a solution setup and select **Submit Job**. *In this case, the information in the **Submit Job To** window will be pre-populated from the setup*.
   - From a Command window, use the -showsubmitjob command.

   The **Submit Job To** window appears. The window header indicates your selected scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains up to three tabs:

- **Analysis Specification** – allows you to specify the product path, project name, setup and analysis options, batchoptions, and environment variables (for advanced users).
- **Compute Resources** – allows you to specify whether to use automatic settings, and to set resource selection and job distribution parameters.
- **Scheduler Options** – allows you to specify the job name and priority. *This tab does not appear if you have selected local RSM as the scheduler*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

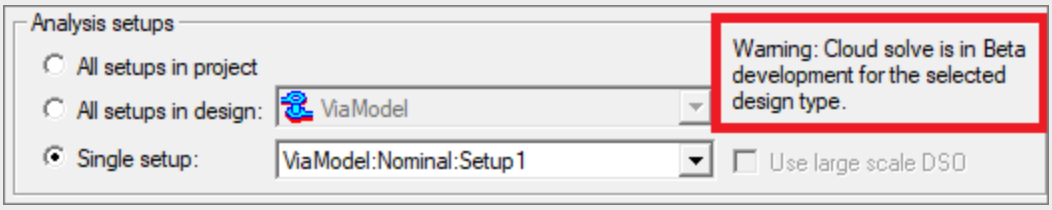
3. On the **Analysis Specification** tab, specify your desired options.

   The **Product path** and **Project** fields support mapped drives. Click the ellipses (**...**) to select files.

   The project can be an archive. The project file pathname must be a UNC path that is accessible from each compute host used for Ansys Electromagnetics jobs. After clicking the **Project path** field's ellipsis button (**...**), a check box allows you to **Use converted UNC path if mapped drive specified**. If you select a project or product on a mapped drive, and check the option, the converted UNC path equivalent to the mapped drive pathname is used.

   Note:

   If you select Ansys Cloud as your scheduler for a design type that is in Beta, you will receive a warning:

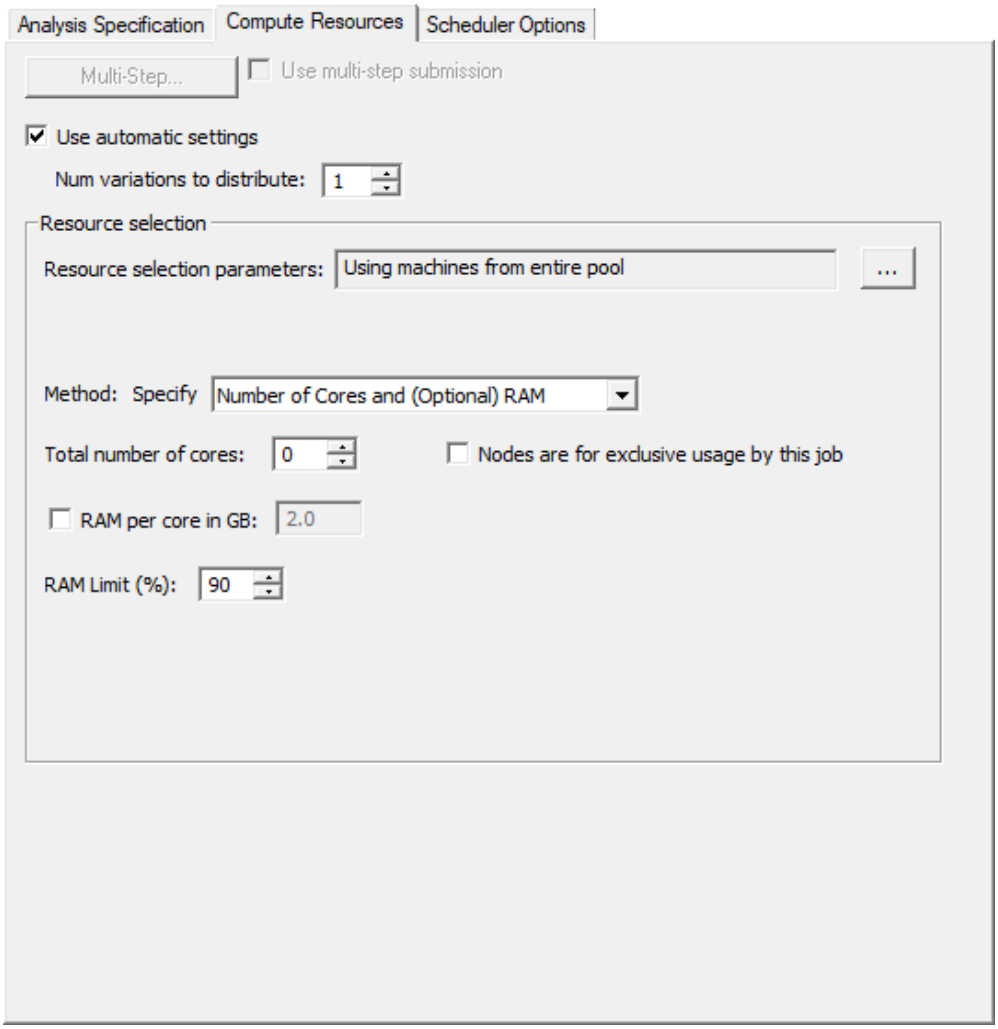
   

   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See: Large Scale DSO for Parametric Analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Compute Resources** tab.



Options may vary slightly depending on your selected scheduler.

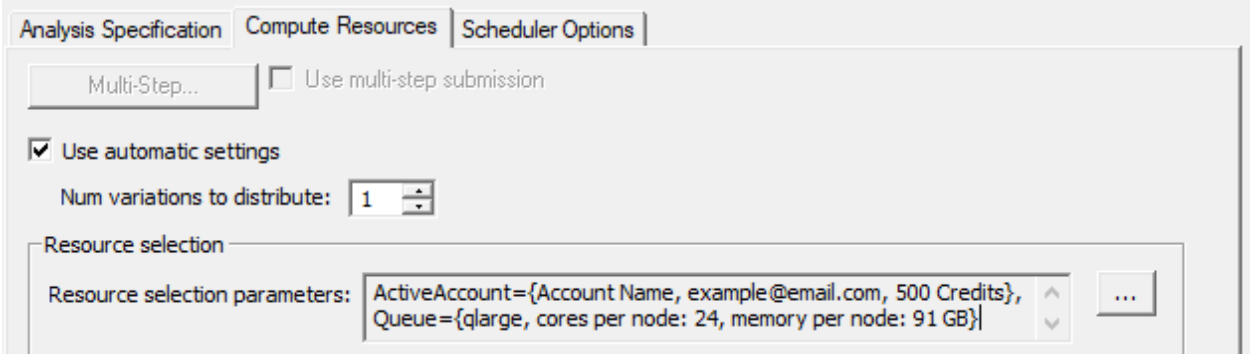
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **Note:**
>
> Ansys strongly recommends using Automatic settings.

7. Use the buttons in the **Resource selection** area to allocate resources. See: Job Management for Large Scale DSO. If you are using Ansys Cloud Burst Compute select an active account using the ellipsis […] button to the right of the **Resource selection parameters:** text box and ensure that it has sufficient Core Hour Balance. This balance

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be displayed in the **Resource selection parameters:** text box.



8. If you opted not to use automatic settings, you can also set **Job distribution** parameters. See: Distributed Analysis.
9. If applicable, select the **Scheduler Options** tab and set the **Job name** and **Priority**.

10. If desired, import or export job configurations.
11. If desired, click **Save Settings as Default** to save the current settings and overwrite defaults. These settings are saved on a per-scheduler basis.
12. If desired, click **Show advanced options** to see options for advanced users. See: Running Electronics Desktop from a Command Line.
13. If desired, click **Preview Submission** to view the commands to be sent to the scheduler. Ansys Cloud Direct submissions contain queue (pool) configuration details, including the job's hourly cost in Ansys Elastic Units (AEUs).

    The preview text can be copied to the clipboard.
14. Click **Submit Job** to submit the job to your selected scheduler.

Using Advanced Job Submission Options

From the job submission window, you can select **Show advanced options** to enable additional analysis options and job submission options.

This topic covers the following:

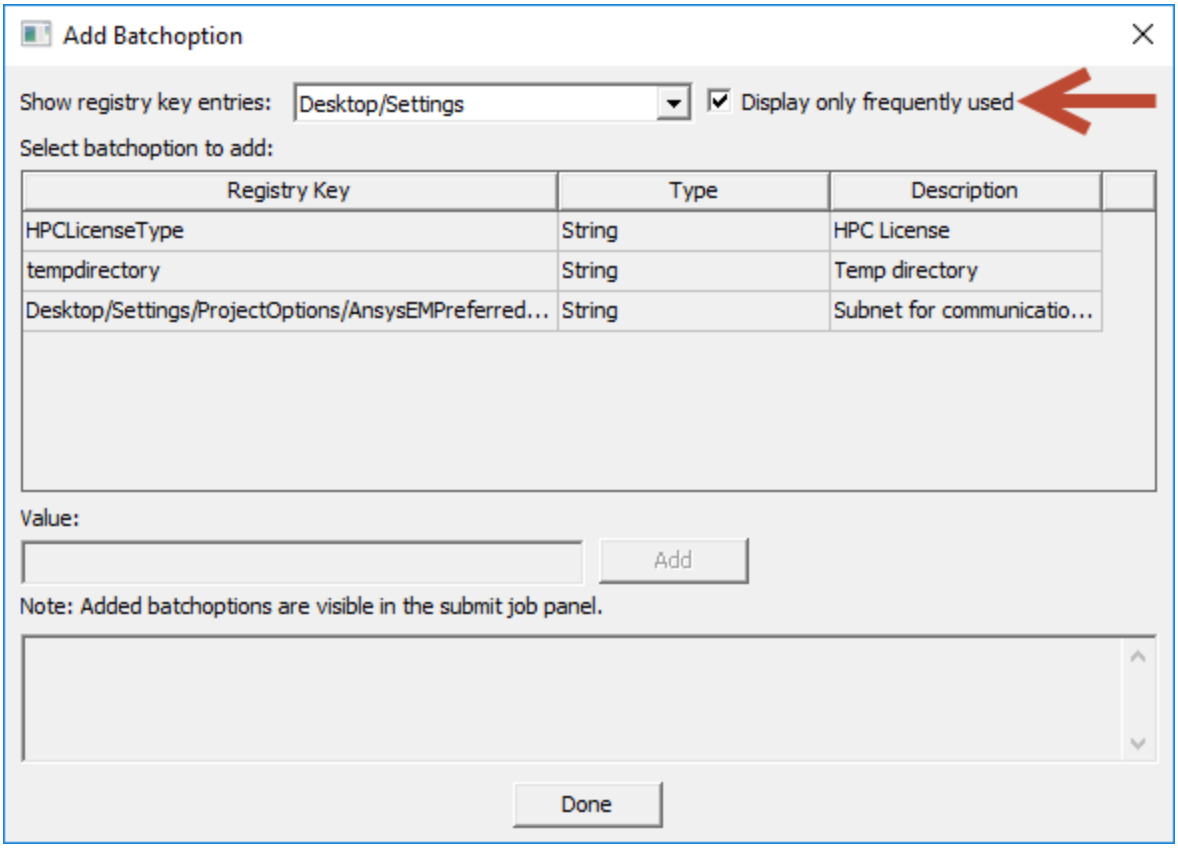
- Batchoptions
- Environment Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Batch Extract
- Customize Job Submission

Batchoptions

In the **Submit Job To** window, under **Batchoptions**, click **Add** to open the **Add Batchoption** window.

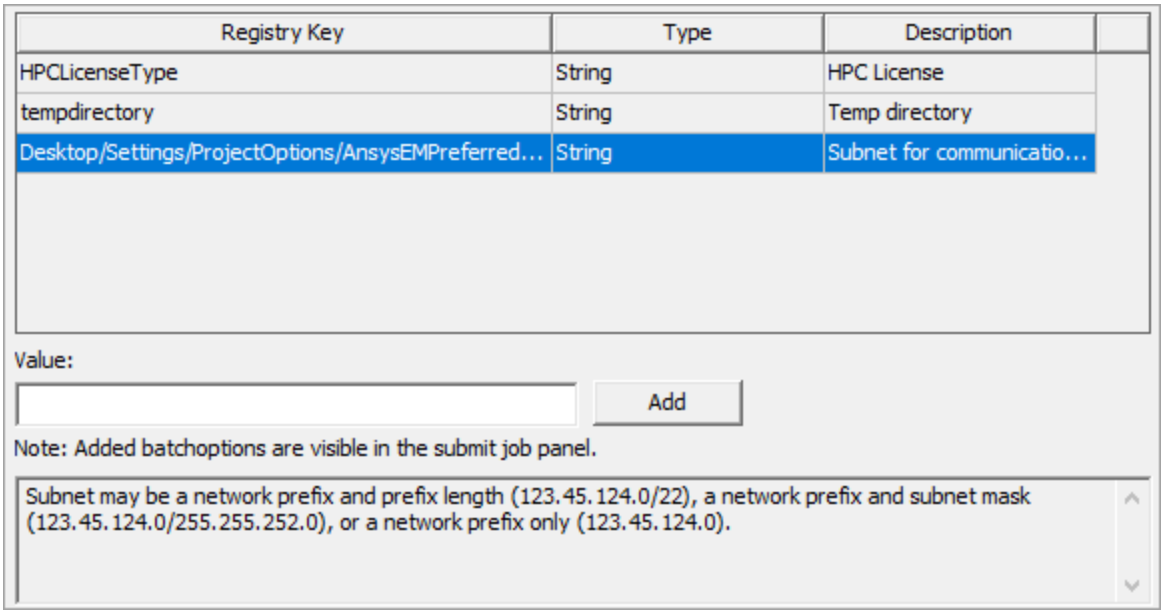


The **Show registry key entries** drop-down menu allows you to select categories of registry keys to display.

Note that **Display only frequently used** is selected by default. Deselect this to view all options for the selected category.

Select a registry key to activate the **Value** field, where you can enter a value. Selecting a key also populates the bottom of the window with a description of that key:

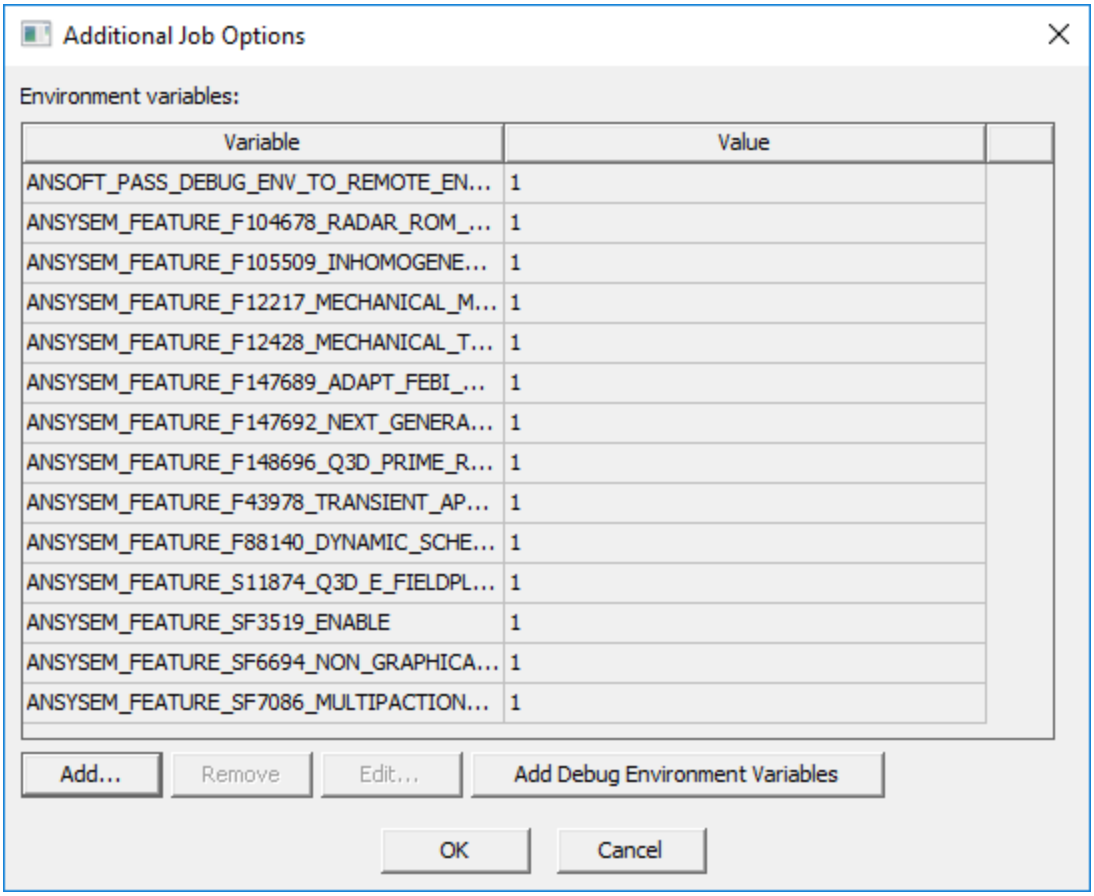
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variables

In the **Submit Job To** window, under **Environment**, click the ellipses (**...**) to open the **Additional Job Options** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lists all currently activated environment variables.

Active variables display a value of 1. Inactive variables display a value of 0. Non-binary variables may contain project paths, integer values, etc.

You can perform the following actions:

- **Add** – Add an environment variable. You will be prompted to enter the variable name and value.
- **Remove** – Select and remove an environment variable from the list.
- **Edit** – Select an environment variable and enter a new value.
- **Add Debug Environment Variables** – Activates a selection of debugging variables and adds them to the list. The log files created are only useful to development and if a customer or an application engineer needs to set these environment variables they should be working with a developer directly or indirectly who will know what needs to be set.

Batch Extract

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Submit Job To** window, select **Use batch extract** to enable the **Script path** field. Click the ellipses (**...**) to browse and select a VBscript or Python script to execute along with the job. See: Running Ansys Electronics Desktop from the Command Line for a description of Batch Extract.

Customize Job Submission

In the **Submit Job To** window's **Scheduler Options** tab, advanced options for some schedulers allow you to **Customize job submission**. When the **Override job submission** radio button is selected, user-specified options replace most of the job submission options. When the **Additional job submission options** radio button is selected, user-specified options are appended to the bsub command.

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" below
- "Distributed Analysis and Multi-Processing in the Same Job" on the next page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v251\AnsysEM\" and you are using RSM for DSO:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Job Import and Export

The bottoms of the scheduler selection and job submission window contain buttons for **Import...**, **Export...**, and **Import Configuration**. Import and Export may be used to save and then restore a frequently used collection of job submission settings, to save multiple sets of settings, or to transfer settings from one machine to another. Electronics Desktop uses Ansoft Electronics Registry Settings (*.areg) files for these purposes:

- **Export...** – exports most of the settings of this window (all tabs) to an *.areg file.
- **Import...** – updates most of the settings in this window (all tabs) from an *.areg file.
- **Import Configuration** – updates DSO settings in this window from any DSO configuration, as shown in the **Configurations** and **Options** tabs of **HPC and Analysis Options**.

> **Important:**
>
> The Design Type of the DSO configuration must match the design type of one of the designs in the project, so the **Project path** must be specified before using the **Import Configuration** button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting

The SubmitJob scripting command uses job submission settings that have been exported from the **Submit Job** window to an *.areg file. The path to this *.areg file is the first argument to the SubmitJob scripting command. See: Job Submission Scripting.

Multi-Step Job Submission

Multi-step job submission allows you to divide the simulation of a project on a cluster as two or more jobs, each of which has unique resource specifications. For example, creating the initial mesh and doing adaptive refinement can use a single machine, while frequency sweeps can easily be distributed over many machines. Breaking up the simulation into multiple steps allows the first job to do initial meshing and adaptive passes, only reserving a single compute node, or maybe even reserving just a partial node. The second job can then do the frequency sweep(s), reserving and using multiple nodes. Note that while the first job runs, because it may only be using one node, other nodes are available for other jobs. The Ansys Electronics Desktop job submission GUI allows you to submit multi-step jobs, and specify compute resources individually for each step. Electronics Desktop can also be used to monitor the execution of multi-step jobs.

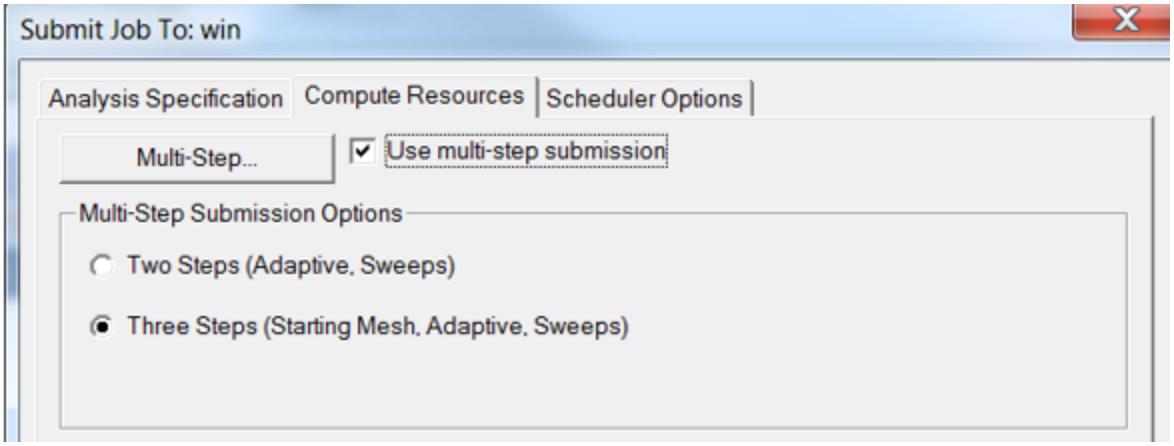
Limitations

Multi-step job submission is only allowed for a single setup of a project. Only certain design types (or setup types for a given design type) offer this functionality. Ansoft RSM does not support multi-step jobs because it does not have queuing capabilities.

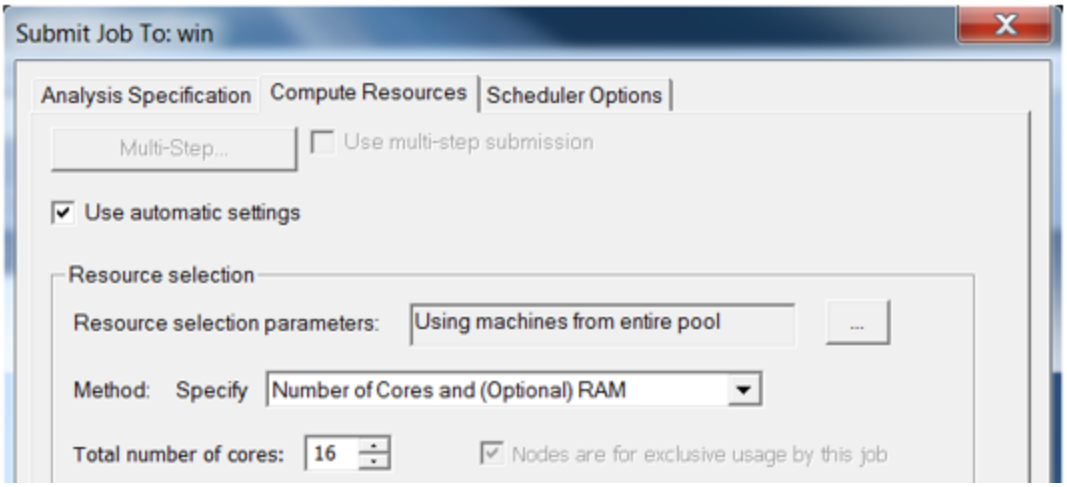
How-to Specify Multi-Step Job Submission

From an open project, right-click a setup in the project tree, and select **Submit Job...** on the shortcut menu. This pre-populates the **Submit Job To** window's **Analysis Specification** tab for the selected setup. Select the **Compute Resources** tab. If the design setup and selected scheduler allow for multi-step submission, and your computing resource supports it, the **Compute Resources** tab shows the **Multi-Step...** button and the **Use multi-step submission** check box is enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the **Multi-Step** button and check box are not enabled, it could be because you have not selected a single setup, the design type of the setup does not support Multi-Step, or the scheduler type (e.g., RSM) does not support it.

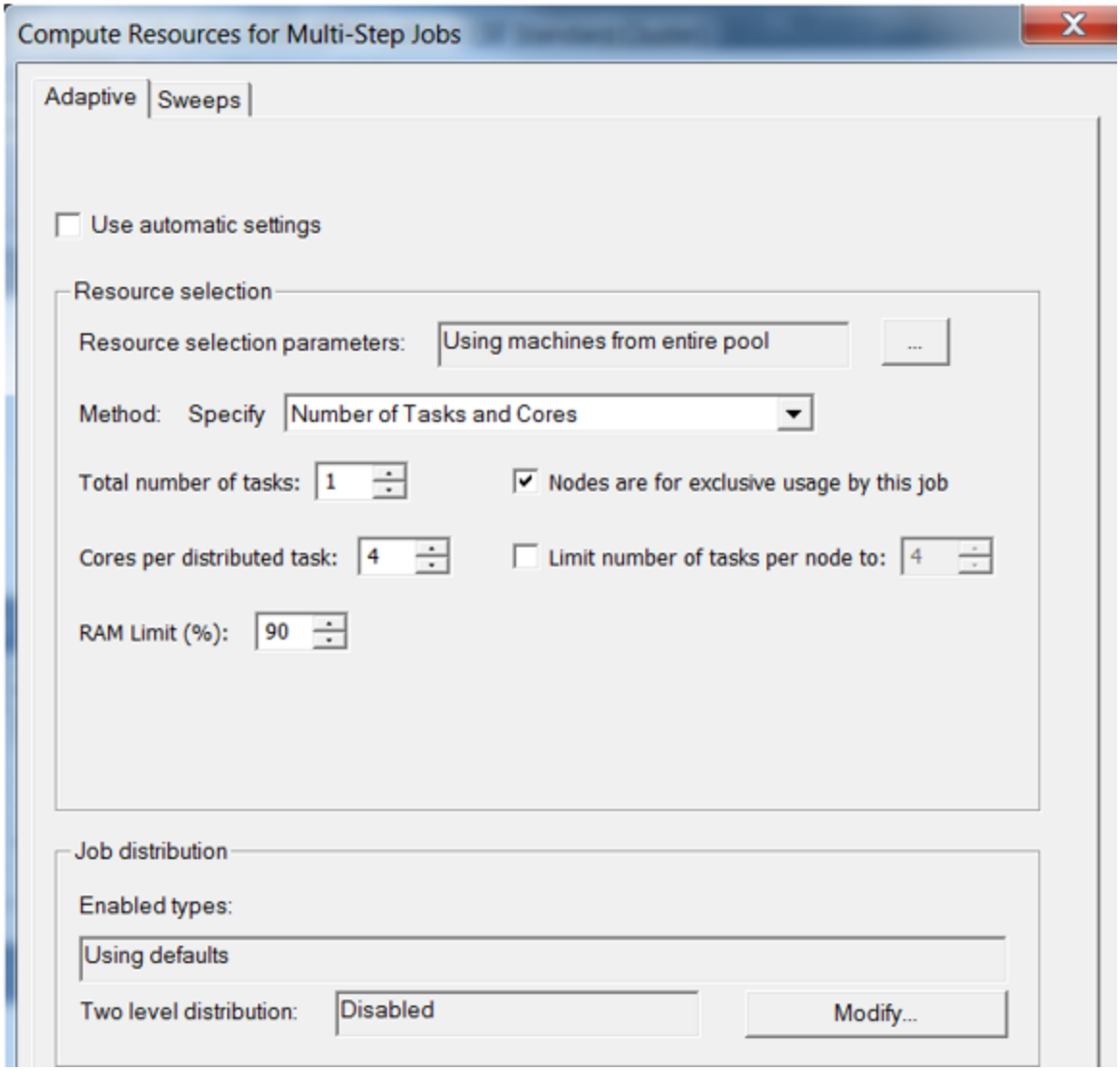


On the **Compute Resources** tab, check the **Use multi-step submission** box and select the appropriate submission option:

- Two Steps for Adaptive and Sweeps
- Three Steps for Mesh, Adaptive, and Sweeps

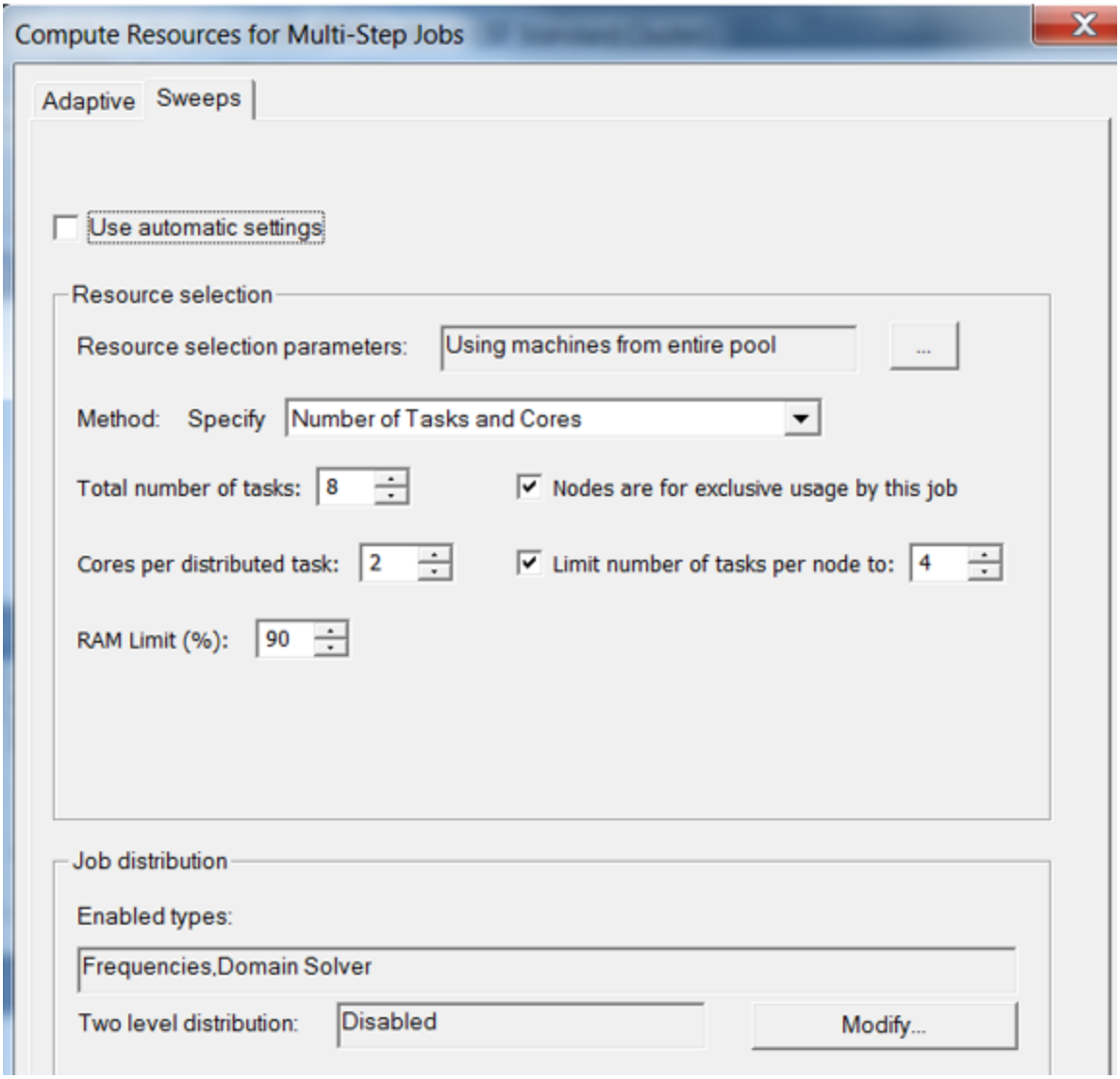
Then click **Multi-Step…**, which will bring up a **Compute Resources for Multi-Step Jobs** dialog box where there is a tab for each step used, that is for potential for Mesh, Adaptive, and Sweeps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Notice, for example, that the **Sweeps** tab lists the same resources choices but they can be assigned differently.

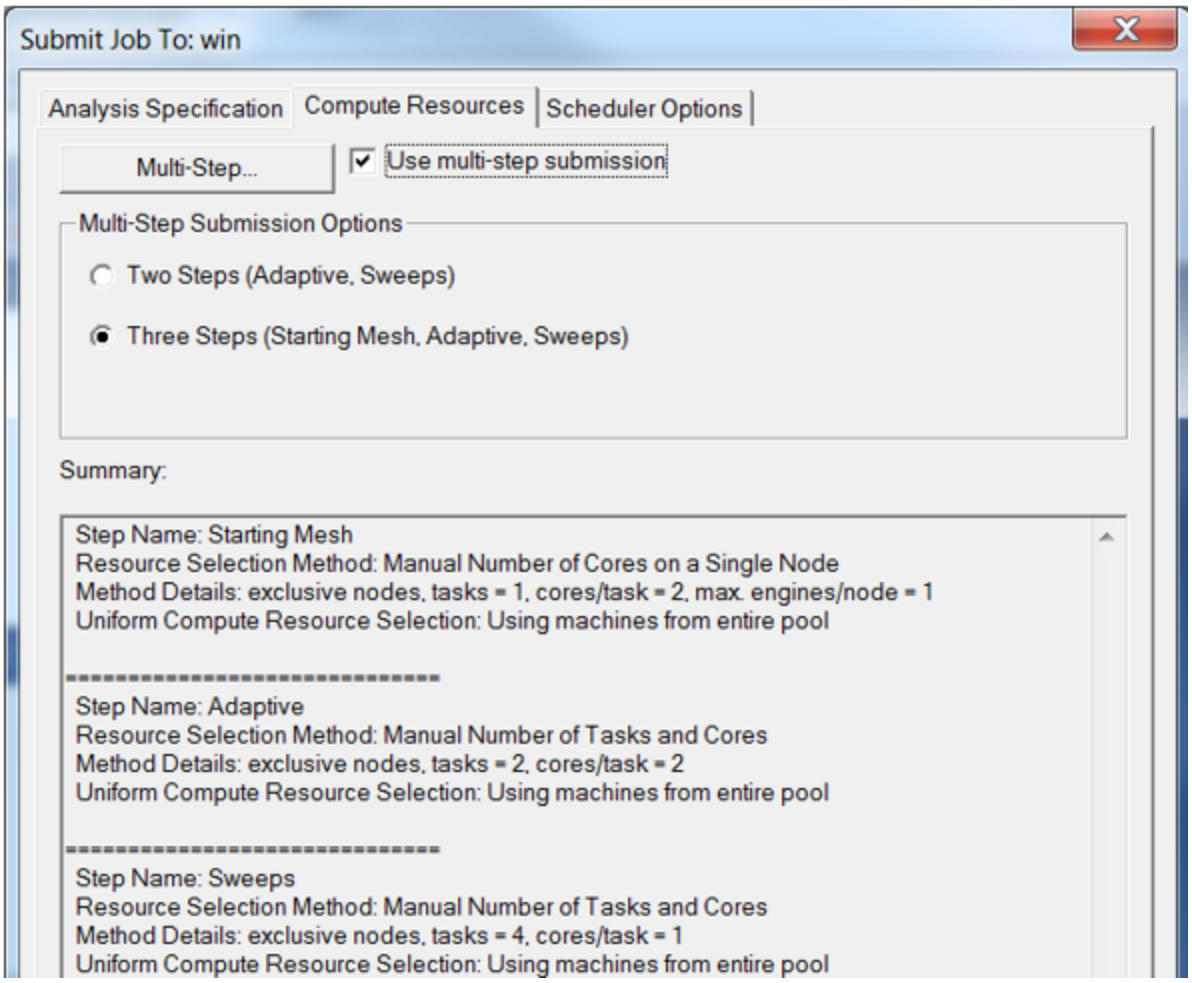
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



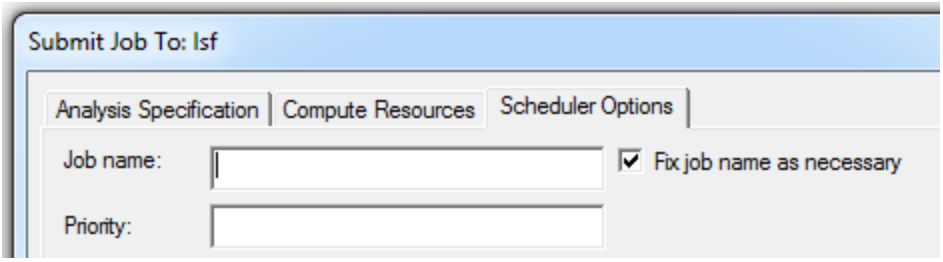
Set the desired compute resources for each step and click **OK**.

The Summary field of the **Compute Resources** tab shows a text summary of resource specifications used for each step.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



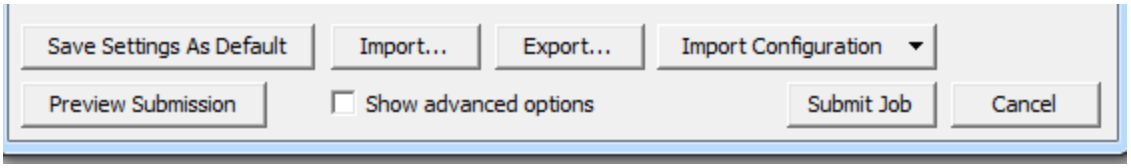
If desired, set the job name on the **Scheduler Options** tab. When the jobs are submitted, each will have a name given by job name (if any) appended with the name of the step for the job. For example, if the job name entered is “MultiStep”, then the individual jobs might be named “MultiStep-Adaptive” and “MultiStep-Sweeps”.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is a new **Fix job name as necessary** check box for the job name. This applies only to certain schedulers where there are constraints on the job name. It is useful to ensure proper submission in the case where appending the step name results in an invalid job name.

When you click **Preview Submission**, you see a preview for each step, all in the same output window. Any errors or warnings for a step, generated during preview validation, are displayed with the text for the preview of that step.



When you click **Submit Job**, each job is submitted individually, and dependencies are set with the scheduler so that subsequent jobs wait for the prior step's job to complete before starting. Any errors in job submission for any step prevent further steps from being submitted.

The window reporting successful submission has been modified slightly for multi-step jobs. It will show the jobs IDs of all jobs that are part of the multi-step job sequence.

Multi-Step Job Monitoring

You can monitor the job step in progress. When one job completes, the status shows "Completed", but monitoring restarts once the next job step is running.

Aborting a Job Step

You can also abort the job step in progress. With multi-step jobs submitted from the Electronics Desktop GUI, this will also cause remaining jobs (which would otherwise remain queued in the scheduler) to be canceled.

Archive Projects for Multi-Step Job Submission

Note that you can submit archive projects. Monitoring is based on the archive for the first step, then on the extracted project for subsequent steps.

Submitting a Job without Opening the Project

You can also submit a job without opening the project. This can be done by choosing **Tools** > **Job Management** > **Submit Job…** and then manually entering the project path (You can also use **Browse** can to select the project.) Note that you must also select a single nominal setup before the **Use multi-step submission** check box is enabled on the **Compute Resources** tab.

Windows to Linux Job Submission

Given a set of prerequisites, Ansys Electronics Desktop can permit Windows to Linux job submission as part of HPC.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites for Job Submission

Directory Shared between Windows and Linux

For all jobs submitted to a Linux cluster, the project file is required to be in a directory that is accessible from all execution hosts used by the job. For submission of jobs from a Windows host to a Linux cluster, the project file must also be accessible from the Windows host where the GUI runs. There must be a directory shared with both Windows and Linux hosts, and the project file may be in a subdirectory (at any level) of the shared directory.

Network Access from Windows Host to Linux Job Management Host

The job is submitted to the cluster from a Linux host configured for submission of jobs to the Linux cluster. We call this Linux host the “Job Management” host. The information about the job to be submitted is transmitted to the Job Management host over the network. As a result, the Windows host where the GUI runs must have network access to the Job Management host. If this communication is blocked, then job submission from a Windows host to the Linux cluster will not be possible. Communication could be blocked if there is a firewall or if the Linux cluster is only on a private network, for example.

RSM Service Running on Job Management Host

The **ansoftrsmservice** must be running on the Linux Job Management host. Before the ansoftrsmservice is started, it must be configured for submission of jobs to the cluster. The SchedulerName and ConfigString fields in the Scheduler block of the ansoftrsmservice.cfg configuration file must be specified. The contents of these fields are described in the following table:

Field Name	Contents	Examples
SchedulerName	Identifier of Scheduler	IBM Spectrum LSF: ’`lsf`’ PBSPro or Torque: ’`pbs`’ Univa, SGE, etc.: ’`sge`’ SLURM: 'generic'
ConfigString	Directory containing scheduler commands	IBM Spectrum LSF: ’’ (not required) PBSPro or Torque: ’`/opt/pbspro/PBSPro_13.0.0/default/bin`’ Univa, SGE, etc.: ’`/opt/univa/bin/lx-amd64`’ SLURM:'{"Proxy":"slurm"}'

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The environment should be configured for job submission before starting the ansoftrsmservice. The ansoftrsmservice should be run as a non-privileged user; no special privileges are required. It should be run as a user without login privileges, so that only privileged users have access to this process.

Prerequisites for Job Monitoring

For job monitoring, all prerequisites for job submission are required. One additional requirement, described below, is also required for job monitoring.

Network Access from Windows Host to Linux Cluster Hosts

In order to obtain full monitoring information from a job, the Windows host needs access to some of the job processes. That is, the Windows monitoring host requires network access to the processes running on the Linux cluster execution hosts. If this communication is blocked, then only limited monitoring information is available.

Supported Schedulers

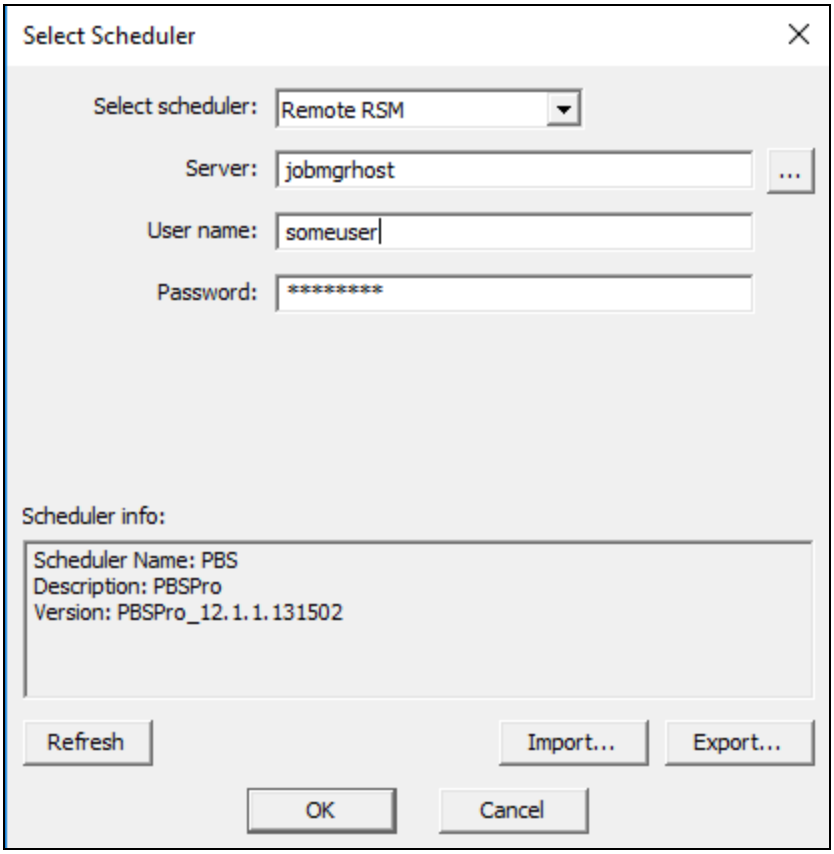
This feature may be used with all Linux schedulers for which job submission from the GUI is supported:

- IBM Spectrum LSF
- Univa Grid Engine (GE)
- PBSPro
- Torque

Select Scheduler Dialog

If you select the **Use a computer on the network** option, you can enter a username and password. This username and password are used when the job is submitted to the Linux scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

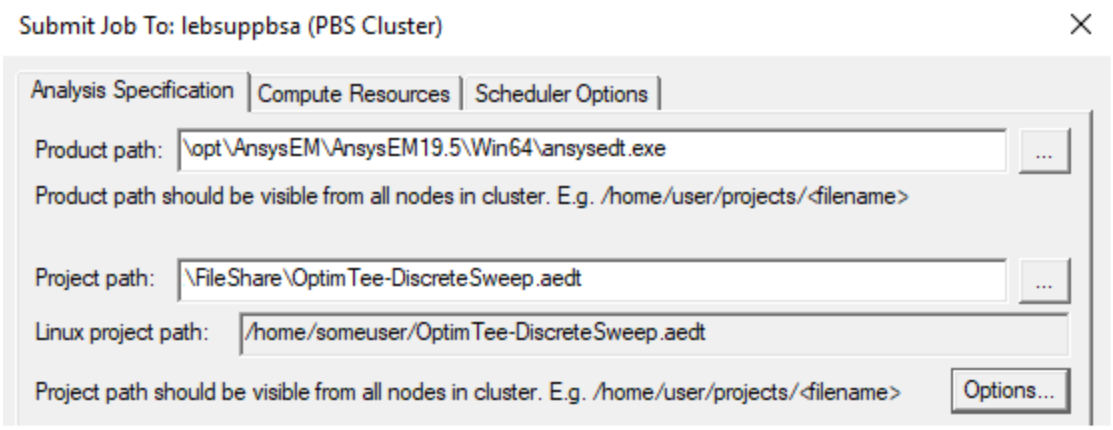


Submit Job

In the ***Submit Job To*** window, you must enter the Linux path to the product in the Product path edit control. The browse button (labeled "...") may not be used to browse for the product. There is no requirement for the product installation directory to be accessible from the Windows GUI host.

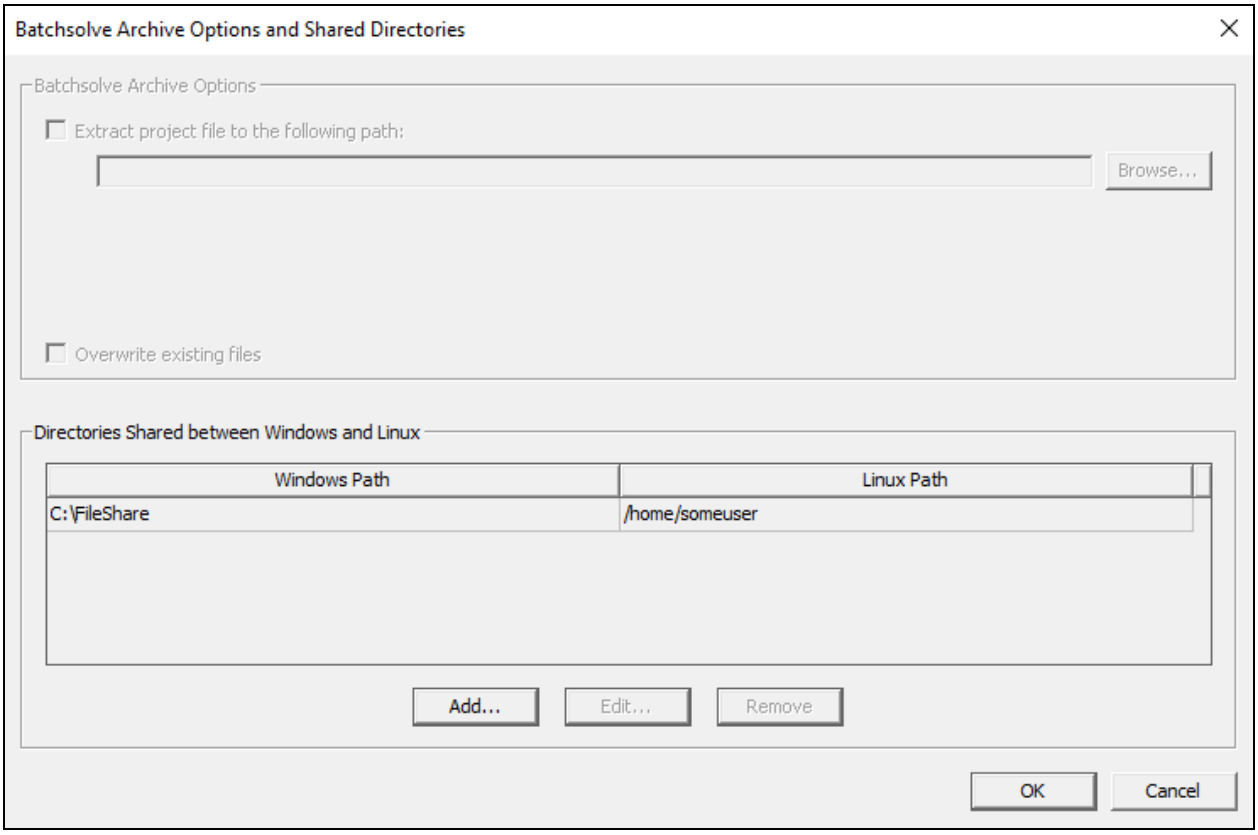
You must enter the Windows path of the project file in the Product path edit control or use the browse button (labeled "...") to select the Windows path of the project file. If the Linux path of the project file can be determined from the specified Windows path and the directories shared between Windows and Linux, then the Linux path of the project file is shown in the Linux project path edit control. This edit control cannot be edited directly.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



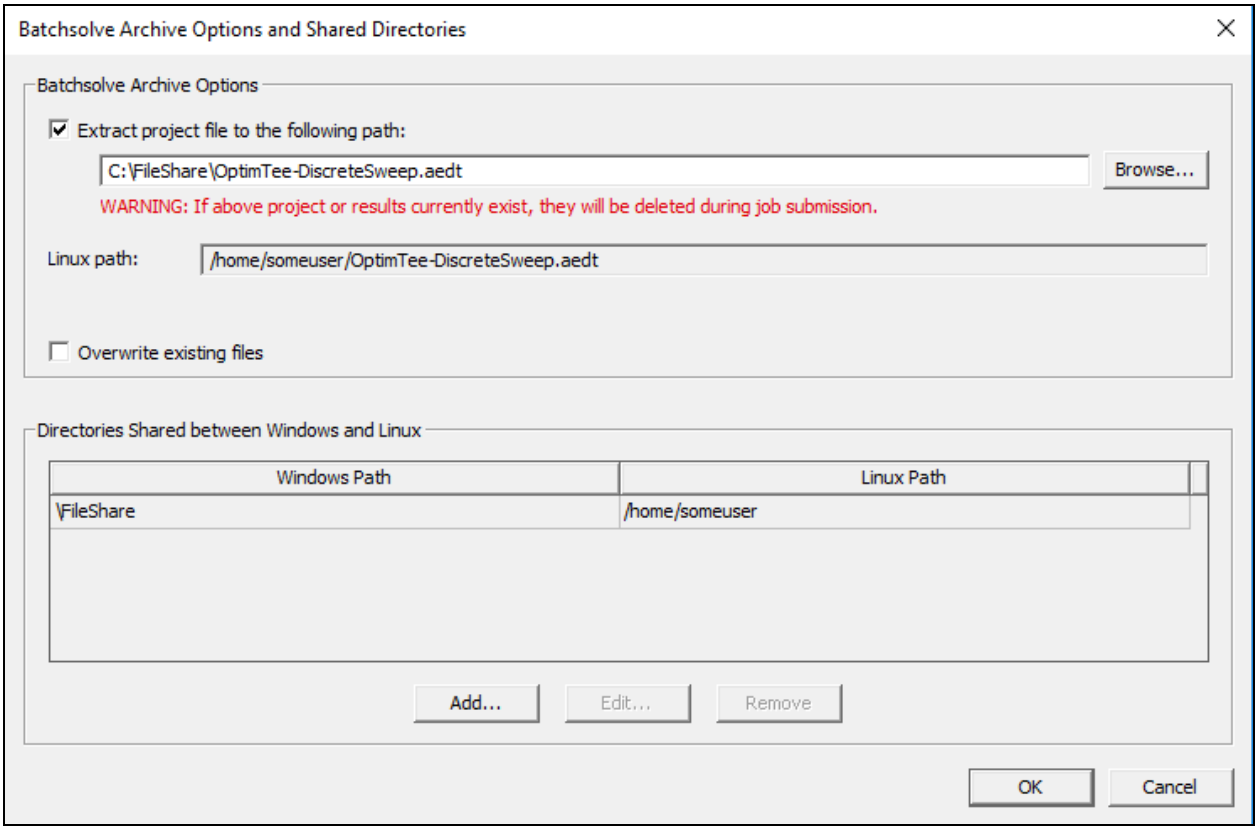
The **Options** button opens a window that you can use to specify archive options for a job. The lower portion of this window allows you to specify one or more directories shared between Windows and Linux. The mapping of directories between Windows and Linux is shown in a grid which displays the Windows path and the Linux path for each shared directory. There are also buttons to add a new shared directory, to edit an existing shared directory, or to delete one or more shared directories. The Windows path or the Linux path of any shared directory may be selected in the grid and directly edited, as well.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



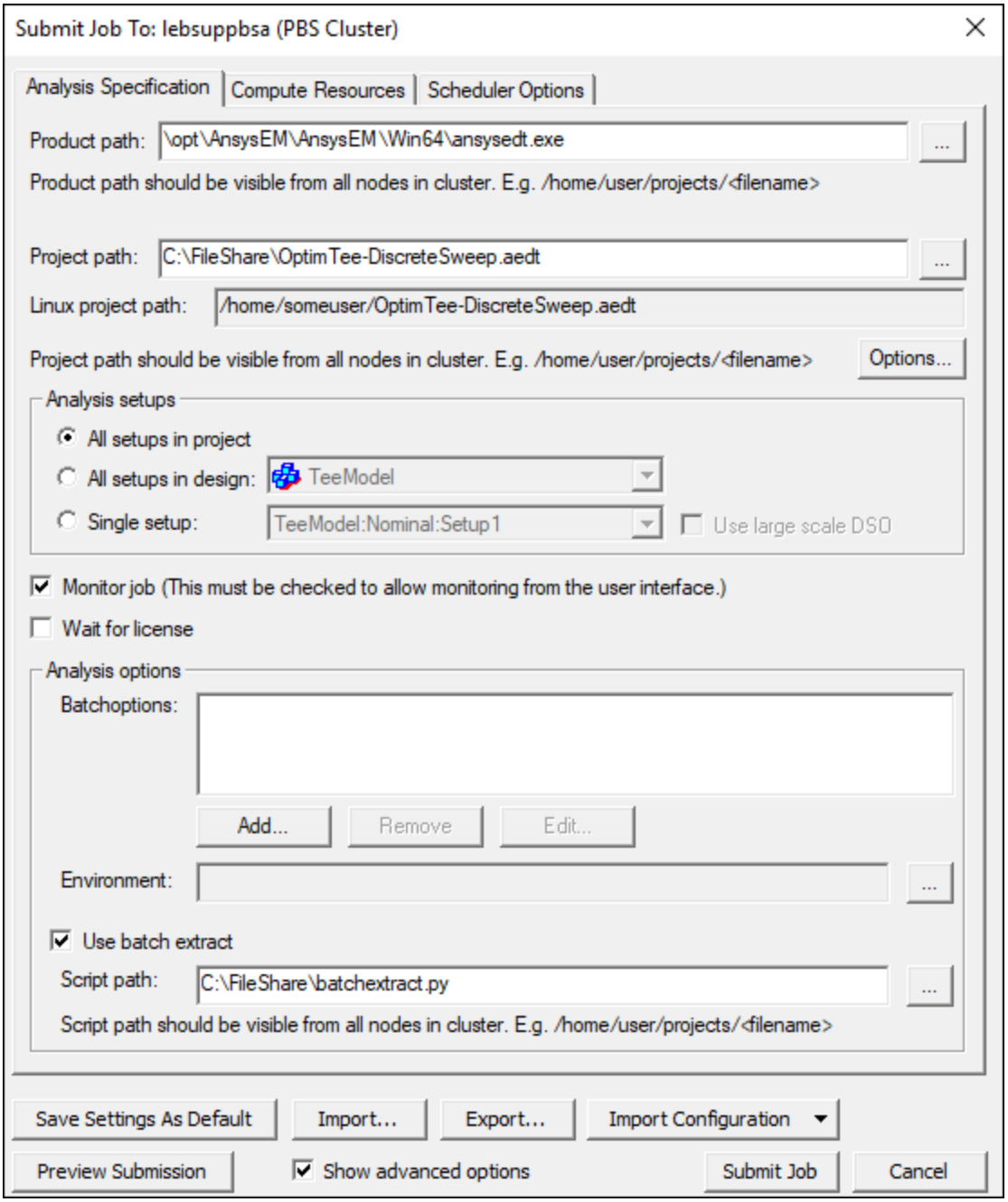
If you specify a project in an archive, the window may be used to specify the Windows pathname of the project to be extracted from the archive. If this is done, the Linux pathname of the target project is determined from the directories shared between Windows and Linux and shown in the upper portion of this window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you specify a batchextract script, the Linux path of the batchextract script is determined from the Windows path of the batchextract script and the directories shared between Windows and Linux.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



User Passwords are Encrypted

Jobs are submitted to the Linux cluster using the user name and password entered in the **Select Scheduler** window. These settings are persistent; in general, these settings need to be entered only if they change. To ensure security, user passwords are stored in an encrypted format. When a job is submitted from a Windows host using the ansoftrsmservice running on a Linux submission host, the user credentials are sent over the network in an encrypted format.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Submission Scripting

To help with automation, you can submit batch jobs can through script commands of the **oDesktop** object. The **SubmitJob** script command uses job submission settings that have been exported from the **Submit Job** window to an .areg file. The path to this .areg file is thus the first argument to the SubmitJob command. Additional arguments include the path to the project file, the design name (if restricting the solve to a particular design), and the setup name (if further restricting the solve to a single setup within a design).

For further automation, you can use the **SelectScheduler** scripting command to determine what scheduler to use for submission, to include options for head node, username, and whether to require password entry from the user. (If the username differs from the cached username, or the force password flag is set, then the *Select Scheduler* dialog box appears.) If there are any issues with the scheduler selection (for example, a password is required or the requested host wasn't found), then the *Select Scheduler* window appears. If Ansys Cloud Direct or Ansys Cloud Burst Compute are selected then the "Login" button will open the default browser and prompt for an AnsysID login. This is the only part of job submission scripting that may required user intervention. This same mechanism is used if, from within the SubmitJob command, there is failure to connect to the scheduler. Even though there are allowances for graphical user intervention if something goes wrong, if the password (if required) is cached and all settings are correct, the entire submission process can run non-graphically and fully automated.

Limitations

All settings besides the arguments passed to the **SubmitJob** command must be stored in the .areg file containing settings exported from the job submission window. These include (but aren't limited to) batch options, environment variables, batch extract settings, and compute resource selections. To run many job submission scripts with variation of these settings, there must be multiple .areg files available.

Note that the same project can be submitted multiple times with a single script. Care must be taken in this situation because each time a project is submitted, the state-keeping files used for monitoring are removed so that the job can create them from scratch to ensure consistency. While this ensures proper monitoring for a job that is just being submitted, it could interfere with monitoring (or even correct solving if a lock file is deleted) of a job that is already in progress. Because of this, if the same project is to be resubmitted from within a single script, the job should be monitored (waiting for completion) before trying to submit it again. This monitoring/waiting can be done with a combination of a single **LaunchJobMonitor** command followed by a loop that checks the result of a **RefreshJobMonitor** command.

How to Perform Job Submission Scripting

The typical scenario for job submission scripting would be to do the following:

1. Manually select the scheduler. Use the **Select Scheduler** window to open the **Submit Job To** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose a representative project (with the desired design type), and select appropriate analysis settings.
3. Make the required compute resource selections and try to preview the job.
4. If preview is successful, export the dialog settings and record the path to the new .areg file.
5. Create a script containing at a minimum a **SubmitJob** command with the path to the .areg file, and the path to the project file. Note that there must be double backslashes for each backslash of a path because the backslash is an escape character. When the script is run, and all is successful, there should be a message in the message windows stating that the job was submitted, including the job ID(s). There could be multiple job IDs if multi-step submission is used.

See the Scripting help (click **Help** > **MechanicalScripting Help**) for details on the **SelectScheduler**, **SubmitJob**, **LaunchJobMonitor** and **RefreshJobMonitor** commands.

Monitoring Jobs

There are a number of tools that allow you to monitor jobs, including the **Monitor Job** window and detailed .log files.

- Monitor Job Window
- Monitoring Ansys Cloud Direct Jobs
- Monitoring Ansys Cloud Burst Jobs
- Monitoring Large Scale DSO Jobs
- Web Client for Batch Solve Monitoring and Reporting

Monitor Job Window

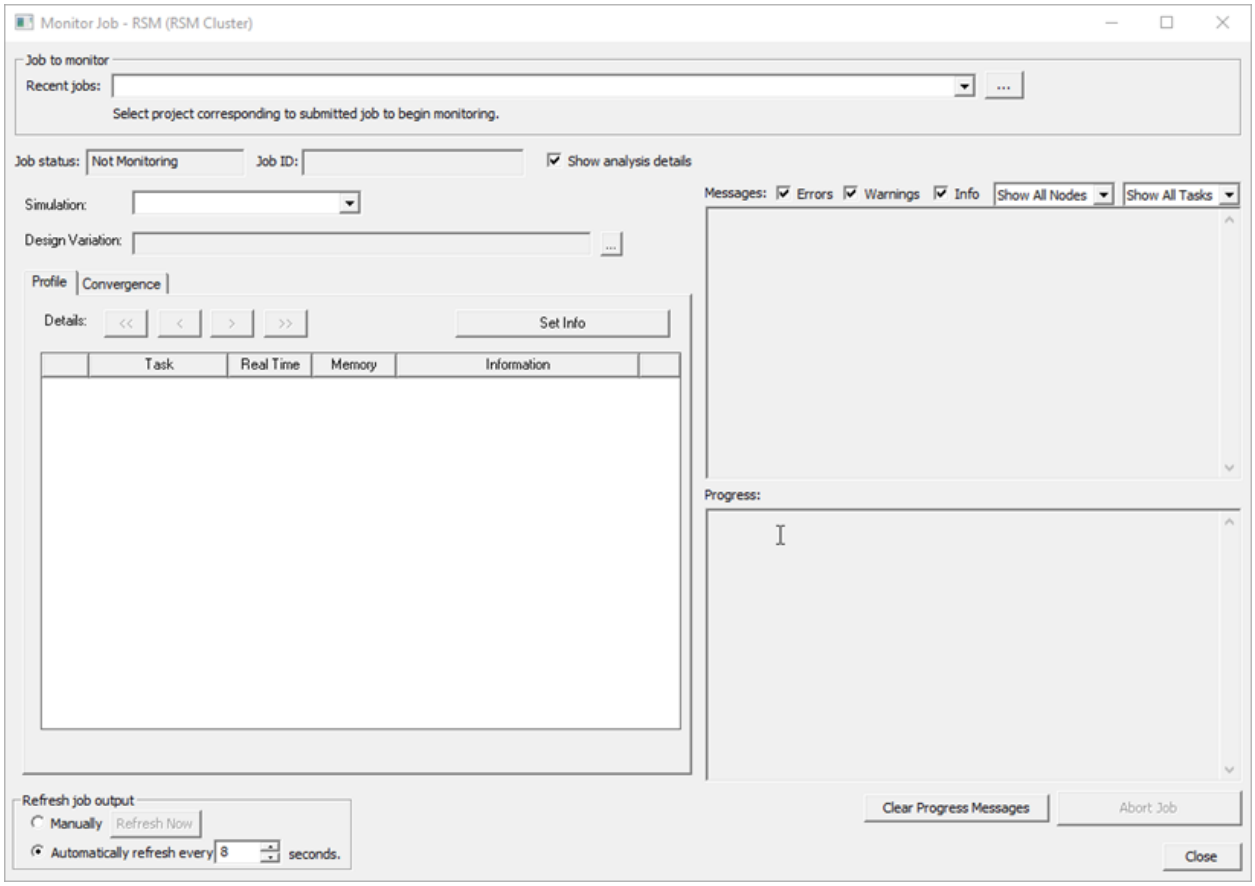
The **Monitor Job** window allows you to monitor the progress and status of jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

To monitor jobs:

1. Access the Monitor Job window one of the following ways:
   - Click **Tools** > **Job Management** > **Monitor Jobs**.
   - Select the **Simulation** tab and click the **Monitor** icon.
   - From a Command window, use the -showmonitorjobs command.

   The **Monitor Job** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> The window may look slightly different, depending on your selected scheduler and design type.

The **Monitor Job** window contains the following areas:

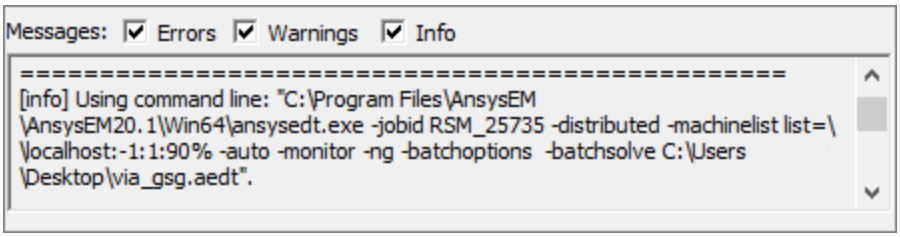
- **Job to Monitor** – Allows you to select either a recent job, or a project containing the job you want to monitor. You can select an archive.
- **Job Status** – The current job status. The normal progression is: Starting Monitoring, Queued, Running, Shutting Down, Completed. A status of Unknown may indicate a connection problem.
- **Job ID** – Identified by the scheduler prefix and job number.
- **Simulation** – Drop-down menu that allows you to select an individual simulation setup.
- **Design Variation** – Click the ellipses button (**...**) to select or deselect design variations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Profile Tab** – Displays detailed information about completed tasks, including execution time and memory usage.

Task	Real Time	CPU Time	Memory	Information
Start				Time: 10/16/2019 14:17:38; Host: : Processor: 12;
				Executing from C:\Program Files\AnsysEM\AnsysEM20.1\Win64\Q
				Performing minimal design validations
InitMesh	00:00:08	00:00:07	106 M	70131 triangles
PreProc	00:00:00	00:00:00	106 M	70131 triangles
AdaptMesh_1	00:00:01	00:00:00	107 M	78514 triangles
Solve(1p)_1	00:00:00	00:00:00	126 M	51207 unknowns
ErrorCalc_1	00:00:00	00:00:00	126 M	78514 triangles
PostProc	00:00:01	00:00:00	126 M	995 elements
Solution Process				Elapsed Time: 00:02:15
Stop				Time: 10/16/2019 15:15:30; Status: Normal Completion

- **Convergence Tab** – Displays the completed, maximum, and minimum **Number of Passes**.
- **Messages** – Displays errors, warnings, and job information. Use the check boxes to choose which of these to display.



- **Progress** – Displays progress bars when tasks are currently in progress.

The arrow button opens a menu that allows you to either **Abort** that analysis, or perform a **Clean Stop**.

- **Refresh Job Output** – Allows you to select either manual or automatic refresh for the **Monitor Job** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

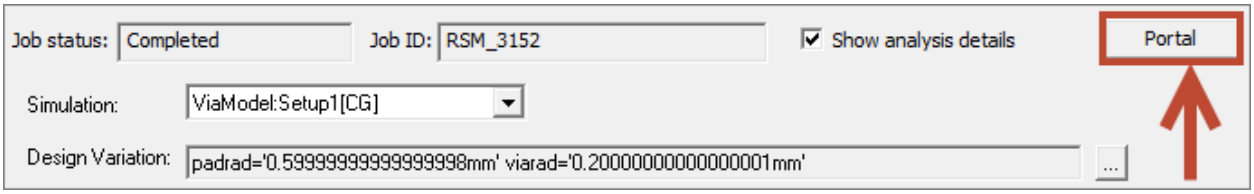
To display less information, deselect **Show analysis details**. This removes the **Simulation**, **Design Variation**, **Profile**, and **Convergence** information, instead displaying only **Messages** and **Progress**.

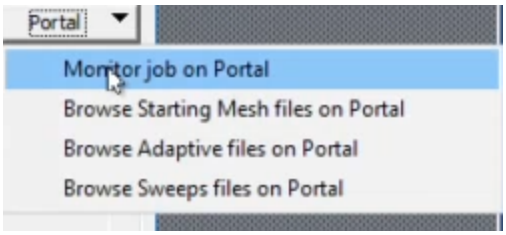
2. Click **Close** when you are done monitoring.

Monitoring Ansys Cloud Direct Jobs

For Ansys Cloud Direct submissions, the **Monitor Job** window includes two additional items:

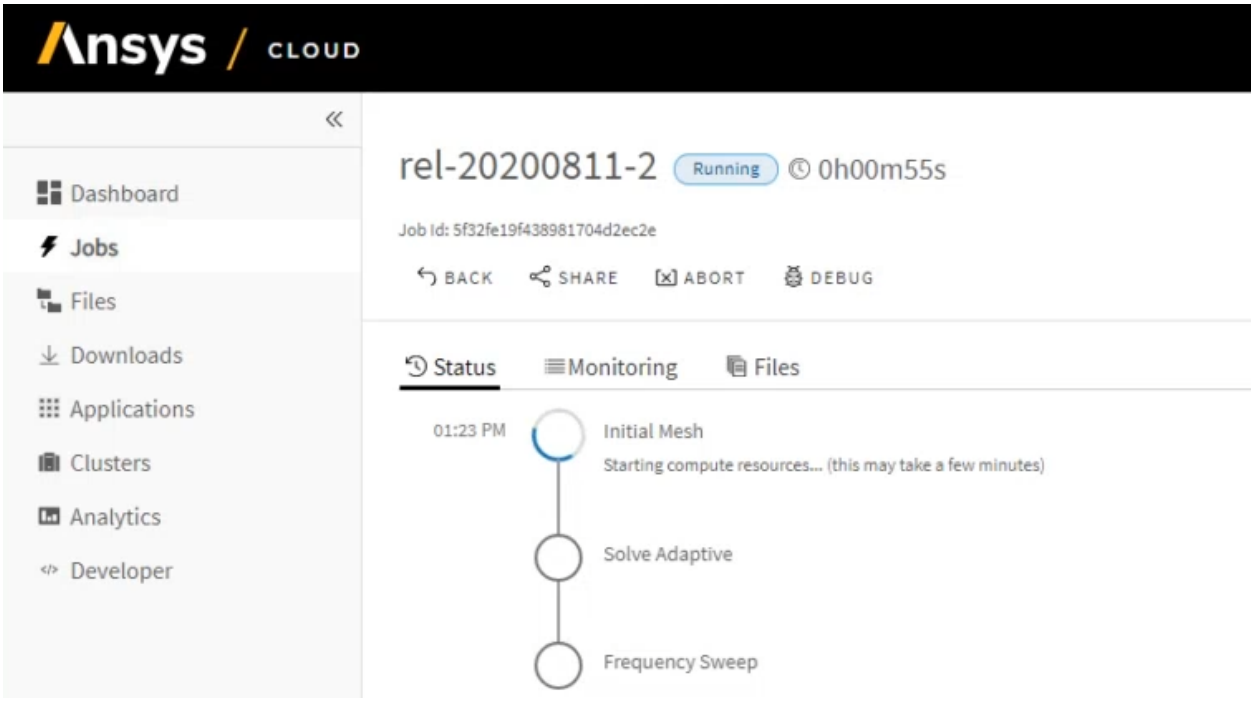
- **Portal** – link to the Ansys Cloud Direct Portal, where job details and additional monitoring information are available.



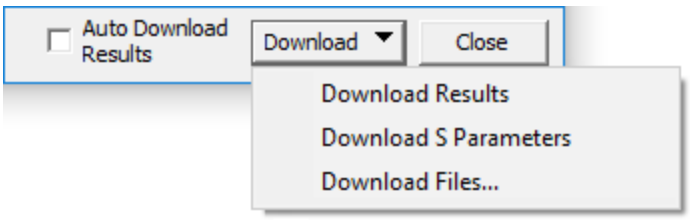


Monitor job on Portal Opens a Portal Web interface that lets you monitor an Ansys Cloud Direct job. The additional menu options for **Portal** are for multi-step submissions, opening the Portal Web interface directly to folders with Starting Mesh Files, Adaptive Files, or Sweeps files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

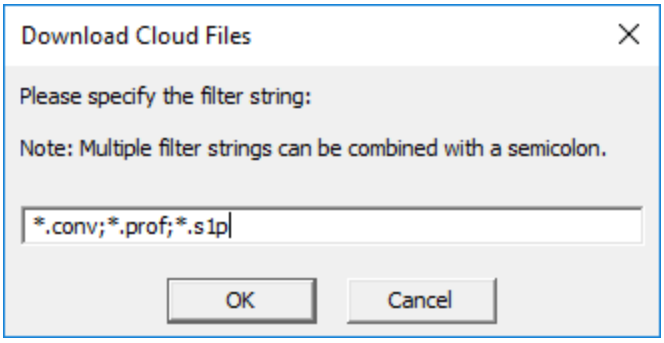


- **Download** – button that allows you to download the results, s-parameters, or specified files. These are saved in a folder with same name as the job ID inside the `/<project-name>.aedtdownload` folder (or `/<project-name>.aedtzdownload` folder, if the submission was based on an archive). You can also select the **Auto Download Results** check box to have the results downloaded to this location automatically.



If you choose to **Download Files**, the **Download Cloud Files** dialog box appears:

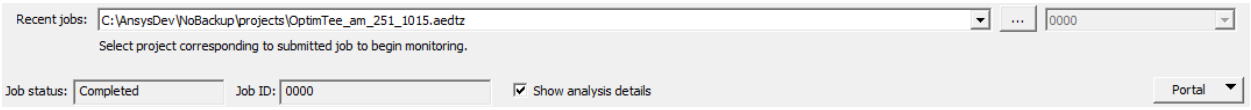
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



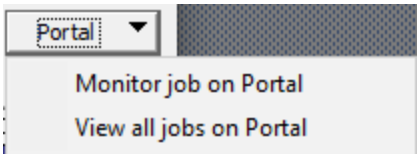
Specify the types of files you want to download (separated by semicolons) and click **OK**.

Monitoring Ansys Cloud Burst Jobs

When you submit a job to Ansys Cloud, the job status is shown in the **Monitor Jobs** window. The Job ID shown here is a short numeric identifier used to distinguish multiple jobs for the same project. Use the drop-down list at the upper right to select a different job for the same project. **The Monitor Job** window includes two additional items, **Portal**, and **Download**.



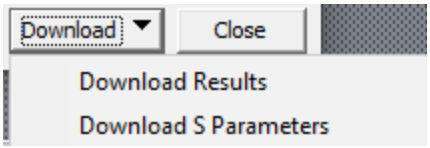
The **Portal** drop-down button provides a link to the Ansys Portal, where job details and additional monitoring information are available. For more information, see the Ansys Portal Guide.



Monitor job on Portal opens a job details view for the current job in Ansys Portal. **View all jobs on Portal** opens a job list view in Ansys Portal, enabling you to select a job to monitor.

Download allows you to download the results or s-parameters. These are saved in a folder with same name as the job ID inside the /<*project-name*>`.aedtdownload` folder (or /<*project-name*>`.aedtzdownload` folder, if the submission was based on an archive). You can also select the **Auto Download Results** check box to have the results downloaded to this location automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

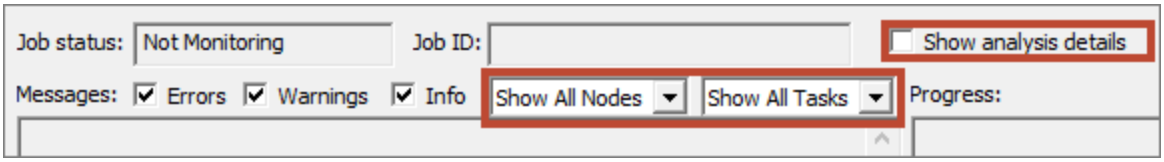


Monitoring Large Scale DSO Jobs

For Large Scale DSO submissions, the Monitor Job window includes the following items:

- **Show Messages in Node Tree** – Instead of viewing job details, you can select this option to view nodes and tasks in a tree format.

- Drop-down boxes allow you to filter which Nodes and Tasks to display in the tree.



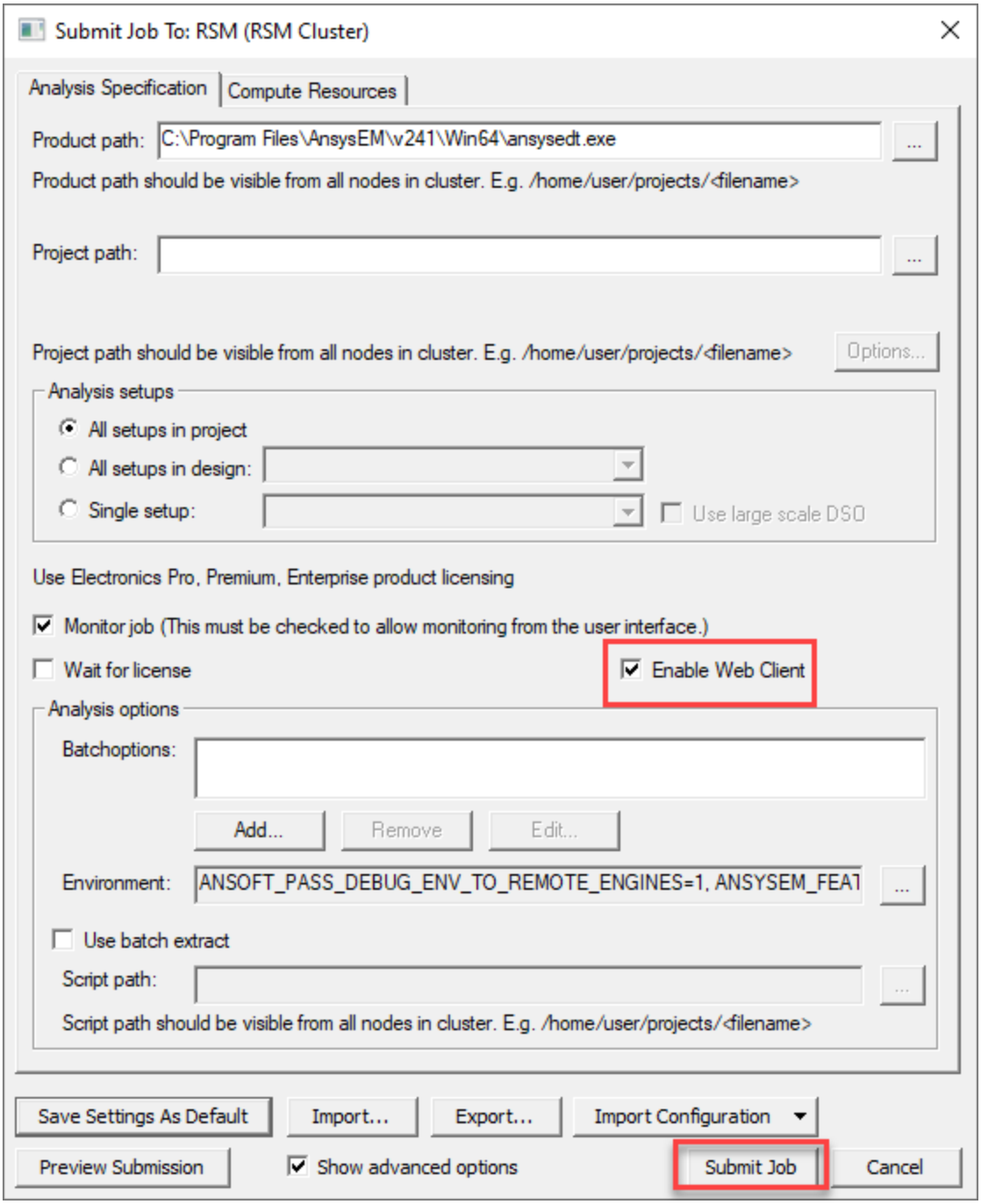
To leave this mode, select **Show analysis details**.

Web Client for Batch Solve Monitoring and Reporting

There is a Beta Option to use a Web client for batch solve monitoring and reporting. If you set the Beta Option f under General for Enable Web Client for Monitoring, you will have access to the feature.

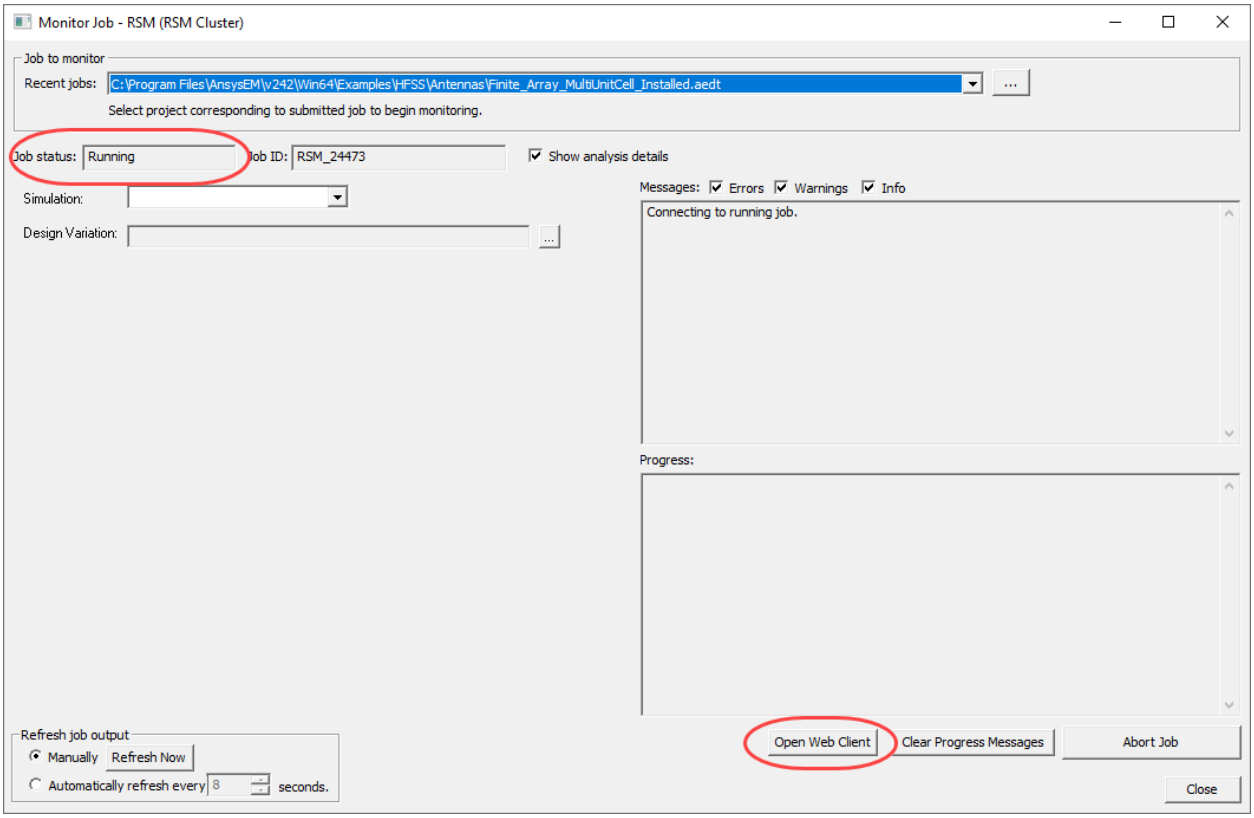
After you have selected the scheduler to use, when you access the **Submit Job To:** dialog, you well see Enable Web Client checked.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



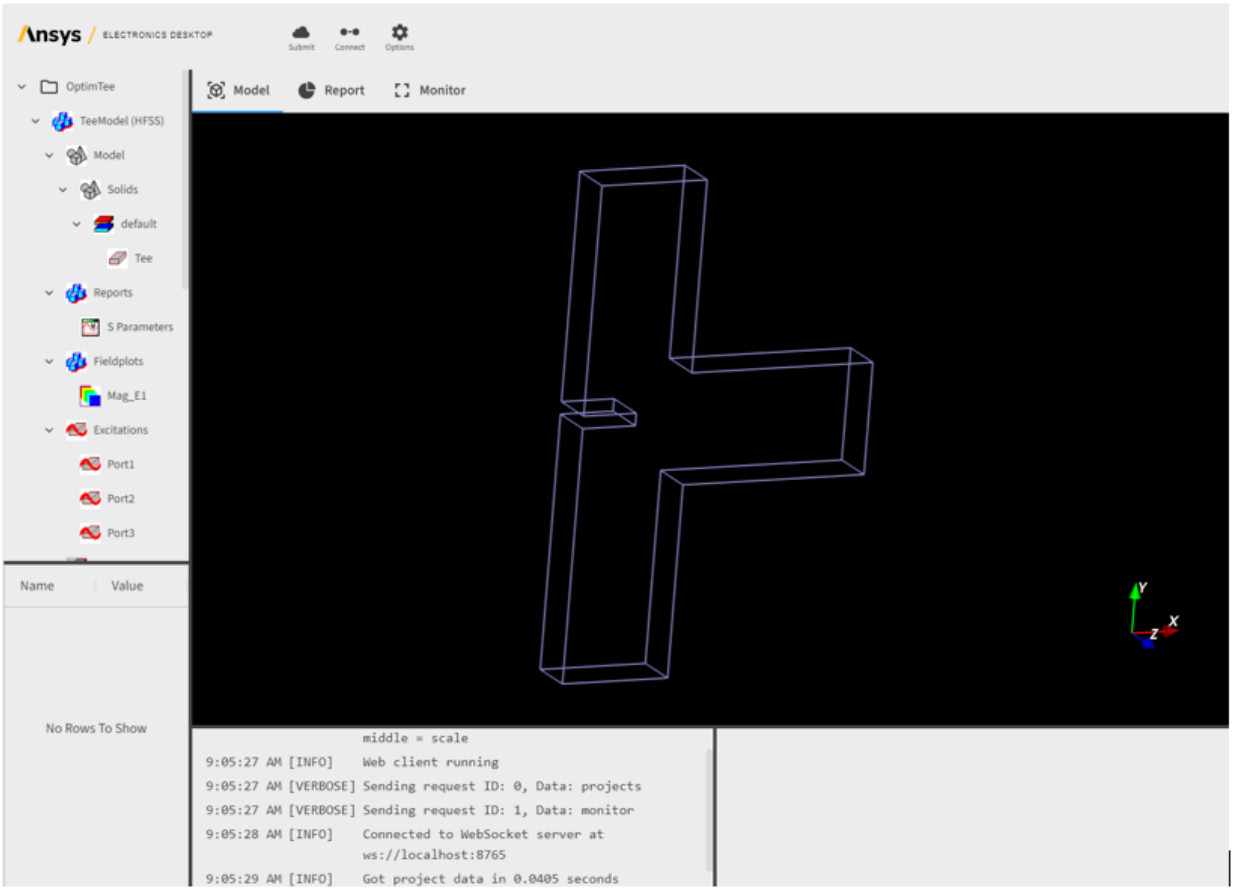
After specifying the project path and setting other parameters, you can then select **Submit Job**. This opens the Monitor Job dialog for your scheduler. Once the **Job Status** changes to Running, press **Open Web Client**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



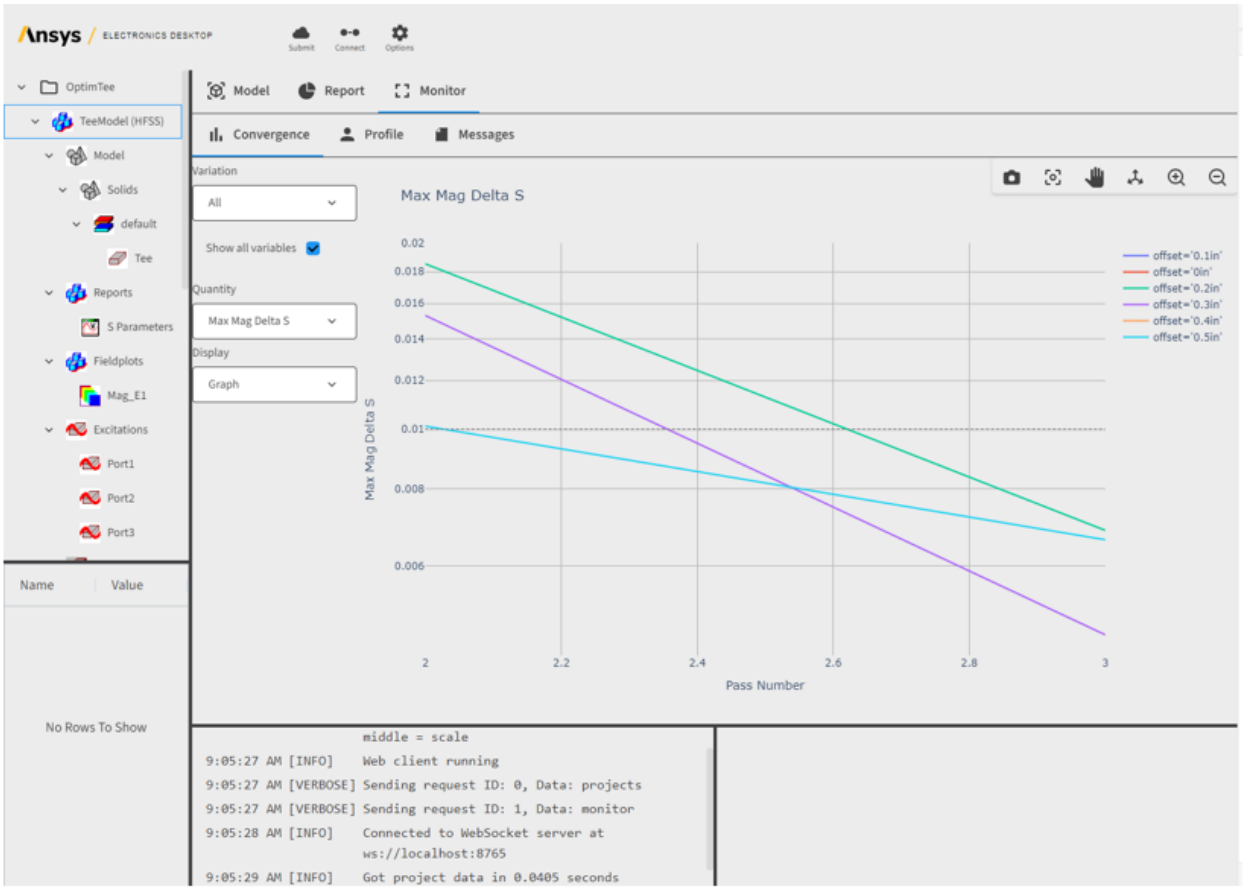
Click **Open Web Client** to open the Web Client in your default browser.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



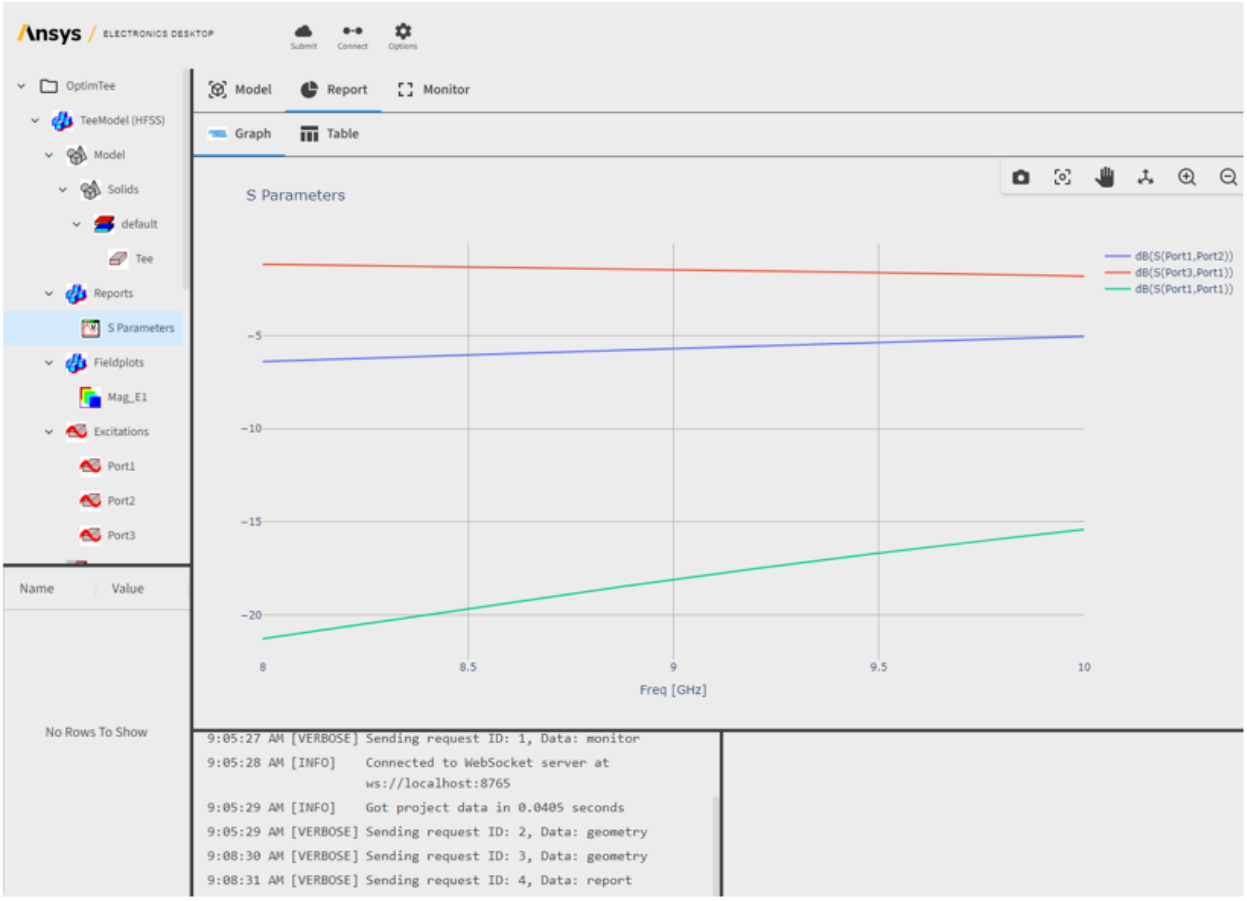
Switch to the **Monitor** tab to gain access to the subtabs for Convergence, Profile and Messages.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



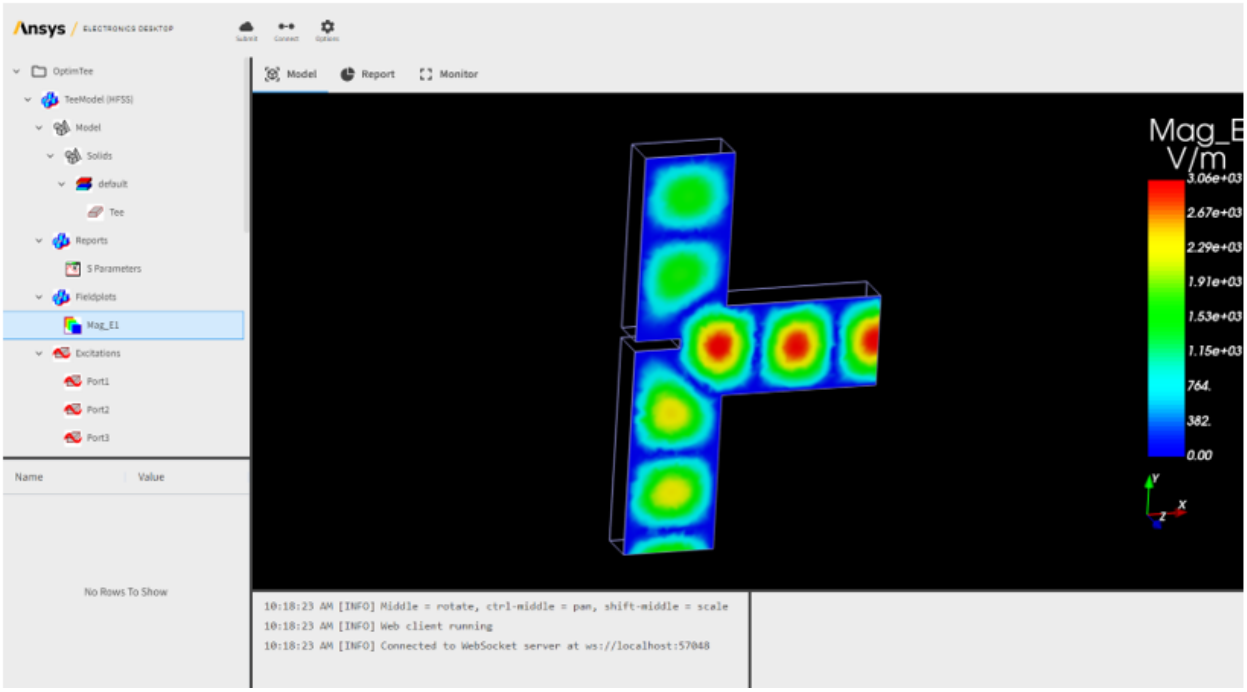
Select a Report in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select a Field Plot in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aborting an Analysis

To end the solution process before it is complete:

- Right-click In the **Progress** window and click **Abort** or click on drop-down button next to progress bar(s).

  The solver ends the analysis immediately.

The solver ends the analysis immediately. The data for the currently running adaptive pass, frequency point, time step, or variation is lost. Previously completed solutions may or may not be retained (depending on what process was interrupted by the **Abort** command). If you want to ensure that the results of all completed solutions are saved, use **Clean Stop** instead of **Abort**.

To abort the solution process after the current convergence iteration is complete:

- Right-click the **Progress** window, or click on drop-down button next to progress bar(s) and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running adaptive pass, frequency point, time step, or variation has been solved. Solutions completed before the stop request are retained.

Linux

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Linux, you can use TERM commands. Sigterm handling is done in the Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

Distribution Command Line Options

The user should include options specifying how the job should be distributed in the command line. These command line options will override options specified using the **HPC and Analysis Options** dialog box or obtained from the registry. See Running Ansys Electronics Desktop from a command line for more information on command line options.

For both scheduler batch jobs and interactive scheduler jobs, the -MachineList num=<num distributed tasks> format is the most common way to specify the number of tasks for the job. The other formats (-MachineList list=... or -MachineList file=...) allow the user to specify the number of tasks and cores to use on each host. These formats may be useful with clusters of heterogeneous machines, by allowing the user to specify different numbers of tasks or cores for different hosts. If either of the latter two formats is used, the user must ensure that the hosts and cores specified on the product command line are compatible with the hosts and cores allocated to the job.

Batchoptions for Interactive Scheduler Jobs

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

Batchoptions

Batchoptions may be specified in the command line used to launch the product. Any valid batchoptions specified in the command line will override the associated registry settings. Batchoptions also override options specified using **HPC and Analysis Options** window or other dialog boxes used to specify options.

Integration with Ansys Remote Simulation Management (RSM)

Ansys Electromagnetics supports its own Remote Simulation Management (RSM) software along with other High Performance Computing (HPC) software management programs (see High Performance Computing (HPC) Integration).

When do you need RSM?

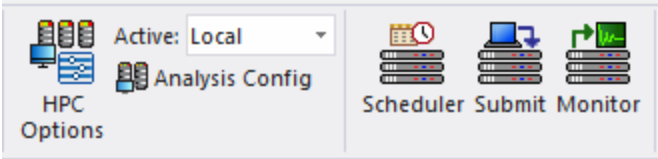
RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, see the *Ansys Electromagnetics Installation Guides*.

Job Management UI for RSM

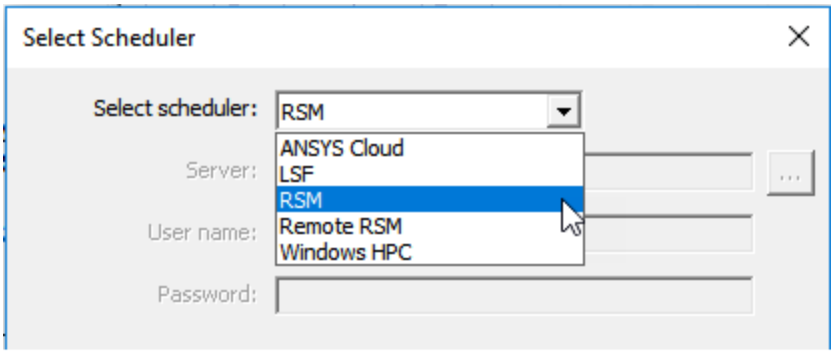
You can use the Job Management UI to submit batch jobs to RSM. The Job Management UI is accessed by running Ansys Electronics Desktop on the designated 'Postprocessing node' of the cluster. The Desktop provides UI commands for Scheduler selection, Job submission and Job monitoring/control. You access the Scheduler User Interface by clicking **Tools** > **Job Management** > **Select Scheduler...**

You can also select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



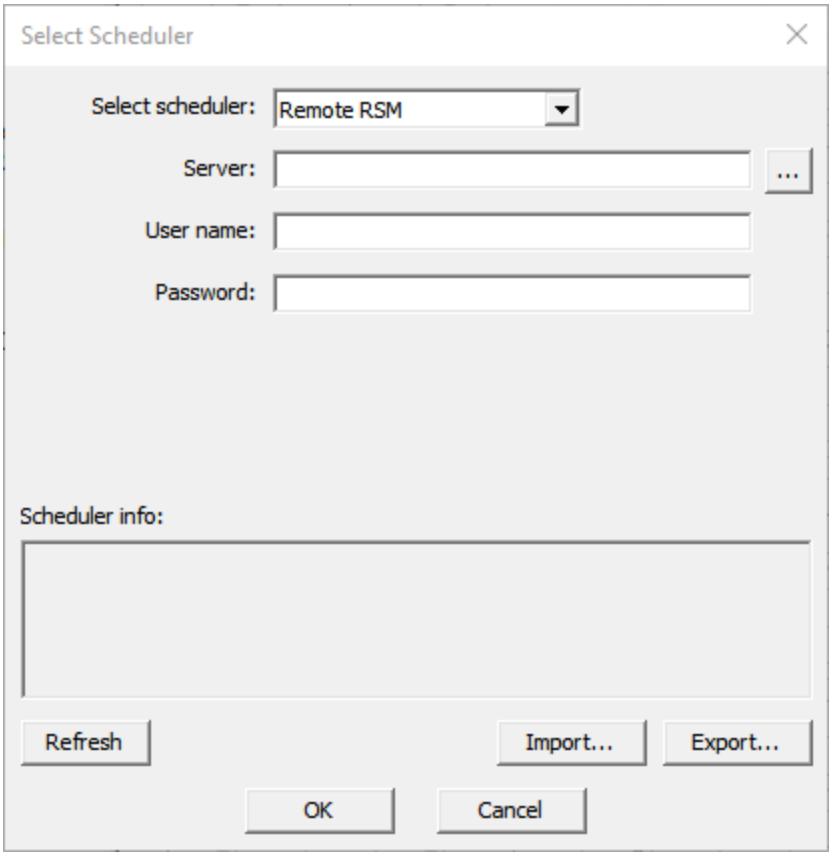
This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (which can include RSM, LSF, Windows HPC, or SGE, depending on the environment).



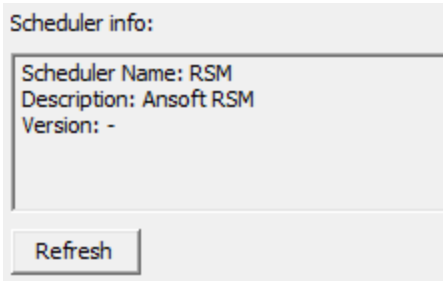
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured , you can select a computer, user name, and password.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a scheduler, click **Refresh** to display information for that scheduler.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the RSM environment, you are required to specify a machine-list. See: HPC and Analysis Options. For example, say the machine-list is: 3 cores from 'm1' and 3 cores from 'm2', for a total of 6 engines. You select the list on the **Compute Resources** tab of the **Submit Job to RSM** window, as described below.

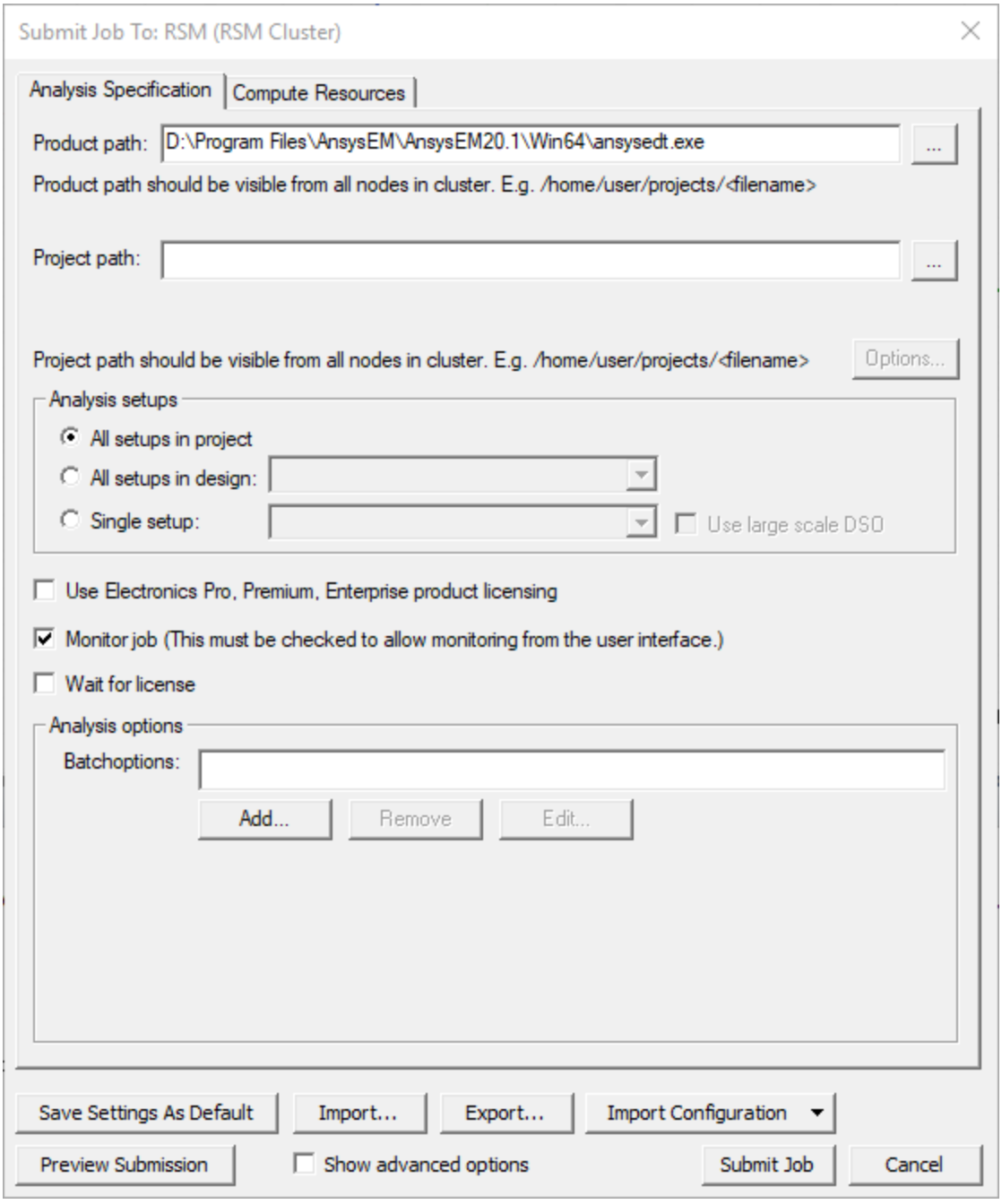
3. Open a remote-desktop session (or equivalent such as vnc session) on the node corresponding to the first machine of job's machine-list, 'm1' in this case. Launch Desktop graphically on 'm1'.
4. After setting the job submission node, select **Tools** > **Job Management** > **Submit Job...** or **Project** > **Submit Job...** or **[ProductName]** > **Submit Job...** to open the **Submit Job To:** window. You can also access **Submit Job** from the shortcut menus for the Project Name, Design name, Analysis Setup, or Optimetrics Setup.

   The **Submit Job To:** window contains two tabs:

   - **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
   - **Compute Resources** – specify the amount of compute resources and how to select specific resources from those available, and automatic settings, if supported by the design types.

   The standard Job Submission window displays.

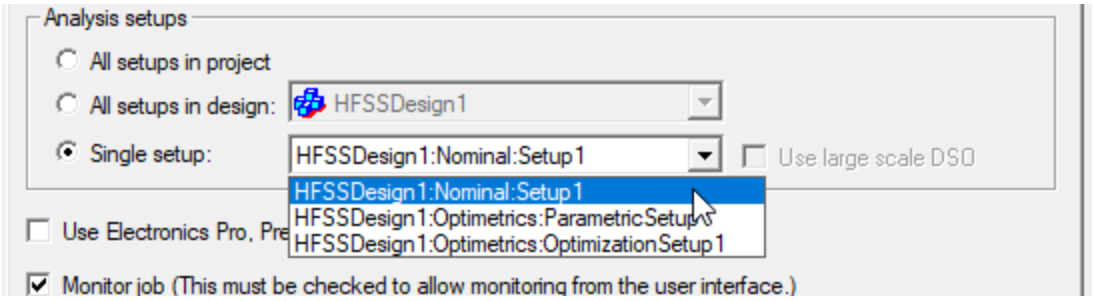
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the ellipsis button [**...**] to open a browser to select the project. The project can be an archive.
6. In the **Analysis setups** area, you can select radio buttons for **All setups in the project**, **All setups in the design**, or a **single setup**. For instance, the OptimTee example includes setups for Nominal, Parametric, and Optimization. If you accessed the **Submit Job** window from the right-click menus for Setup or Optimetrics Setup, this field can be

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

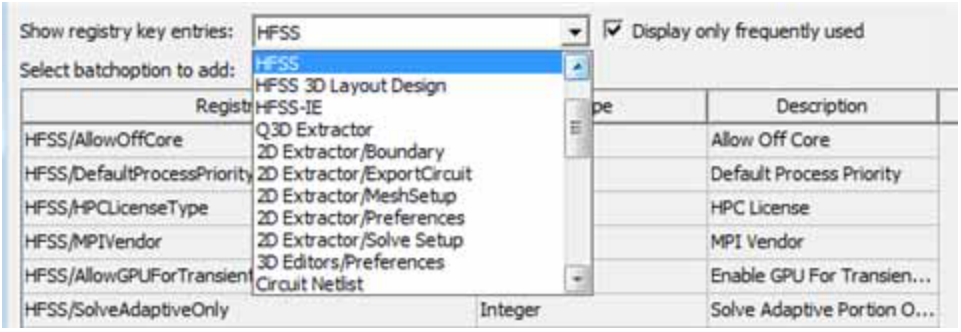
pre-populated.



For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

7. The Analysis options include check boxes for Monitoring the job, whether to wait for a license, and a field for adding Batchoptions via a graphical interface, or as text.
    - If you intend to monitor the job through a user interface, you must select **Monitor job**. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** command or by checking the dialog that opens when you submit the job.
    - The Batchoptions field allows you to add additional -batchoptions parameters, either as text, or by using a dialog with selection menus. Click the **Add** button to view the **Add Batchoption** dialog box.

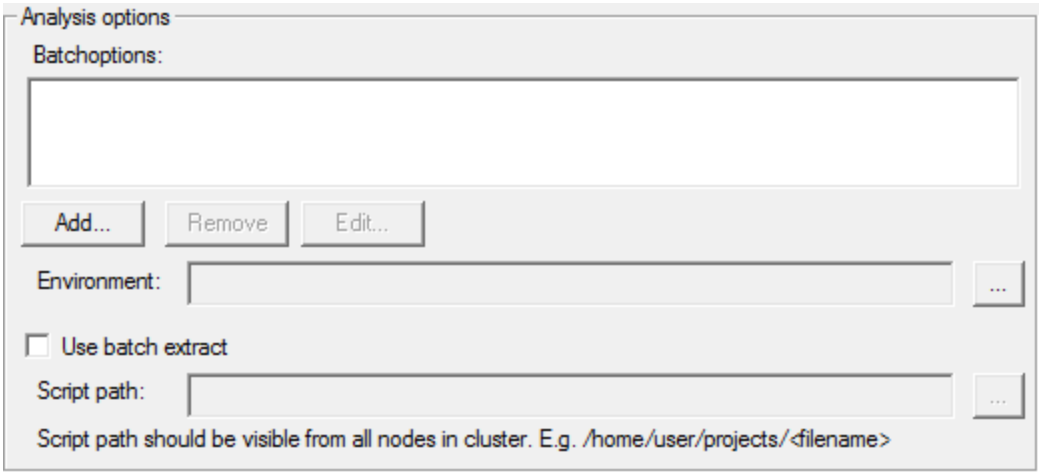
The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.



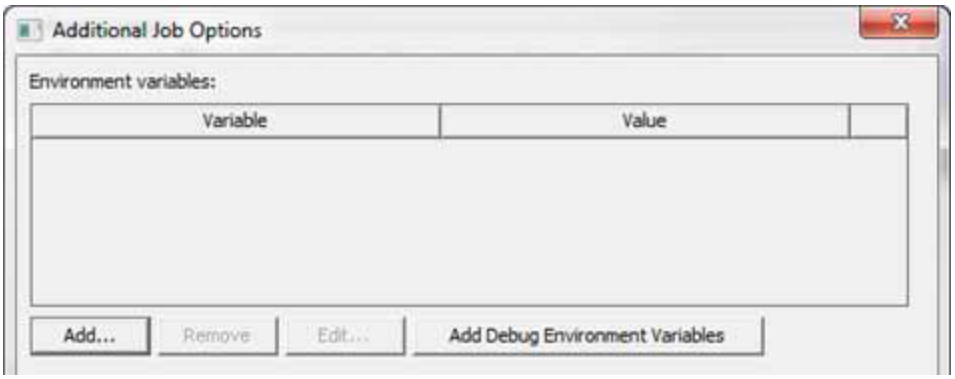
- When you have selected a batchoption, you can type the value in the field, and click the **Add** button to add the option to the batchcommand.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and whether to **Use batch**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

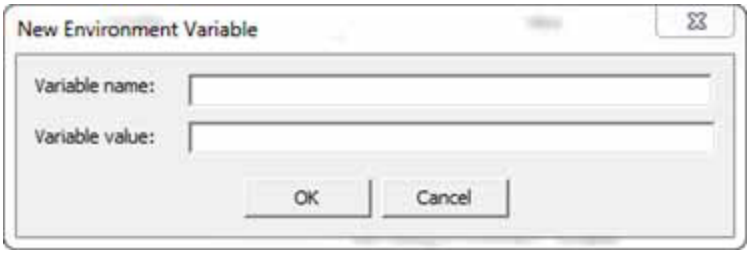
extract.



The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

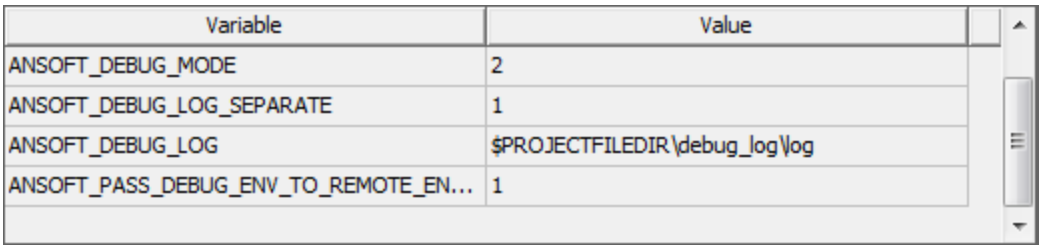


The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



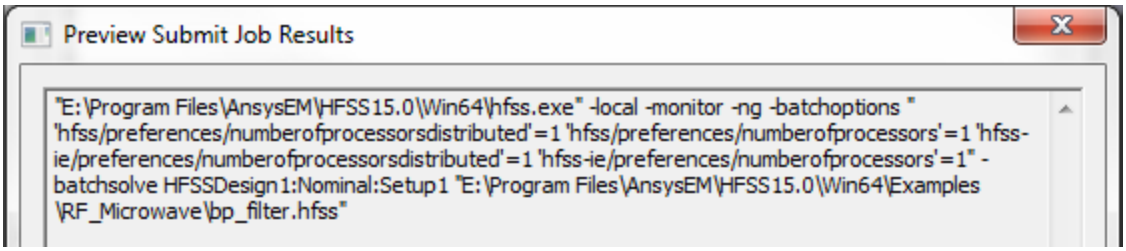
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



Selecting a variable in the dialog enables the **Remove** and **Edit** buttons. The **Edit** button opens a dialog box where you can change the variable and value.

8. To see the command-line to be submitted to the scheduler, click **Preview Submission.** This opens a dialog box showing the command to be sent to the scheduler.

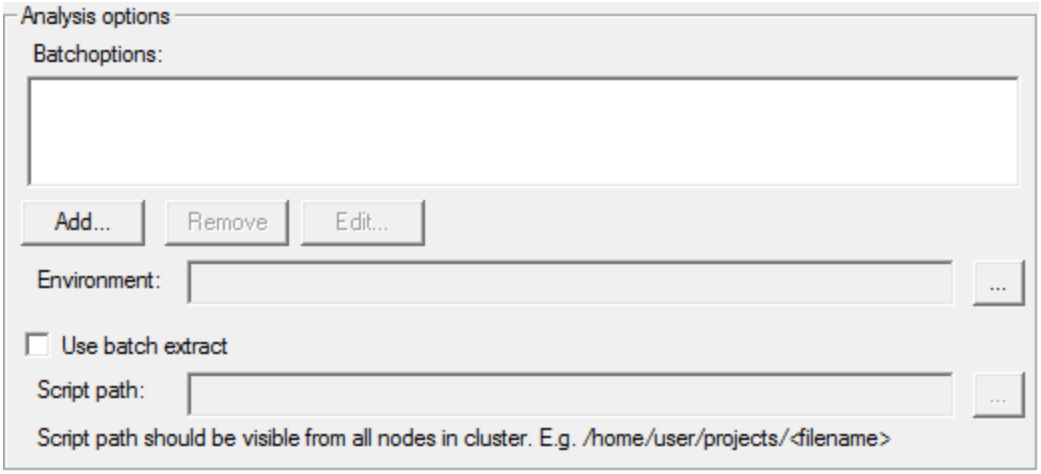


The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

Selecting **Show advanced options for RSM** also shows the Use batch extract fields.

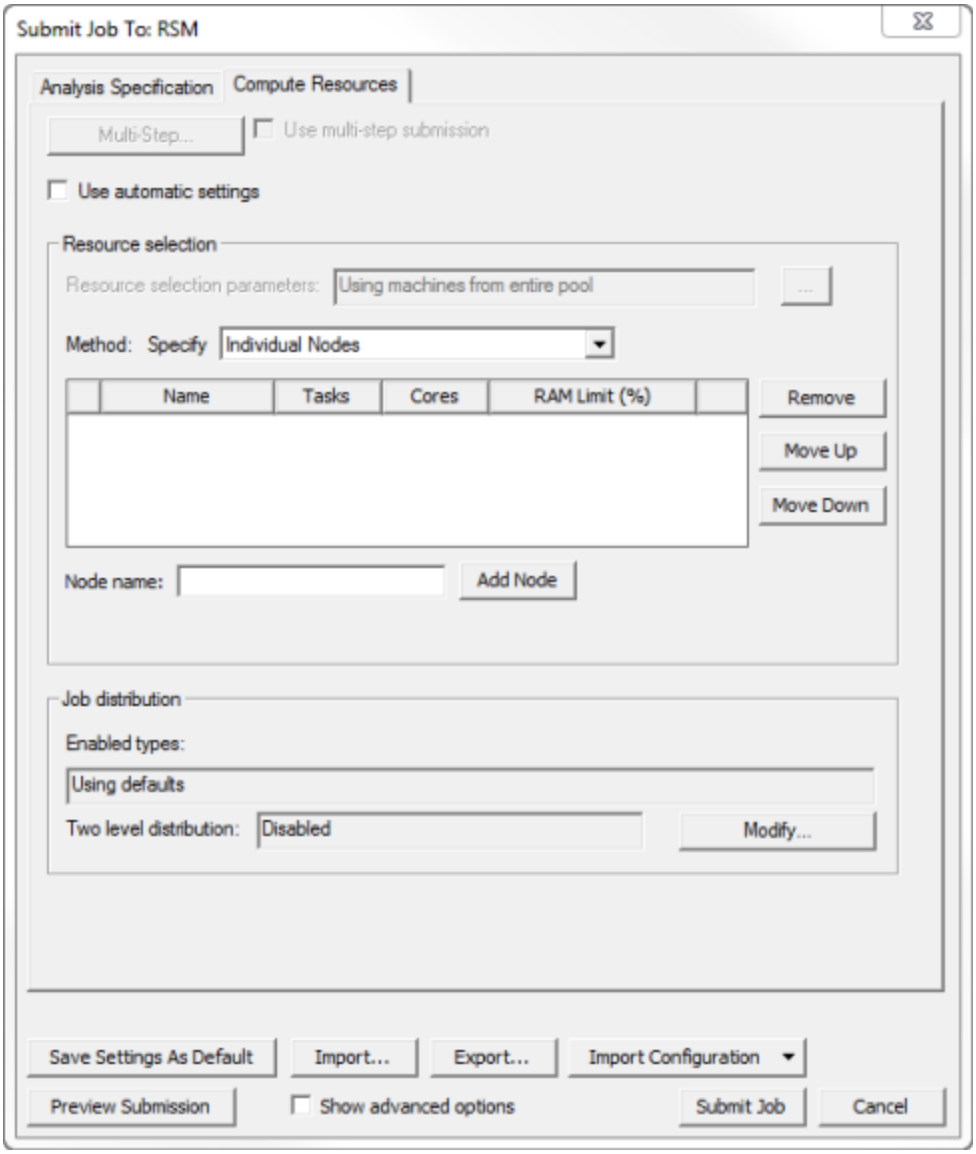
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

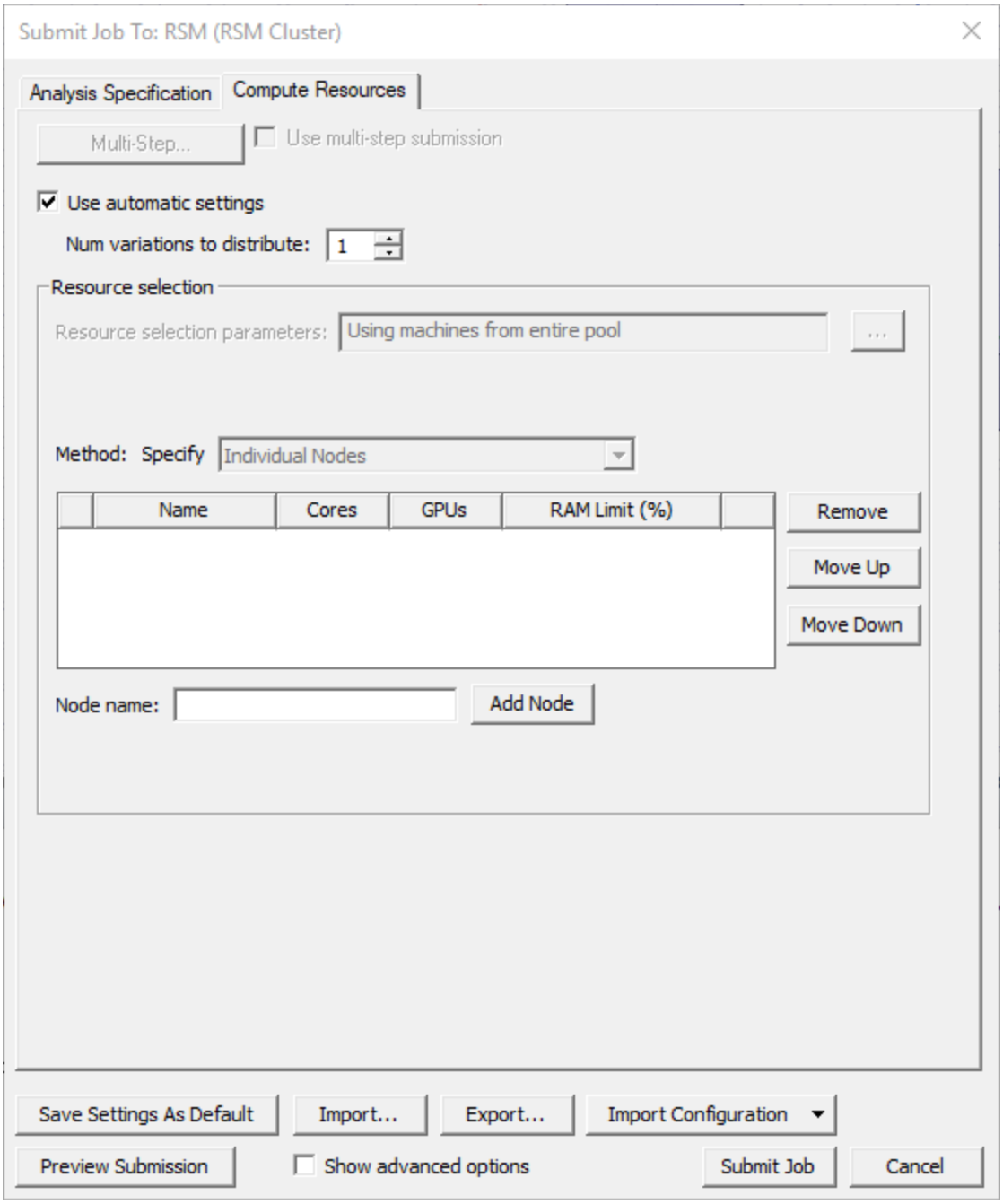
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution field** is removed and the **Use automatic settings** check box and **Num variations to distribute** field appear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



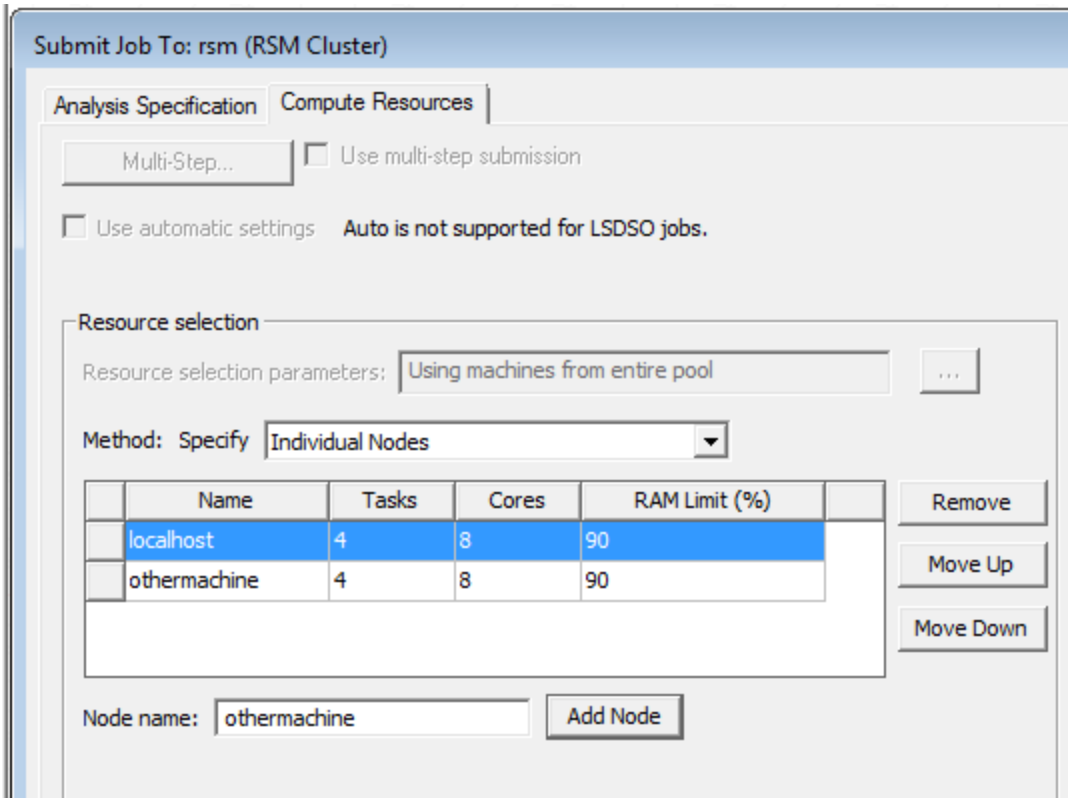
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Otherwise:

- Specify node list

  Here you can specify a node list. In a computing environment where the available cores are not uniform, you can use this to control which resources your job will use. For use with Large Scale DSO for RSM, for jobs that are submitted from job submission panel, localhost must be the first node in the resource selection node list, other wise LSDSO solve with RSM will fail.

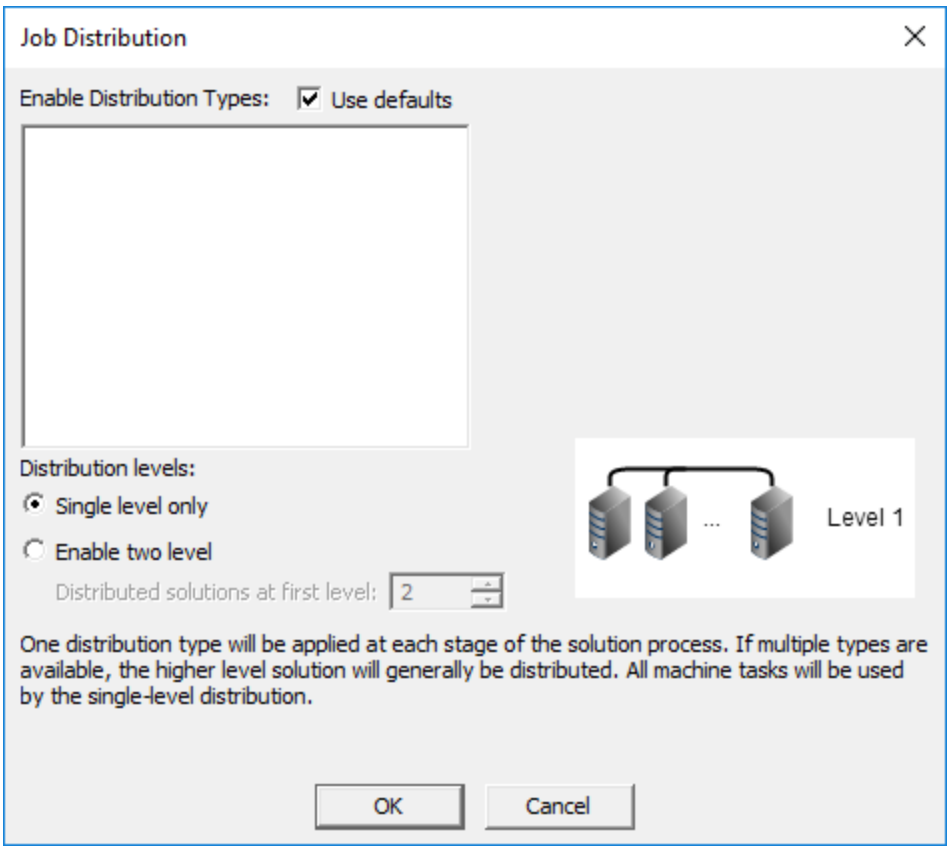


Job Distribution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you disable User automatic selection, you can modify the Job distribution settings.

- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.

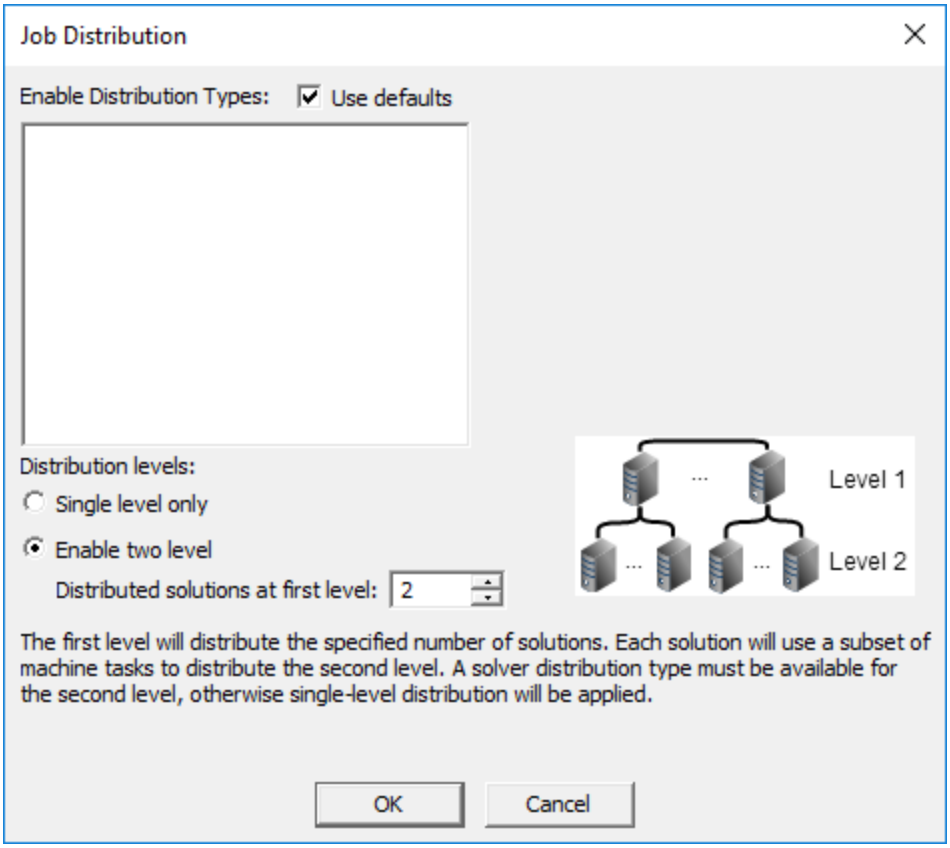


> **Note:**
>
> Two-level distribution is not currently supported for Mechanical designs. Keep the default **Single level only** option for *Mechanical – Modal* and *Mechanical – Thermal* solutions.

Enabled distribution types can be modified here.

Second-level distribution operates within DSO. If available and enabled you can specify a number of engines for level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



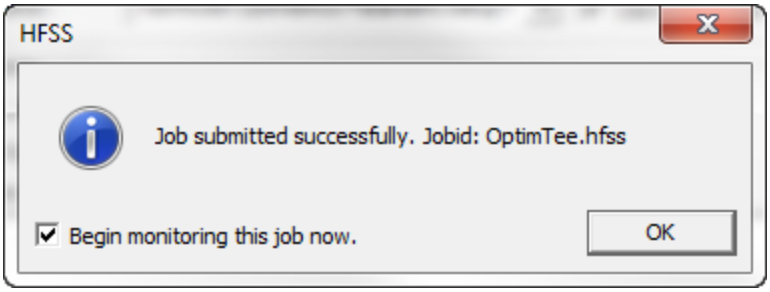
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

1. To submit the command with the specified parameters, click **Submit Job**.

> **Note:**
>
> The RSM environment does not support for queuing, so **Submit Job** will immediately start running the job.

A dialog box displays. You can select **Begin monitoring this job now** and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can monitor this job either automatically (by selecting the option above) or through the **Tools** > **Job Management** > **Monitor Jobs...** command. For more details, see Monitor Jobs window.

Process for Changing the Listening Port used by AnsoftRSM Service

To change the listening port used by the AnsoftRSMService, you need to change the configuration file, ansoftrsmservice.cfg, as follows:

You must specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file, the 'AnsoftCOMDaemon' block. The following example shows the listen port changed from 32958 to 32957, with these blocks at the beginning of the file:

```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

Changing the AnsoftRSMService Listening Port

For Remote Analysis or Distributed Analsysis, processes may need to be started on multiple hosts. If the Ansys Electromagnetics Desktop needs to start a process on a remote host, the AnsoftRSMService is used to start these remote processes. By default, the AnsoftRSMService listens for socket connections from the Ansys Electromagnetics Desktop on port 32958.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section describes how to change the port number used by the AnsoftRSMService.

To change the port number, both the AnsoftRSMService and the Ansys Electromagnetics Desktop must be configured to use the new port number. The same port number must be used for Ansys Electromagnetics Desktop and for the AnsoftRSMService process running on each host used for the analysis.

AnsoftRSMService Configuration

The AnsoftRSMService port number is configured in the ansoftrsmservice.cfg configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

The default location of this directory is "C:\Program Files\ANSYS Inc\v251\EM_RSM" on Windows.

The default location of this directory is /opt/ansys_inc/v251/EM_RSM on Linux.

To modify the AnsoftRSMService configuration, first stop the ansoftrsmservice, then modify the ansoftrsmservice.cfg configuration file, then restart the ansoftrsmservice.

> **Note:**
>
> When using Linux, root maintains exclusive control over systemd-managed services. Once the startonboot command has been given, only root users can start and stop AnsoftRSMService, using the commands:
>
> ```
> sudo ansoftrsmservice start
>
> sudo ansoftrsmservice stop
> ```

The beginning of the configuration file should appear as follows:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If there are additional lines between the following two lines, then they should not be modified:

```
$begin 'AnsoftCOMDaemon'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$begin 'Default:CommDetails'

To change the port number, modify the ListenPort setting within the single quotes from 32958 to the desired port number. The single quotes should not be removed or changed.

For previous versions of the software, the ansoftrsmservice.cfg file may not contain the lines:

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32958'

$end 'CommDetails'

$end 'Default:CommDetails'

If these lines are not present, then add them to the ansoftrsmservice.cfg file after the first line of the file, and change the ListenPort to the desired port number. For example, to change the port number to 32000, the beginning of the ansoftrsmservice.cfg file should look like the following after the changes:

$begin 'AnsoftCOMDaemon'

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32000'

$end 'CommDetails'

$end 'Default:CommDetails'

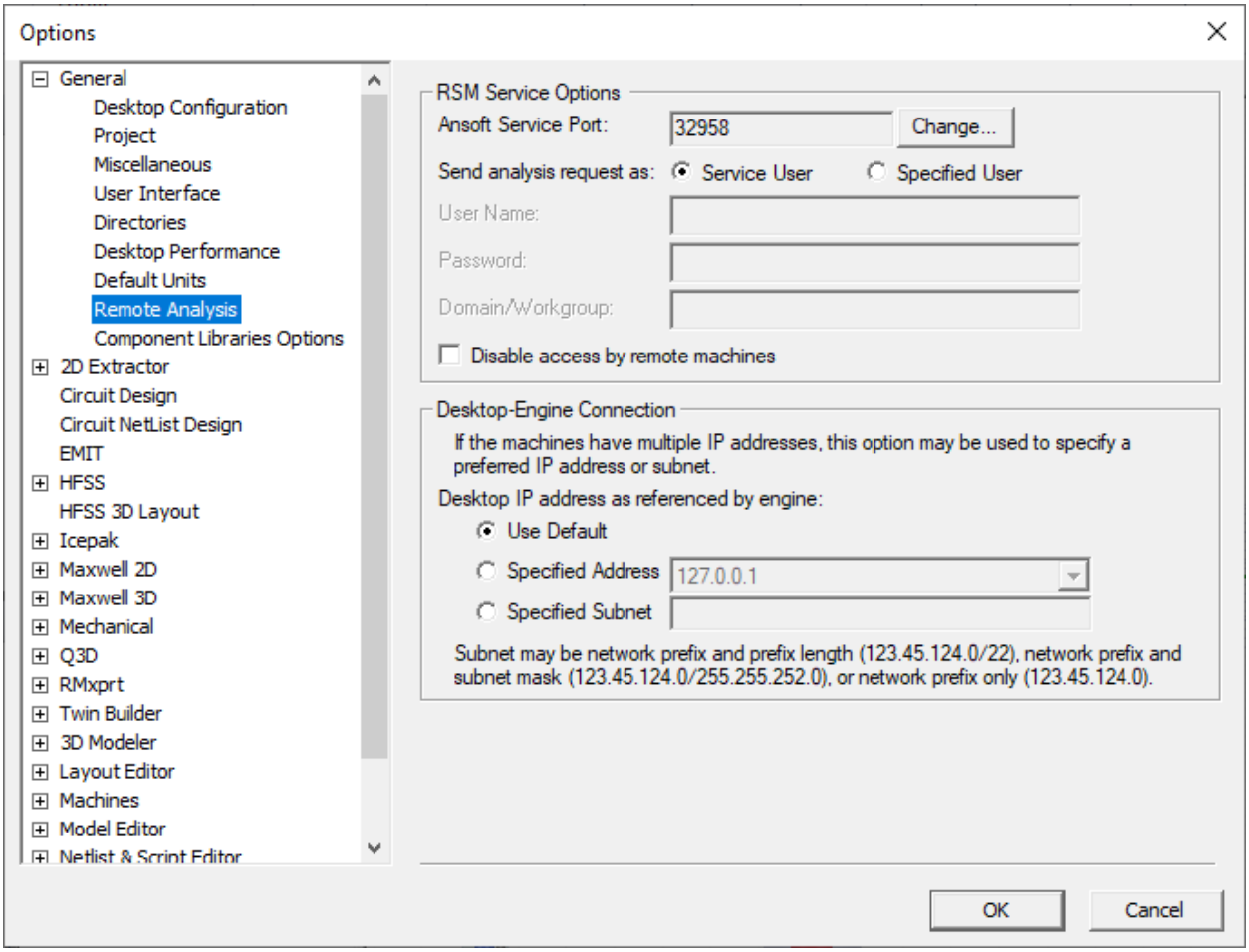
Ansoft Electromagnetics Desktop Configuration

The port number for connecting to the AnsoftRSMService is configured from the **General Options** window.

To access these options:

- Select **Tools** > **Options** > **General Options** to open the **General Options** window.
- If it is not already expanded, expand the **General** tree and click **Remote Analysis**.

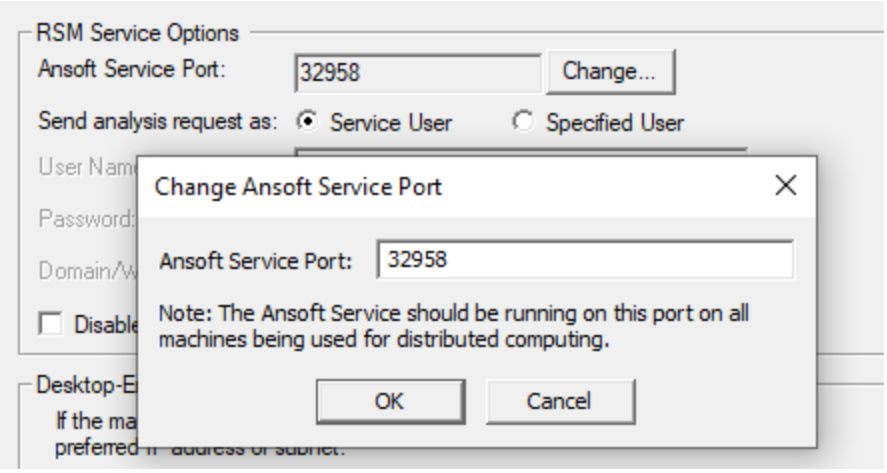
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under **RSM Service Options**, the **Ansoft Service Port** field shows the current AnsoftRSMService port number.

1. Click **Change** to modify this setting. This opens the **Change Ansoft Service Port** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter the new port number in the **Ansoft Service Port** field.
3. Click **OK** to close the dialog box and accept the new port number.
4. Click **OK** to close the **General Options** window and accept the changes.

Running HPC Diagnostics

The Ansys EM HPC diagnostics tool simplifies HPC troubleshooting by automating diagnosis of routine issues. The diagnostics tool is run on the cluster as a scheduler managed job. Using its HTML based diagnostics report, cluster administrator or Ansys support staff can either resolve the issue, or guide the user with steps for further troubleshooting. In some cases, Ansys support staff may request to rerun the diagnostics with additional diagnostics tests. The user may extend the diagnostic scripts to suite their HPC environment.

This note describes how to use the diagnostics tool.

- Supported schedulers
- Running the diagnostics job
- Standard diagnostic job
- Using diagnostics scripts on Linux clusters
- Using Windows HPC job file
- Diagnostic report
- Site-specific diagnostics job
- Environment variables
- ANSYSEM_DIAG_PROD_DIR contents
- ANSYSEM_DIAG_RESULTS_DIR contents
- How does the diagnostic tool work

Supported Schedulers

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The tool supports diagnosis of issues on Linux and windows clusters managed by the following schedulers:

- LSF
- SGE
- PBS/Torque
- Windows HPC

For the above schedulers (see High Performance Computing (HPC) Integration), the tool includes basic diagnostic scripts. Further, if password-ssh has been enabled, it also supports generic Linux clusters using ssh. Please note that currently diagnostics tool does not support PBSPro and LSF/Windows.

Running the Diagnostics Job

The diagnostics are run as a scheduler managed job. Once the job finishes, you locate the resulting HTML file and provide it to the cluster administrator or to Ansys support staff. In case, there are any job or test failures, please also provide the networking*.json files from the Hosts subdirectory as well.

Basic Diagnostic Job

To run the basic diagnostics, submit a diagnostic job to the scheduler using a provided job submission script. Each basic diagnostic job is a 12 core job with 4 cores per host. On Linux, running this script submits a scheduler job to run the diagnostic tool on the cluster. On Windows, you need to submit a job using a job file.

Basic scripts for each supported scheduler are available in diagnostics subdirectory of schedulers directory.

Linux:

```
opt/ansys_inc/v251/AnsysEM/schedulers/diagnostics
```

Windows:

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers\diagnostics
```

Using Diagnostics Scripts on Linux Clusters

The following basic scripts are provided in the diagnostics directory (.../ansys_inc/v251/AnsysEM/schedulers/diagnostics):

These job submission scripts are scheduler specific.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scheduler	Basic job submission script	Comment
LSF	test_lsf	Supports both lsrun and blaunch
SGE	test_sge	Supports both qrsh and rsh
PBS/Torque	test_torque	Requires changing the PATH and PBS_BINARY_PATH environment variable
Generic Linux cluster	test_ssh	Supports only ssh. Requires password-less ssh. Requires creating a file with the names of hosts and saving it in ${HOME}/ansysem_hostfile

Using Windows HPC Job File

A sample job file winhpctest.xml is available in the diagnostics directory:

`...\schedulers\diagnostics.`

To submit this diagnostic job, you must change the job description to suite your environment as following:

1. Select a directory for saving the diagnostic results. This directory must be accessible at the same path from all the hosts of the cluster.
2. Locate the directory for Ansys EM installation. This directory also must be accessible at the same path from all the hosts of the cluster.
3. Locate the winhpctest.xml in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
4. Start Windows HPC job manager, and choose "New job from XML File…" action.
5. Select the winhpctest.xml job file.
6. Change the value of both the following environment variables with the directories located in the first two steps:

   ANSYSEM_DIAG_PROD_DIR

   ...

   ANSYSEM_DIAG_RESULTS_DIR
7. Now submit the job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> After making the above changes, you can also save the resulting XML file using "Submit Job XML File…". Then you can submit the job using the job command as following:
>
> job submit /jobfile: XMLfile name

Diagnostic Report

The diagnostic report is an HTML file that (along with other related diagnostics results) is placed in the following directory

Linux:

```
${HOME}/Ansoft/HPCDiag/Results/JOBID
```

Windows:

```
%ANSYSEM_DIAG_DIR%\Results\JOBID
```

Report file:

```
.../HTML/report.html
```

where JOBID is the job ID assigned by the scheduler. On Windows, the user must specify ANSSEM_DIAG_DIR directory.

Site-Specific Diagnostics Job

To run a diagnostic job with job submission parameters of your choice, you need to create your own job submission script. For example, you may want to specify a different LSF queue, or select a different SGE parallel environment. To run such a job, you need to create your own job submission script starting from the basic diagnostic scripts with the following steps:

1. Locate the relevant basic diagnostic script in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
2. Make a copy of the diagnostics script into a directory that is accessible from a submit host for the cluster.
3. Edit the script file to change the value of ANSYSEM_DIAG_PROD_DIR environment variable to point it to the installation directory (See below).
4. Modify the job submission parameters as needed.
5. Optionally, copy any site-specific diagnostic tests provided by Ansys support staff in the *../Custom* subfolder of the *ANSYSEM_DIAG_RESULTS_DIR* directory.
6. Run the diagnostics script from a submit host for the cluster.

Environment Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following environment variables are applicable for both Linux and Windows environment.

ANSYSEM_DIAG_PROD_DIR

Environment variable	**ANSYSEM_DIAG_PROD_DIR**
Description	Location of the Ansys EM installation. This must be available at the same path from all the hosts of the cluster.
Windows example	"\\filer\AnsyEM\v251\AnsysEM"
Linux example	/shared/ansys_inc/v251/AnsysEM
Comments	Windows: Required. Linux: Optional. Export this environment variable if you make a copy of the diagnostic script.

ANSYSEM_DIAG_RESULTS_DIR

Environment variable	**ANSYSEM_DIAG_RESULTS_DIR**
Description	Location of the diagnostic report and other results on a shared drive. This must be available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag
Linux example	/shared/home/user/Ansoft/HPCDiag
Comments	Windows: Required. Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

ANSYSEM_DIAG_CUSTOM_DIR

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Description	Location of the configuration of product tests and other custom site-specific tests. This location must be on a shared drive that is available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag\Custom
Linux example	/shared/home/user/Ansoft/HPCDiag/Custom

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Comments	Windows: Optional. You may want to specify it if the path %ANSYSEM_DIAG_RESULTS_DIR%\..\Custom is not suitable Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

How the Diagnostic Tool Works

The diagnostics are run as a scheduler managed job. Running the diagnostic script submits a scheduler job that runs the diagnostic tool on the hosts allocated to the job. Once the diagnostic job starts, the tool executes a set of diagnostic tests. These tests run on each host allocated to the job, and collect diagnostic information relevant for running HPC jobs. The tool combines the diagnostic information to produce an HTML report. The tool saves HTML diagnostic report and other results in a shared drive, which must be available at the same path from all the hosts of the cluster. On Linux, the default is Ansoft/HPCDiag subdirectory under user's home directory. On Windows, the user must specify this location using ANSYSEM_DIAG_RESULTS_DIR environment variable.

Changing Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
    - To affect priority for future simulation runs, click **Tools** > **Options** > **HPC and Analysis** to access the **HPC and Analysis** dialog box, and click the **Options** tab.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lowest Priority	
Below Normal	
Normal	The default.
Above Normal	
Highest	

3. Click **OK**.

desktopproxy Utility to Abort or Cleanstop a BatchSolve

The desktopproxy Utility to abort or cleanstop a batch solve is available in the installation directory: ("...\ANSYS Inc\v251/AnsysEM" on Windows, ".../ansys_inc/v251/AnsysEM" on Linux).

To abort a job, run it on a host which has network access to the first execution host of the job, as well as has access to the directory of the project file. This utility does not abort Large scale DSO jobs for which you need to use desktopjob utility itself.

Usage:

```
desktopproxy -abort <projectpath>
-cleanstop <projectpath>
```

Where the <projectpath> is the full path to the project file.

The behavior (e.g., saving the results computed so far) should be same as if you interactively aborted AnsysEDT solve. if AnsysEDT is 'busy' (e.g., doing Optimetrics calculation, computing a report), abort (using desktopproxy) might take a long time.

The utility exits silently after it requests ansysedt process to abort the job cleanly. After the job ends, the batch log file may contain a message such as:

```
[error] Project:TestProject, Design:HFSSDesign1 (DrivenModal),
Simulation was aborted by user on server: sjohpc-lsf-0.ansys.com.
(3:52:30 PM Oct 11, 2013)
```

The lock file will be automatically deleted.

Example (Linux):

```
desktopproxy -abort /home/user/test/Diff_Via.aedt
```

Example (Windows):

```
desktopproxy -abort \\server\share\projectfolder\Diff_Via.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

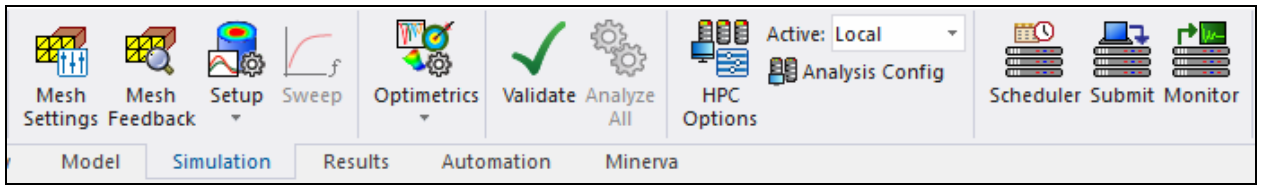
Limitations:

- Aborting a job makes a best effort to delete all the temp directories on all the hosts
- For a hung job, a clean shutdown may not be possible. In that case, the user must use OS/scheduler's facilities to kill the job.

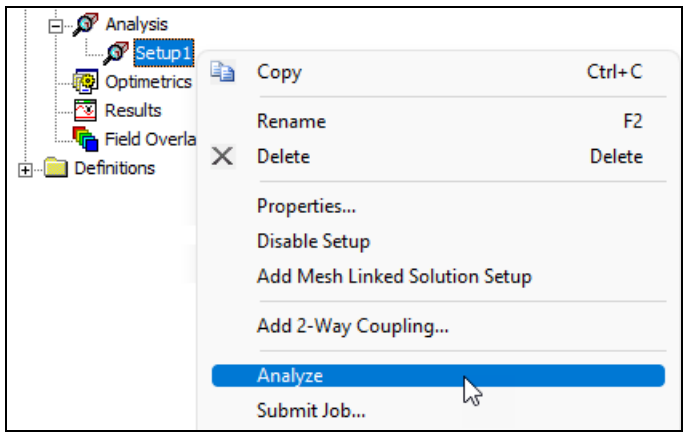
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17 - Running Simulations

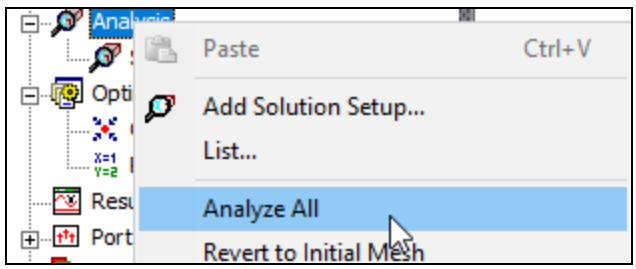
After you specify how Ansys Electronics Desktop is to compute the solution, you need to begin the solution process. You can access the **Analyze** commands for a specific setup by right-clicking on a solution or sweep in the **Project** tree and selecting from the shortcut menu, you by selecting the **Analyze All** from the solver menu, or the **Analyze All** icon on the **Simulation** tab of the ribbon.



In general, the **Analyze** command on the shortcut menu applies to the selected setup and associated sweeps, if any, or to a selected sweep. To use this command, right-click on a setup or sweep in the Project Manager and click the command on the context menu.



The **Analyze All** command applies to all, and sweeps at or below the level invoked in the Project Manager. To use this command, either click **Mechanical**> **Analyze All** or right-click the **Analysis** icon in the *Project Manager* and select **Analyze All**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

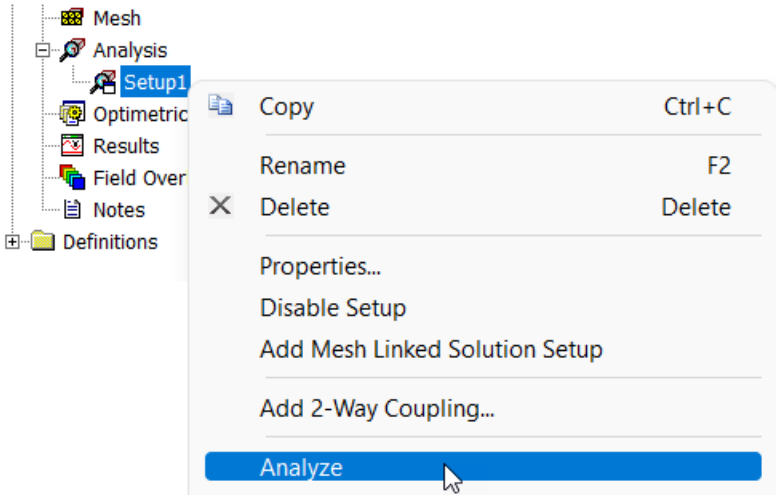
What do you want to do?

- Local Analysis
- Solve a single setup with or without sweeps (when applicable)
- Enable a queue so that multiple simulations can run sequentially as resources become available.
- Run more than one simulation, whether multiple setups, or multiple sweeps under a single setup, or setups with dependencies.
- Enable or Disable one or more solution setups or sweeps.
- Submit Job to RSM or HPC Scheduler
- Monitor queued simulations
- Monitor the solution process
- Change a solution priority for system resources
- Abort an analysis
- Re-solve after modifying a design

Solving a Single Setup

To run a single setup:

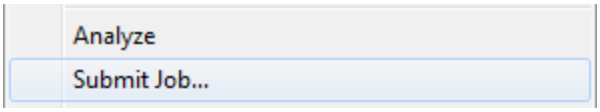
1. Select a solution setup in the Project Manager.
2. Right-click and select **Analyze** from the shortcut menu. The graphic shows a single setup selected for analysis.



The 3D field solution is computed inside the structure for a solution.

When you right-click a Setup the shortcut menu also includes the **Submit Job...** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information on the **Submit Job...** command see, Distributed Analysis and High Performance Computing (HPC) Integration.

To run more than one analysis at a time, follow the same procedure while a simulation is running. If you have enabled queuing, the next solution setup will be solved when the previous solution is complete.

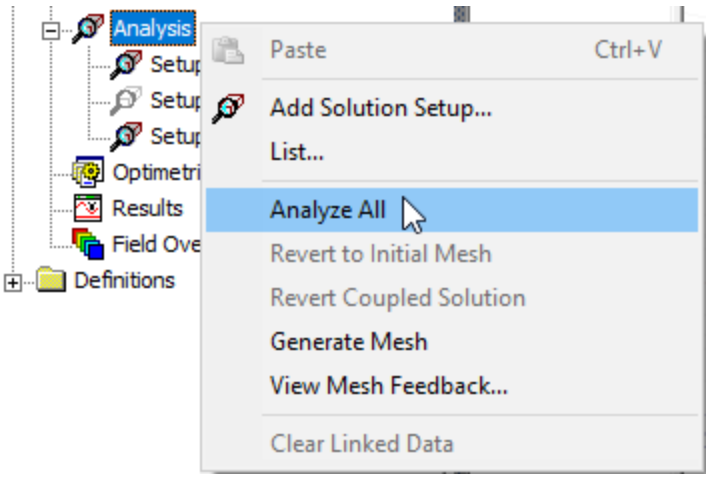
Running More than One Simulation

To solve every enabled solution setup in a design:

1. In the project tree, under the design you want to solve, select **Analysis**.
2. Click **Mechanical**> **Analyze All**.

Each enabled solution setup is solved in the order it appears in the Project Manager.

The example here shows an analysis invoked from the Project Manager shortcut menu with three setups: one disabled, two enabled.



Note:

The **General** tab for the Setup includes an **Enabled** check box. By default, this is checked. Clearing the **Enabled** check box excludes a setup from running

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can simulate all designs in a project by clicking **Project** > **Analyze All**.

Monitoring Queued Simulations

If you have multiple setups for a design and have selected **Analyze All**, the simulations can be queued until there is a machine available. You enable queuing in the HPC and Analysis Options: Options tab. If queuing is enabled, and you run multiple setups, they are solved in the order that they appear in the Project Manager. You can prioritize setups by changing their order in the queue.

1. To view the solution queue, click **Tools> Show Queued Simulations**.

   This displays a dialog box showing all the simulations and their current status. You can select and remove any simulation from the queue.

   You can also select any setup and use the **Move up** and **Move down** buttons to prioritize them.

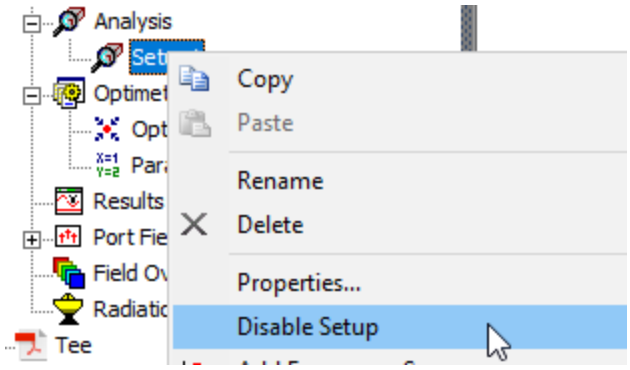
2. To remove a simulation from the queue, select the simulation and click **Remove from Queue**.

   This removes the selected simulation from the queue.

Disabling and Enabling an Analysis Setup

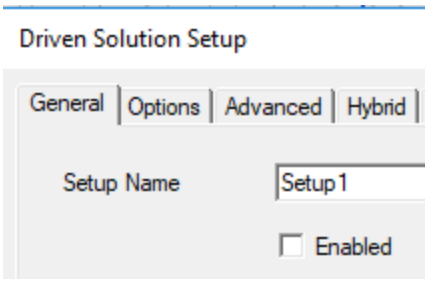
To disable an Analysis setup definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the Project Manager.
2. Right-click the icon for the setup definition. In the shortcut menu that appears, click **Disable Setup**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also disable an analysis setup by selecting **Properties** from the shortcut menu to open the **Edit Properties** window. In the upper right corner of the **Edit Properties** window, uncheck the **Enabled** box. Then click **OK**.



To reactivate the analysis setup, open the shortcut menu again and select **Enable Setup**.

You can also reactivate the analysis setup by selecting **Properties** from the shortcut menu for the sweep to open the **Edit Properties** window. Check the **Enabled** box in the upper right corner, and click OK to apply the change and close the window.

Monitoring the Solution Process

While a simulation is running, you can monitor the solution's progress in the **Progress** window. The progress bar shows the relative progress of the simulation.

You can also view the following solution data at any time during or after the solution:

- Mesh statistics (numbers of elements, lengths, volumes, and volume standard deviation; all on a per-object basis).
- A profile showing the status of the analysis process, elapsed time, memory usage, and more

To view the *Solutions* window:

1. Right-click the solution **Setup** in the Project Manager.

   A shortcut menu appears.

2. Select **Convergence**, **Solutions**, **Mesh Statistics**, or **Profile** from the shortcut menu.

   The *Solutions* window appears with the corresponding tab selected and the current data displayed.

To view the solution status:

- Click **Mechanical** > **Results** > **Browse Solutions**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Solutions* dialog box appears with the **Browse** tab selected. It contains a tree structure showing the solutions listed according to Setup, Solution, and Variation. A table lists the setup, the solution, and the state of the solution.

- You can use the **Properties** button to display a dialog box that lets you change the way the Setup, Solution, and Variation are listed in the tree structure of the *Solutions* dialog box.
- You can delete one or more solutions by selecting from the table and clicking **Delete**. Click on a solution to select it, and use Ctrl+click to select multiple solutions, or Shift+click to select a range of solutions. You can also select all solutions using the **Select All** button.

> **Note:**
>
> If a license is lost, the software waits for the license to be regained, checking every 2 minutes or until you abort.

- The **Statistics** tab of the *Solutions* dialog box displays path information as well as format, number of files, and size.

Changing a Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
    - To affect priority for future simulation runs, click **Tools> Options> HPC and Analysis** and click the **Options** tab of the resulting dialog box.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

    Highest

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Above Normal	
Normal	The default.
Below Normal	
Lowest Priority	

3. Click **OK**.

Related Topics

Monitoring Queued Simulations

Aborting an Analysis

To end the solution before the currently running process is complete:

- Right-click in the **Progress** window and click **Abort**.

  The solver ends the analysis immediately. The data for the currently running solution, variation, or time step (for transient solutions) is lost. Previously completed solutions may or may not be retained (depending on what process was interrupted by the Abort command). If you want to ensure that the results of all completed solutions are saved, use *Clean Stop* instead of *Abort*.

To abort the solution after the currently running process is complete:

- Right-click the **Progress** window and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running solution, variation, or time step (for transient solutions) has been solved.

> **Note:**
>
> Restart analyses are not currently supported for Mechanical solutions, regardless of whether a solution is stopped using **Abort** or **Clean Stop**. Partial results are written and retained but not made available for post-processing. Additionally, if you reanalyze the model, the solution starts back at the beginning. The exception is the mesh, which is retained if completed prior to stopping the solution. When you rerun the solution, the mesh is not regenerated (assuming the geometry is unchanged).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Re-solving after Modifying a Design

In some cases, if you modify a design after generating a solution, the solution in memory will no longer match the design. In such cases you receive a warning message that "Solutions have been invalidated. Undo to recover."

To generate a new solution after modifying a design, follow the procedure for running a simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18 - Postprocessing and Generating Reports

Much of the content within this branch of the help is general in nature and applicable to multiple design types (HFSS, Q3D, Icepak, and so on). For **Mechanical** designs, the types of reports and overlays that you can generate are dependent on the *Solution Type* (*Modal, Steady-State Thermal, Transient Thermal,* or *Structural*). Mechanical-specific postprocessing content is covered within the *Modal Solutions*, *Thermal Solutions*, and *Structural Solutions* branches of the Mechanical help. Use the links below to access the Mechanical-specific postprocessing content of interest:

Modal Solutions:

- Results (Modal)
- Modal Report
  - Creating a Modal Report
- Fields Report
  - Creating a Fields Report
- Field Overlay and Animation

Thermal Solutions: (Steady-State or Transient)

- Results (Thermal)
- Fields Report
  - Creating a Fields Report
- Monitor Report (*Transient Thermal* solutions only)
  - Creating a Monitor Report
- Field Overlay and Animation
- Creating a Fields Summary

Structural Solutions:

- Results (Structural)
- Fields Report
  - Creating a Fields Report
- Field Overlay and Animation
- Creating a Fields Summary

Additionally, for general information covering plot attributes, importing/exporting plot data, using the Fields Calculator, plotting the mesh, and more, drill down through the various subtopics in this branch.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Solution Data

While Ansys Electronics Desktop is generating a solution, or when it is complete, you can view the following information about the solution:

- Computing resources, or profile information, that were used during the solution process.
- Mesh statistics

To access the **Solution Data** window, in which the information above can be accessed, do one of the following:

- Click **Mechanical**> **Results**> **Solution Data**.
- Right-click **Results** in the Project Manager, and then click **Solution Data** on the shortcut menu.
- Select the **Results** tab on the ribbon, and click the **Solution Data** icon.

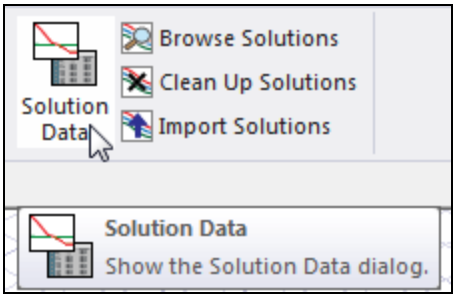
Viewing a Solution Profile

At any time during or after the solution process, you can examine the computing resources or profile data that were used by Ansys Electronics Desktop solvers during the analysis. The profile data is essentially a log of the tasks performed by Ansys Electronics Desktop during analysis. The log indicates the length of time each task took and how much physical and disk memory was required.

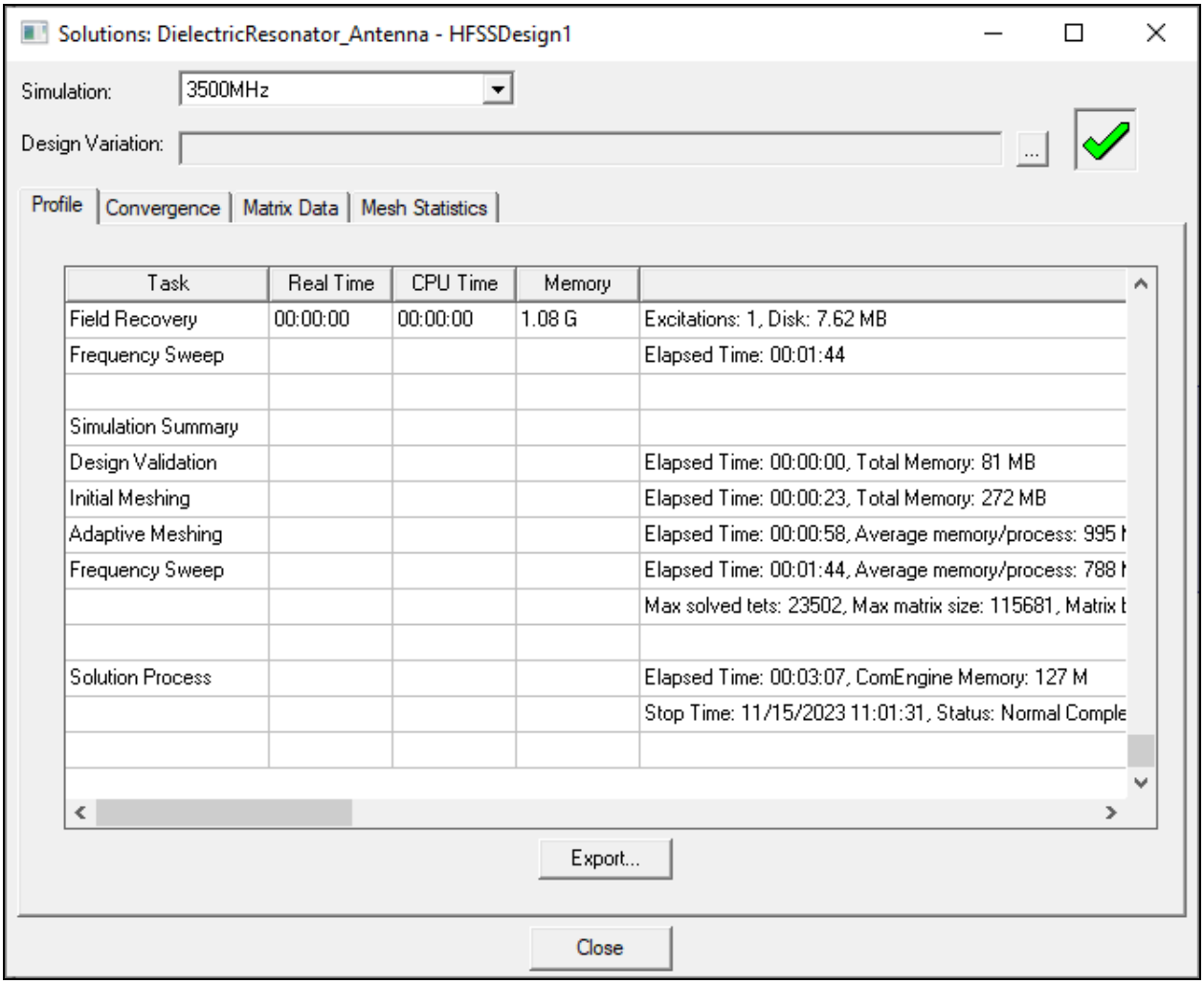
1. In the project tree, right-click a solution setup, and select **Profile**.

   The **Solutions** dialog box appears, with the **Profile** tab selected. You can also click the **Solution Data** icon on the ribbon to open the **Solutions** dialog box, and then select the **Profile** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

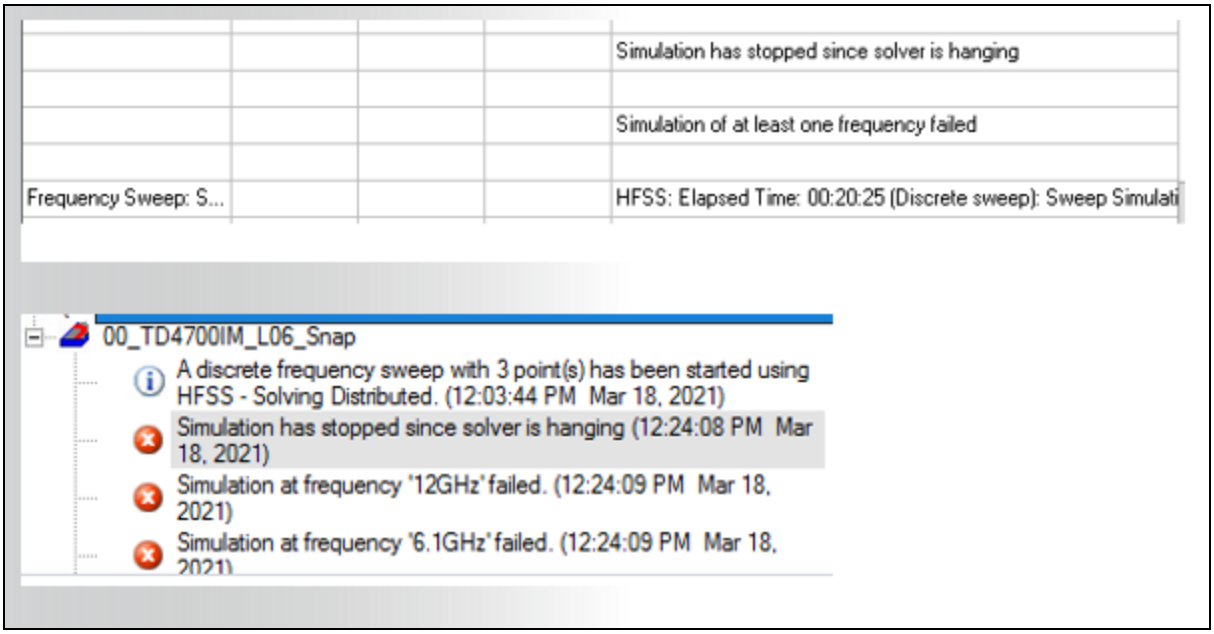


The displayed data depends on the type of problem and solution setup.



If the simulation hangs, equivalent messages appear in the Message window and the Solution **Profile** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If one or more setups exist, you can select the desired profile information from the **Simulation** drop-down menu. In general, the information displayed includes:

Task	Lists the type of task that was performed. Tasks include HPC, Design Validation, Meshing Process, Convert, Get Thermal Load, Populate Solver Input, Solve, Fields Result Recovery, and Solution Process, among others. The tasks performed depend on the solution type.
Real Time	The difference in time between the start of the task and the end of the task (elapsed time).
CPU Time	The amount of CPU time required to perform the task.
Memory	The peak amount of physical memory (RAM) used by the individual executable running the task. The memory is freed for other uses after each task is complete.
Information	General information about the solution (for example, the number of tetrahedra used in the mesh, disk use, solver information, sweep information, and totals).

To export the profile data:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open the **Solutions** dialog box and select the **Profile** tab.
2. Click **Export Profile**.

   This opens a file save dialog box that lets you provide a file name and location.

3. Click **Save**.

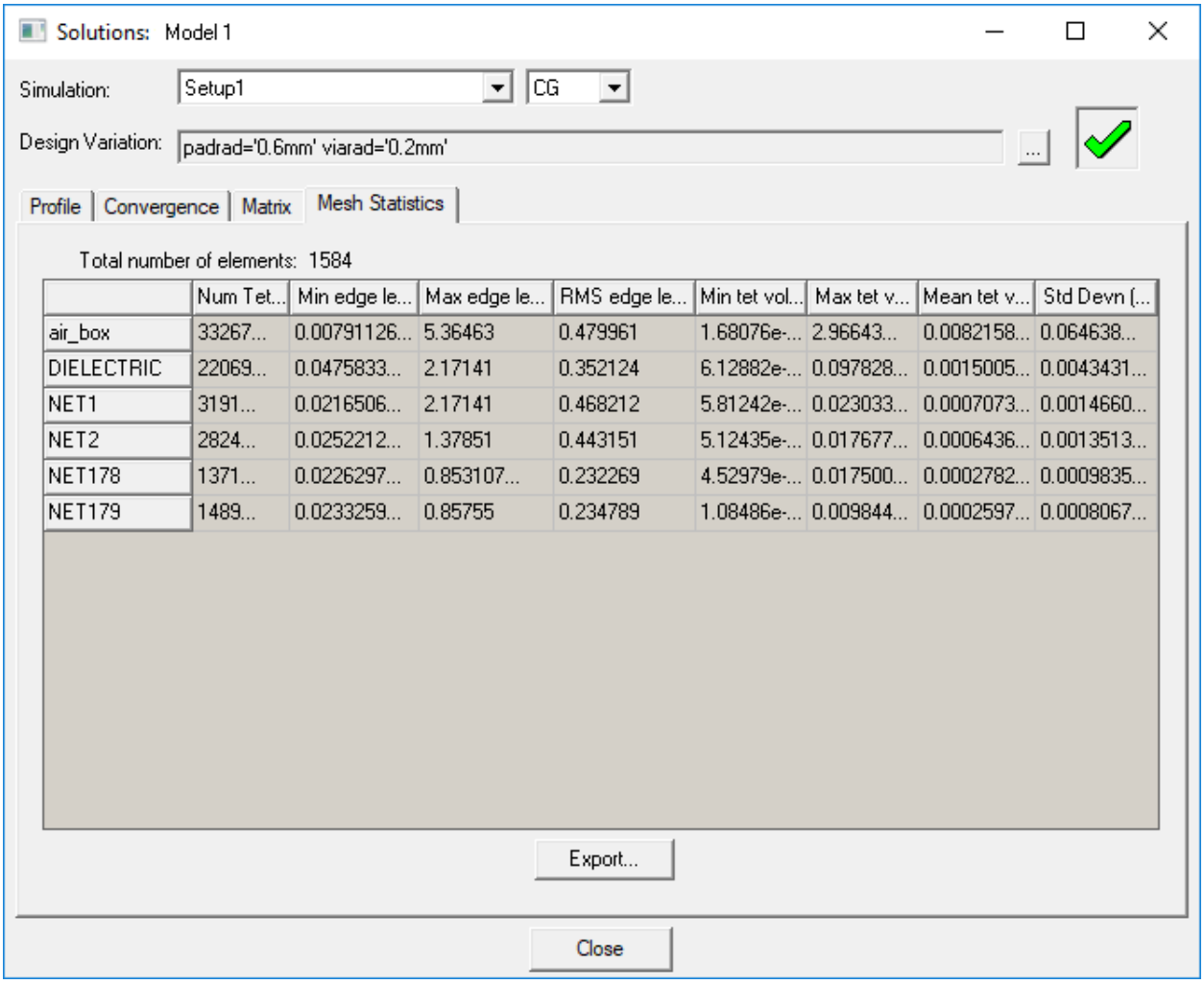
   The data is saved in a text file with a .prof extension.

Viewing Mesh Statistics

To view an adaptive solution's mesh information, either during or after the solution process:

1. In the Project Tree, right-click the solution setup of interest, and then click **Mesh Statistics**. Alternately, click the **Solution Data** icon on **Results** tab of the ribbon.

   The **Solutions** window appears with the **Mesh Statistics** tab displayed.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The table lists design elements and for each includes: Num Tets, Min edge length, Max edge length, RMS edge length, min tet vol., max tet vol., mean tet vol. and standard deviation (min elem area, max elem area, mean elem area and standard deviation for 2D designs).

If mesh repairs have been performed, two additional columns appear in the table: **Recovered %** and **Repaired %**. These columns indicate the fraction of an object that was successfully recovered, and the fraction that needed some repair.

To sort mesh statistics by column from low to high values or vice-versa, click a column header. This displays a shadowed triangle pointing down to indicate a list ordered from highest to lowest; and a triangle pointing up to indicate a list ordered from lowest to highest. Clicking again inverts the order.

Click on a blank cell above the object list to invert the order of objects. In this case, the cell does not display a directional triangle.

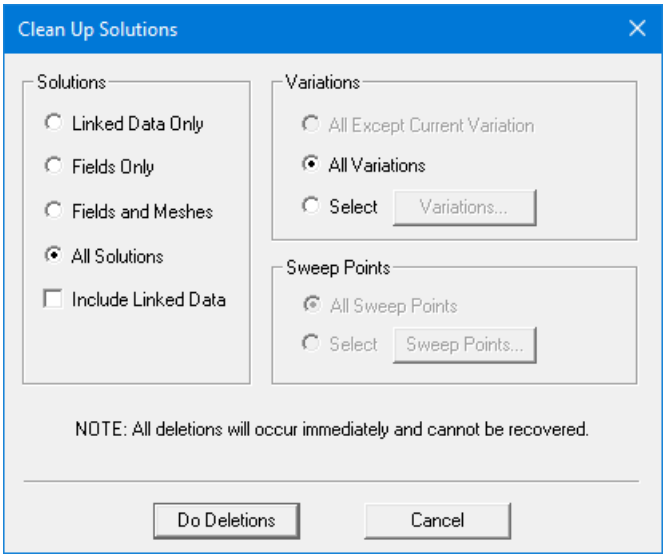
Deleting Solution Data

You can use **Clean Up Solutions** to selectively make deletions, or remove all solutions from the results.

To use **Clean Up Solutions**:

1. Click **Mechanical** > **Results** > **Clean Up Solutions**.

   The **Clean Up Solutions** dialog box appears.



2. Under **Solutions**, select whether you want to delete only fields data, only fields and mesh data, only linked data, or all solution data. Deleting all solution data erases all mesh, matrix, and fields data for all adaptive passes and frequency sweeps for the selected

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variations.

By option, you can **Include Linked Data** in the deletions.

Linked data can be mesh, field or some other post-processing data that the source design generated. The target design for the link caches these data internally to minimize the need to activate the source design. If you don't need to resolve the source design, for example, if it has a lengthy solution time, you should not check **Include Linked Data**. If you need to make some change in the source design that would affect the solution, and you want to ensure that the next solve result includes those changes, you should check Include **Linked Data**.

3. Under **Variations**, select which solution data you want to delete:
   - Select **All Except Current Variation** to delete all solution data that does not correspond to the current project and design variable values for the current design.
   - Select **All Variations** to delete all solution data for the current design.
   - Select **Select** to specify the variations you wish to delete. Click **Variations** to select the variations for deletion.
4. Click **Do Deletions**.

   The solution data you selected are deleted. Any post processing reports or field overlays you created that included data you deleted will be marked with an X in the project tree. They will be invalid until new solution data are generated.

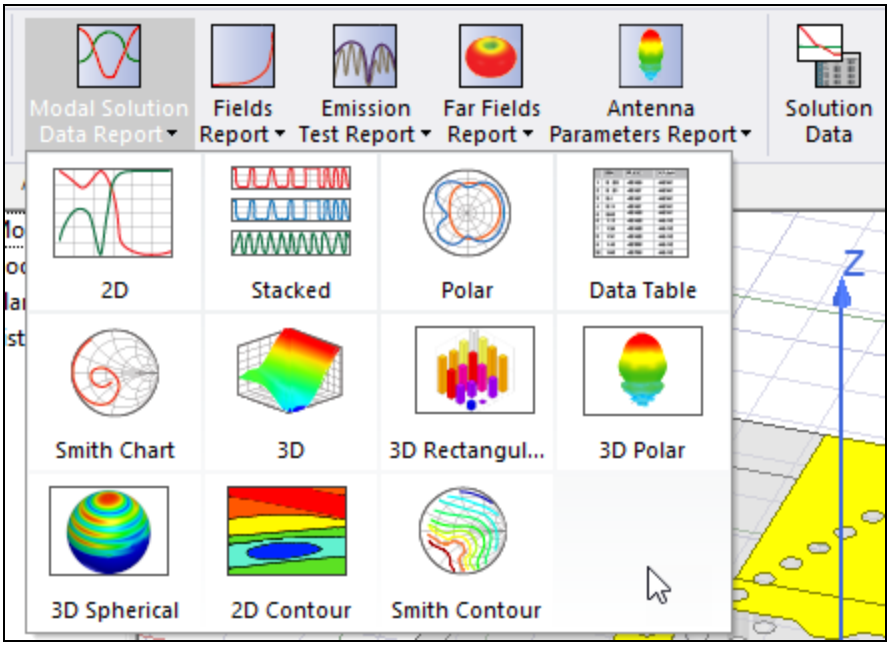
Creating Reports

After a solver generates a solution, you can analyze all the results for that solution. Ansys Electronics Desktop lets you create 2D or 3D plots. A 2D or 3D plot shows the relationship between a design's values and the corresponding results of the analysis. You can create reports using the **Create *<type>* Report** commands.

For each solution **<type>** (for instance, Modal, Terminal, Transient, Eigenmode, Characteristic mode, Fields, Far Fields, Emission test, Partial Discharge, Multipaction, and so forth), the **Results** menus present a list of **Create *<type>* Report** commands based on the solution data of direct interest for the design. For example, for the Eigenmode solution type, the **Results** menu contains templates for Eigenmode Parameters and for Fields. These appear on the menus as **Create Eigenmode Parameters Report** and **Create Fields Report**. For the Modal and Terminal Solution types, several different types appear, appropriate to each solution type. Each of these **Create *<type>* Report** menu items includes a further cascading menu that lists the Display Types available for that report. For some reports you can modify the Display Type from the Properties for that Report.

The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **Mechanical** > **Results** > **Report Templates** > <***templateName***>. You can also access previously defined 2D templates using **Report2D** > **Report Templates** > **Apply Settings...**. You can save the properties for a modified report to provide the custom default settings for all new reports.

You can also use the **Report2D** > **Export** or **Report 3D** >**Export** feature, select the *Ansoft ReportData Files (.rdat)* format, and **Save** the file, which you can later select to Create Report from File.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Much of the content within this branch of the help is general in nature and applicable to multiple design types (HFSS, Q3D, Icepak, and so on). For **Mechanical** designs, the types of reports that you can generate are dependent on the *Solution Type* (*Modal, Steady-State Thermal, Transient Thermal,* or *Structural*). Mechanical-specific reports content is covered within the *Modal Solutions*, *Structural Solutions*, and *Thermal Solutions* branches of the Mechanical help. Use the links below to access the Mechanical-specific reports content of interest:

Modal Solutions:

- Results (Modal)
- Modal Report
  - Creating a Modal Report
- Fields Report
  - Creating a Fields Report

Thermal Solutions: (Steady-State or Transient)

- Results (Thermal)
- Fields Report
  - Creating a Fields Report
- Monitor Report (*Transient Thermal* solutions only)
  - Creating a Monitor Report

Structural Solutions:

- Results (Structural)
- Fields Report
  - Creating a Fields Report

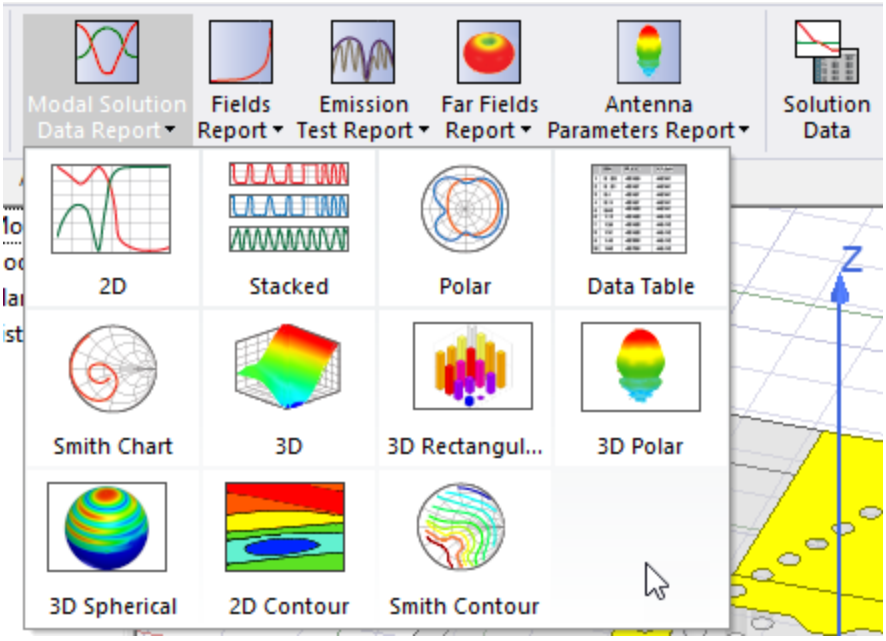
Creating a New Report

Following is the general procedure for creating a new report:

1. On the **Mechanical** menu or the Project Manager, point to **Results**, select **Create *<type>* Report**, and select the **Display Type** template.

   The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **Mechanical** > **Results** > **Report Templates** > **PersonalLib** > *<templateName>*. You can also make such changes the default for new reports by right-clicking a modified report and selecting **Report Templates** > **Save Settings as default**.

When you have selected the report and display type from the **Results** menu, the **Report** dialog box appears, with the **Trace** tab selected by default.

2. In the Context section, you make selections depending on the design and solution type.
3. In the **Y** Component section of the dialog box, make selections for the following:
    a. Categories – those depend on the Solution type and the design. The selected Category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button – opens the **Set Range Function** dialog box. This applies to the currently specified Quantity and Function.

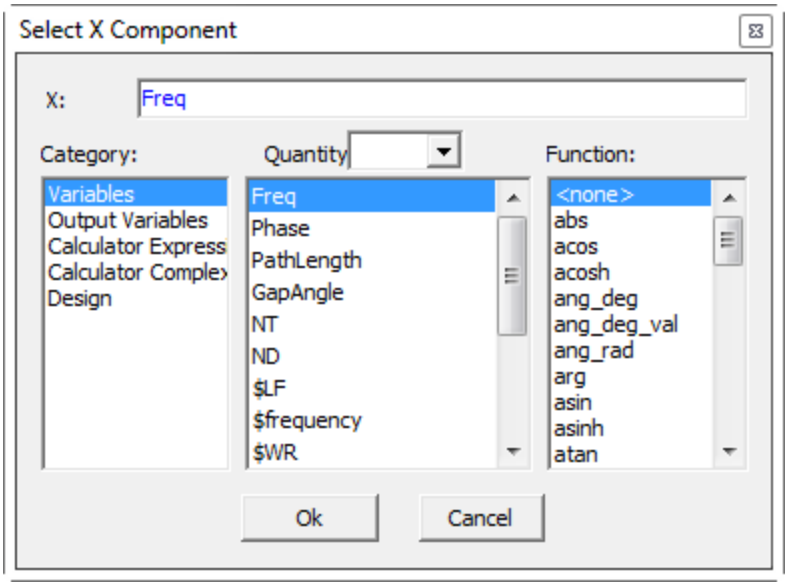
f. To use a dataset in this report, set the **Y** component to a Piecewise Linear Function `pwl(dataset_expression, primary_sweep)` where is the `primary_sweep` is what you set in the next step.

4. In the **X (Primary Sweep)** section, make selections for the following:

   a. Select the Primary value(s) from the drop-down menu.

      To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog box to assign the X component.

b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the **Edit Sweep** dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

5. **Update Report** setting
   - **Real Time** checked -- enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked -- enables drop down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
6. The **Report** dialog command buttons permit you create a new report with the settings you provide, or to modify an existing report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Output Variables** – opens the **Output Variables** dialog box.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Update Trace** – updates the selected traces in a report based on further processing or changes.
- **New Report**. Adds a report to the Project tree under the Results icon. The new Report is displayed in the Project window.
- **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **Close** – closes the **Report** dialog box.

7. Click **New Report** to create a new report in the Project Manager.

   The report appears in the view window. It will be listed in the project tree under Results, with the default name based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes. Traces within the report also appear in the project tree.

   Some plots may take time to complete. Performing a **File>Save** in such cases after the plot has been created will permit you to review the plot later without having to repeat the calculation time when you reopen the project later.

8. To speed redraw times for changed plots, perform a **Save**. This saves the data that comprises expressions. For example if re(S_{11})*re(S_{22}) is requested over multiple widths, each of the S_{11} and S_{22} are stored when you save. If you do not do a save of a changed plot, the changed version is not stored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Remember that the evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. If you want to plot the polar angle of a complex simulation result, S11 say, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity but in degree and radian units respectively.

Note that when used in expressions, some surprising outcomes might result. For example, the expression "1+ang_deg(S11)" represents an ‘angle’ and the number "1" is treated as "1 rad". The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val (S_{11}) would be 90 and "1 + ang_deg_val(S_{11})" will be 91.

Context Section for Reports

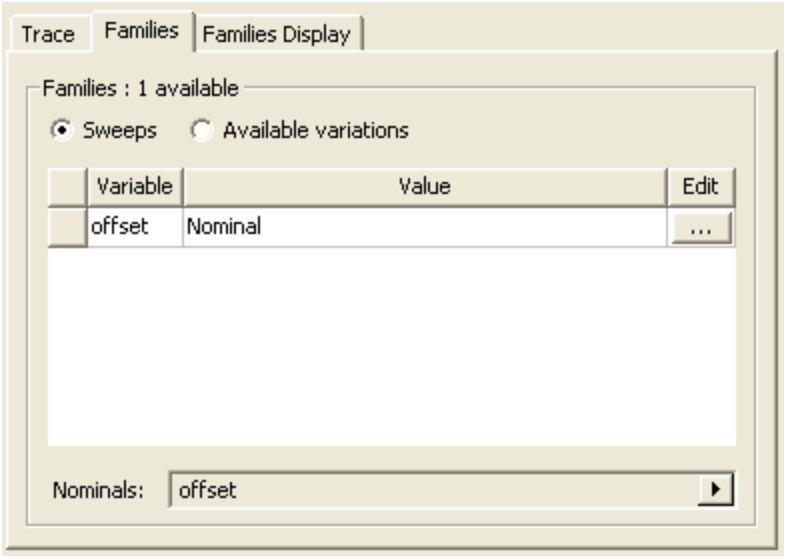
In the **Context** section, make selections from the following field or fields, depending on the design and solution type.

1. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
2. Geometry field with a drop-down selection list.

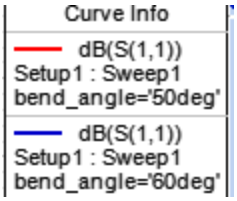
Using Families Tab for Reports

The **Families** tab of the **Report** dialog box provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined and solutions exist for multiple variable values (for example, for a parametric sweep or re-running an analysis with a different variable value). If no variables are defined, or none have solutions for different values, 0 families will be available. If so, the variables other than the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the solution value (which may be All, Nominal, or a Specific value), and an Edit column with an ellipsis [**...**] button. Families gives the number available. If an existing variable is specified as Nominal, only that value is currently available. You can set any solved variables as Nominal, All, or select from values provided for Available solutions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



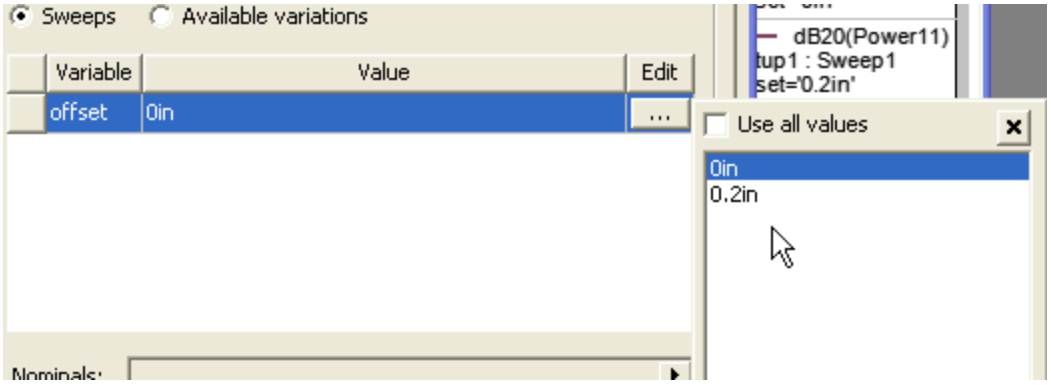
When you select a variable with multiple solved values, a trace for each solved value appears in the Report, with the variable value appended to the trace name in the Report legend.



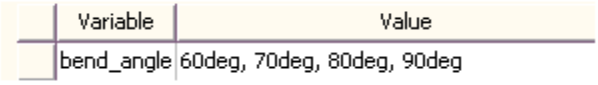
When families are available, you can make selections for the following:

1. Select the **Sweeps** radio button (the default) to list the swept variables you can select or the Available variations button to list and select variation values for which solutions exist.
2. With the **Sweeps** radio button selected, click the ellipsis [**...**] button to display a list of variable values for a particular variable. If many variables exist, you can use a scroll bar to navigate the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

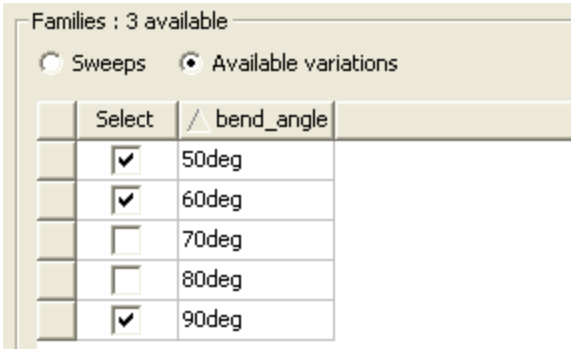


- To select all values, click the check box for **Use all** values. This writes "All" in the value field for that variable. You can also select individual values by clicking on them.
- To select a range of values, hold down the shift key, and click again.
- To select intermittent additional values, hold the Ctrl key and click additional. The values you select are highlighted in the list, and are also listed in the Values column for that variable.
- To select all, use the **Select All** button. This highlights the complete list, as well as listing all values for the variable in the Value field.



- To clear the selections, use the **Clear All** button.

Select the **Available variations** radio button to list the choices that derive from variable combination.

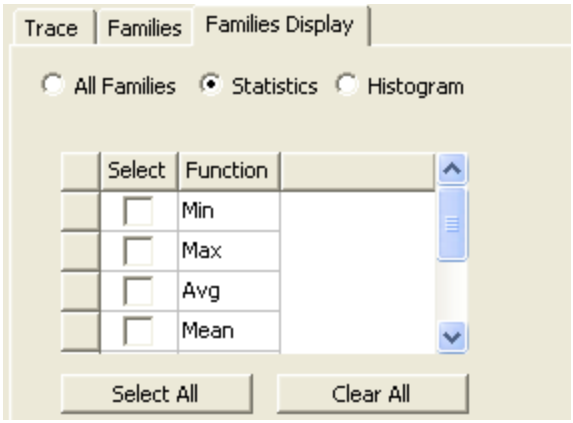


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

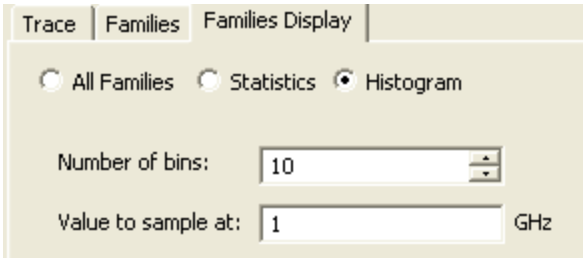
a. To select individual variations, check the select box.
b. To check or clear all variations at once, click the **Select** button at the top of the column.
c. To invert the list order, click the triangle beside the variable name.

The **Families Display** tab has three radio button selections.

a. **All Families**
b. **Statistics** lists a table statistical functions that you can select to apply to the plot. The functions include Min, Max, Avg, Mean, Variance, Std Dev, and Sum. You can use the Select check boxes or the Select All and Clear All button.



c. **Histogram** lets you select the number of bins to use for a histogram plot, and the sampling frequency to use.



Modifying Reports

To modify the data that is plotted in a report:

1. In the Project Manager, click the report you want to modify.
2. Right-click **Modify Report**.

   The **Report** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The **Report** dialog box command buttons permit you create a new report with the settings you provide, or to modify an existing report.
   - **Output Variables** – opens the **Output Variables** dialog box.
   - **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
   - **Update Trace** – updates the selected traces in a report based on further processing or changes.
   - **New Report -** adds a report to the Project tree under the Results icon. The new Report is displayed in the main window.
   - **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
   - **Close** – closes the **Report** dialog box.

   The updated report appears in the view window.
4. **Update Report** setting
   - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked – enables drop-down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
5. In the **Context** section you make selections depending on the design and solution type.
6. The Families tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.
7. In the **Y** Component section of the dialog make selections for the following:
   a. Categories – those depend on the Solution type and the design. The selected category provides the default name of the plot, for instance S Parameter Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

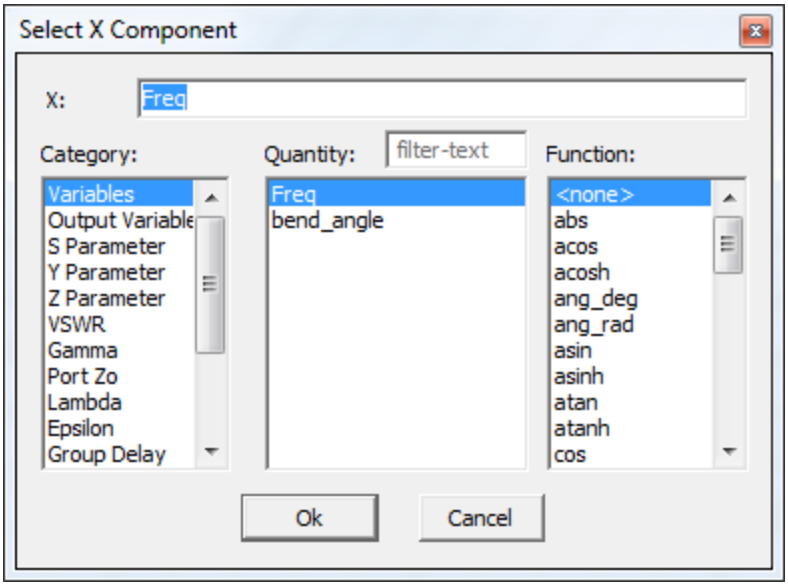
e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

8. In the **X (Primary Sweep)** section, make selections for the following:

a. Select the Primary value(s) from the drop-down menu.

To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

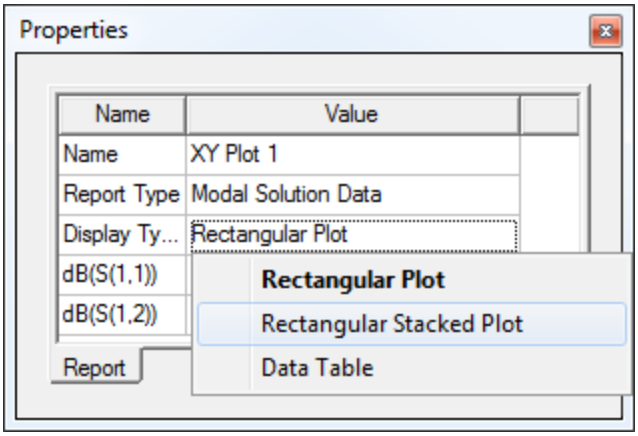
b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the *Edit Sweep* dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

You can also view and edit the properties of Reports and their traces via their Properties windows. See Modifying the Background Properties of a Report.

You can also modify the display type of an existing plot from the Properties dialog box for that plot. Select the Report icon in the Project Manager to display the **Properties** dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



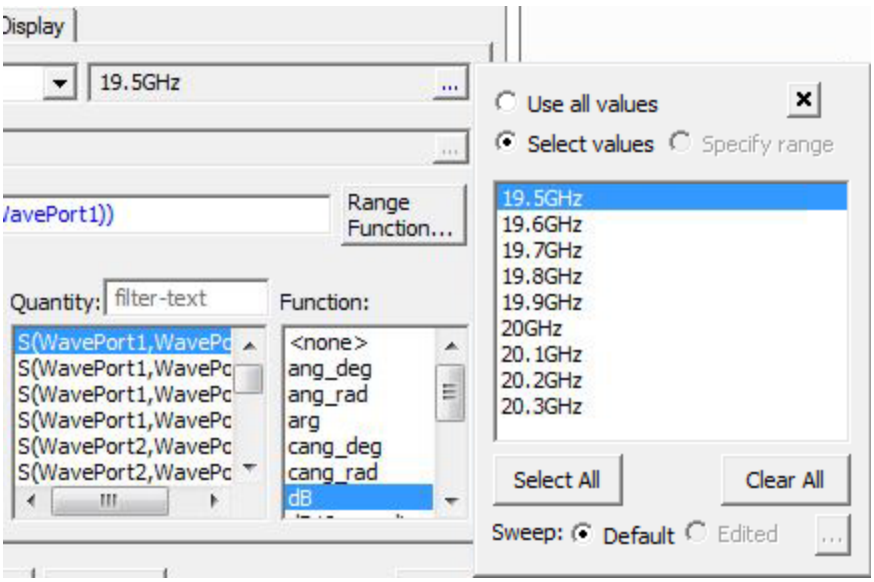
Once you make a selection, the plot display updates for the current selection.

Note:

Remember that for many excitations of interest for plotting, you can control the default base names through the dialog box described here: Setting Default Boundary/Excitation Base Names. This may save you the need to edit individual names in the plots.

Modify Report: Selecting Use All Values or Making Selection

Clicking the browse button on **Primary Sweep** line shows the default selection of **Use all values**. Choose **Select values** to display the sweeps and enable editing, including the **Select All** and **Clear All** buttons.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With **Select values** selected, you can select one or more by clicking an individual value, dragging to select multiple values, or using Alt+click to select specific values.

You select either the Sweep radio button for Default or Edited selection.

You can also select **Browse** [...] here to display the Edit Sweep dialog for Modify Reports, which includes additional editing features.

Modify Report: Using the Edit Sweep Dialog Box

Clicking **Browse** [...] at the lower right corner of the **Use all values** pane opens the **Edit Sweep** dialog box for Modify Report. This lets you edit the current Primary sweep variable values, including radio button selections for Single value, Linear step or count, and Decade, Octave, or Exponential Counts.

You can specify start, stop and step values and units, and add specific values to the list of current sweep values. The **Add>>** and **Update>>** buttons let you edit the value list.

You can use the mouse click, drag, and Alt+click to select values. You can also edit individual values.

Creating a Report from an Ansoft Report Data File

If you have previously saved an Ansoft Report Data Format (using Report2D > Export), you can create a report from that rdat file. This provides a way to reuse the data and/or the format of a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

previously created report.

1. Right-click the Results icon in the Project Manager to display the short cut menu and select **Create Report From File**, or click **Mechanical** > **Results** > **Create Report From File**.

   A file browser displays.

2. Select an .rdat format file, and click **Open**.

   The report is created. If it contains data, it displays the report with traces. If not, the report uses the format exported to the .rdat file.

Zooming and Fitting Reports

The standard Zoom and Fit commands operate on reports. After clicking in an open report, you can also use a mouse wheel to zoom in and out.

Modifying the Background Properties of a Report

To modify the appearance of a report, or to modify the display properties any object in a report, including traces, axis labels, grids, colors, fonts, legends, color maps, contour color, and so forth:

1. Open the report you want to modify.
2. Select an editable object in the report to be able to edit its properties. Click on an object to select it and to view its properties in the docked **Properties** window. To open a floating **Properties** window, either double-click the selected object, or select **Edit** > **Properties** from the menu bar.

   The **Properties** tabs and options displayed for editable plot objects varies depending on the report type (for example, whether 2D rectangular, 2D polar, Smith, Stacked, or 3D), but can include the following:

   - **Cartesian** – controls the scroll bar and thumb properties for 2D rectangular plots.
   - **Header** – controls the properties for the text displayed at the top of the report, including the Title font, Company Name, Show Design Name, Subtitle Font. The plot title is tied to the report's name and is not a Header property. If you change the report name in the Project tree, plot title synchronizes. The Company Name and the Show Design Name check box are grouped in the **Properties** dialog box as Subtitle. Edits to the Subtitle Font Property affects both of them.
   - **General** – controls the background color (the perimeter around the trace display) for the plot, the contrast color (the trace display background), the Field width, the Precision, and whether to use scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.) An

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Scale Fonts property is on by default and scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.

- **Legend** – controls the properties for whether to include a Legend Name, Show Trace Name, Solution Name, and Variation Key. At least one of these must be selected. Legend Name is blank by default. When non-empty, a header row for the Legend in plot shows up with that string. You can also specify the File Name Display as Full Path, File Name without Path, or as an Array Index. You can also edit the Font, the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width. See: Modifying the Legend in a Report.
- **Color Key** – for 3D plots, controls the appearance of the color key (colors, transparency, border appearance, fonts, number format, field width and precision.
- **Contour** – for 3D plots, controls the appearance of the color map, including map type, ramp color, spectrum, IsoValType, levels, number of contours, and values shown.
- **Radiation Pattern** – for 2D polar plots, controls whether to show the circular grid and angle lines.
- **Stacked** – for stacked plots, controls properties for X scrollbar, thumb properties, and stack layout, auto fit, and stack height.
- **Smith** – for Smith charts, controls whether to show grids for Imp., Adm., Cir, and angle lines.
- **Traces** – controls the properties for traces, including: Color, Line Style, Line Width, Trace Type, whether to Show a symbol, Symbol Frequency, Symbol style, whether to Fill symbol, symbol color, and whether to show arrows. You can select traces either in the Legend or on the plot. See: Editing the Display Properties of Traces.
- **Axis** for X, Y or Z, or for Phi, Theta or Rho, and circular – the defaults for most of these values (applying to 2D and 3D both) are set in the **Report 2D Options**Axis tab.
    - Display name – check box for whether to display the axis name.
    - Specify name – check box for specifying the Axis name.
    - Name – this describes the axis to which the following properties/options refer. These are selected in the **Report** dialog box.
    - Axis Color – set the color by double-clicking to display the **Set color** dialog box. Select a default or custom color and click OK.
    - Axis Font – click the cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog box also contains a preview field. OK the selections to apply the font edits and to close the dialog box.
    - Display Units – this specifies whether to display the axis units.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Window (section):**
  - Window Mode – can be Axis range, Continuous moving window, or Step moving window.
  - Window Width (in) – provide an integer value for the previous selection.
- **Manual Format** (section):
  - Number format – select from the drop down menu, Auto, Decimal, or Scientific notation.
  - Field Width – enter a real value.
  - Field Precision – enter a real value.

- X or Y or Z **Scaling Tab** – These properties provide control over scaling:
  - Axis Scaling – use the drop-down menu to select scaling as Linear or Log. For the Y axis, all zero or negative values are discarded before log scaling is applied. For 3D plots, scaling is on the Axis tabs
  - Specify Min – check box
  - Min – text entry in same units as axis units. Saved as SI internally.
  - Specify Max – check box
  - Max – text entry in same units as axis units. Saved as SI internally.
  - Specify Spacing – check box
  - Spacing – text entry in same units as axis units. Saved as SI internally
  - **Manual Units** (section):
    - Auto Units – use the check box compute the correct units for the axis.
    - Units – click on the cell to select from a menu of available units if you have not checked Auto Units.
  - **Infinity Visualization** (section):
    - Map Infinity Mode – check box.

      Each axis can be set to treat infinity values in a user defined way. When you check the Map Infinity Mode, any infinity values in the input data get the infinityMap value (negative infinity get the value*-1 and positive infinity the positive value specified). This can be useful if there are zeros, or very small values that Mechanicaltreats as zero, in the data, for example, dB Gain.

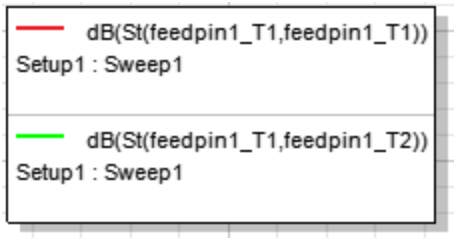
    - Map Infinity To – enter a real value for the Map Infinity Mode.
- **Grid** – properties for grid labels and grid style, appearance, line styles, color, major and minor lines, major and minor circles on polar grids, and scaling. For the 3D rectangular plots, there are separate tabs for the XY, YZ and ZX axes, and for 3D Polar plots, tabs for phi-rho, or theta-rho grids.

3. Edit the properties, and **OK** the dialog box to apply the changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying the Legend in a Report

The legend in a report is a list of the curves being plotted. For each curve, the legend gives the name, shows the line color, and lists the setup and the adaptive pass used to generate the curve.



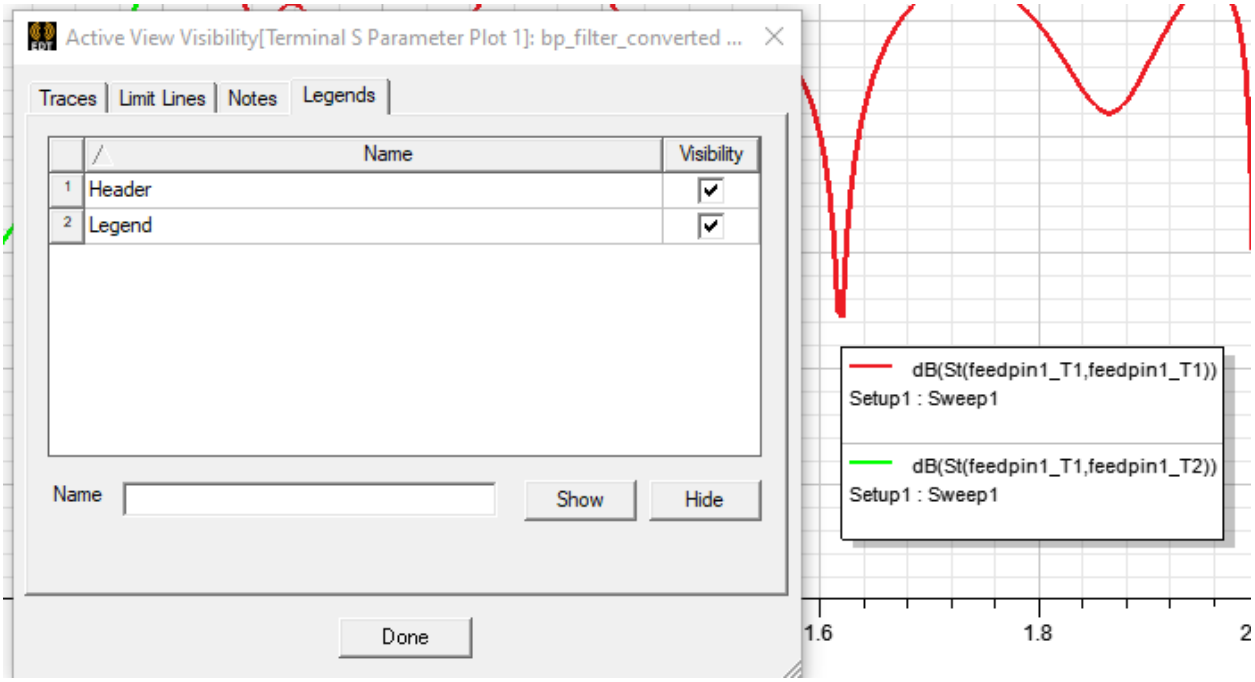
To show or hide a legend in a report:

1. Make the report the active view.
2. Use **View** > **Active View Visibility** or the **Show/Hide** icons on the toolbar to display or hide the report.

   Either command displays the **Active View** dialog box.

3. Select the **Legend** tab.

   This lists the legend (or legends) in the report.



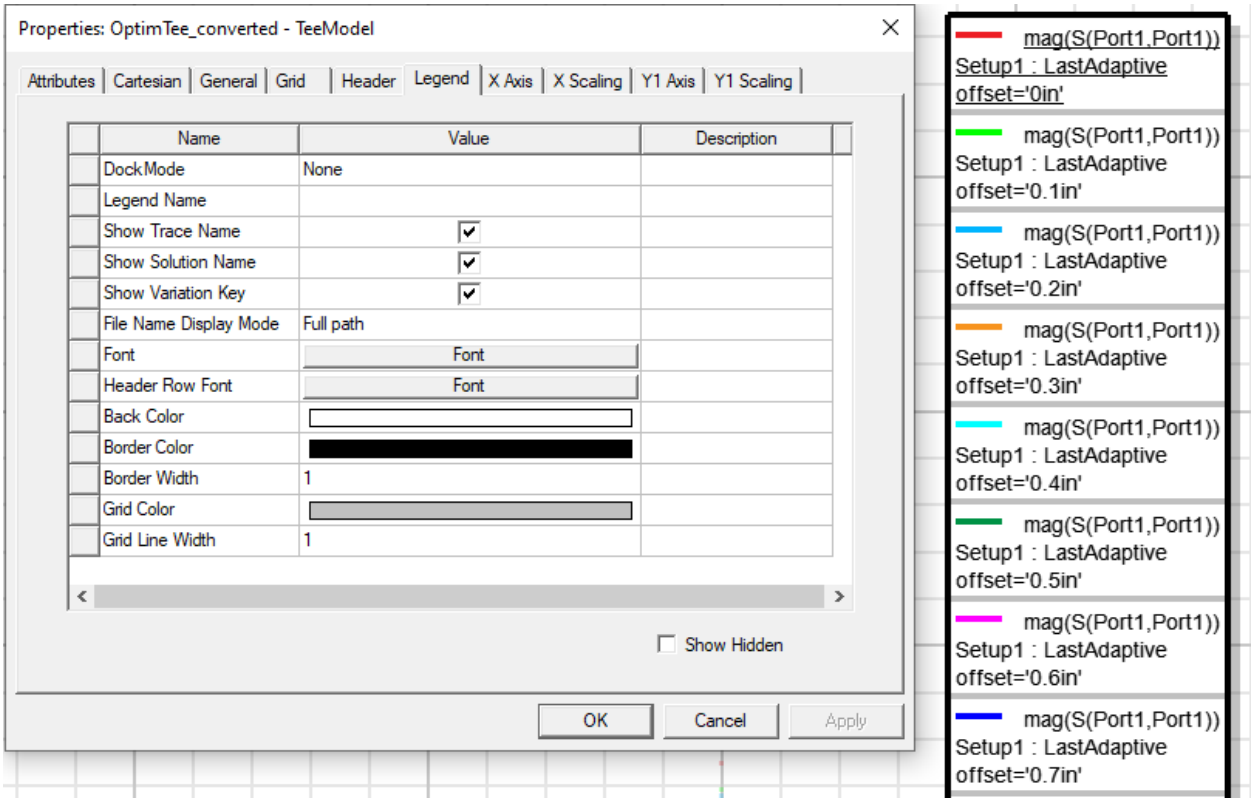
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

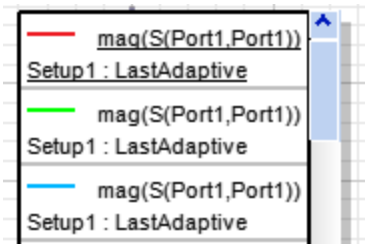
To edit the display properties of a legend:

1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   This lets you edit the Properties for Dock Mode, Legend Name (default is no name. When non-empty, a header row for the Legend in plot shows up with that string.), whether to Show Trace Name, Solution Name, and Variation Key (which applies to parametric variables, if present). If none of these three are selected, only a trace color shows.



   Here is an example with Variation Key off.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also edit the Font by clicking the Font cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog also contains a preview field. OK the selections to apply the font edits and to close the dialog box.

You can also edit the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width.

2. Click **OK** to close the **Properties** dialog box and apply the selections.

To change the display name for traces, see: Editing Trace Properties.

To move a legend in a report:

1. Click and hold on the legend.

   The cursor changes to crossed lines with arrow tips.

2. Still holding, drag the legend to a new location and release.

   The legend is released and the crossed lines change back to a mouse pointer.

To resize a legend in a report:

1. Position the mouse tip over the edge you want to resize.

   The mouse pointer changes to a horizontal or vertical line with arrow tips.

2. Click and drag the horizontal or vertical edge to the desire size.
3. Release.

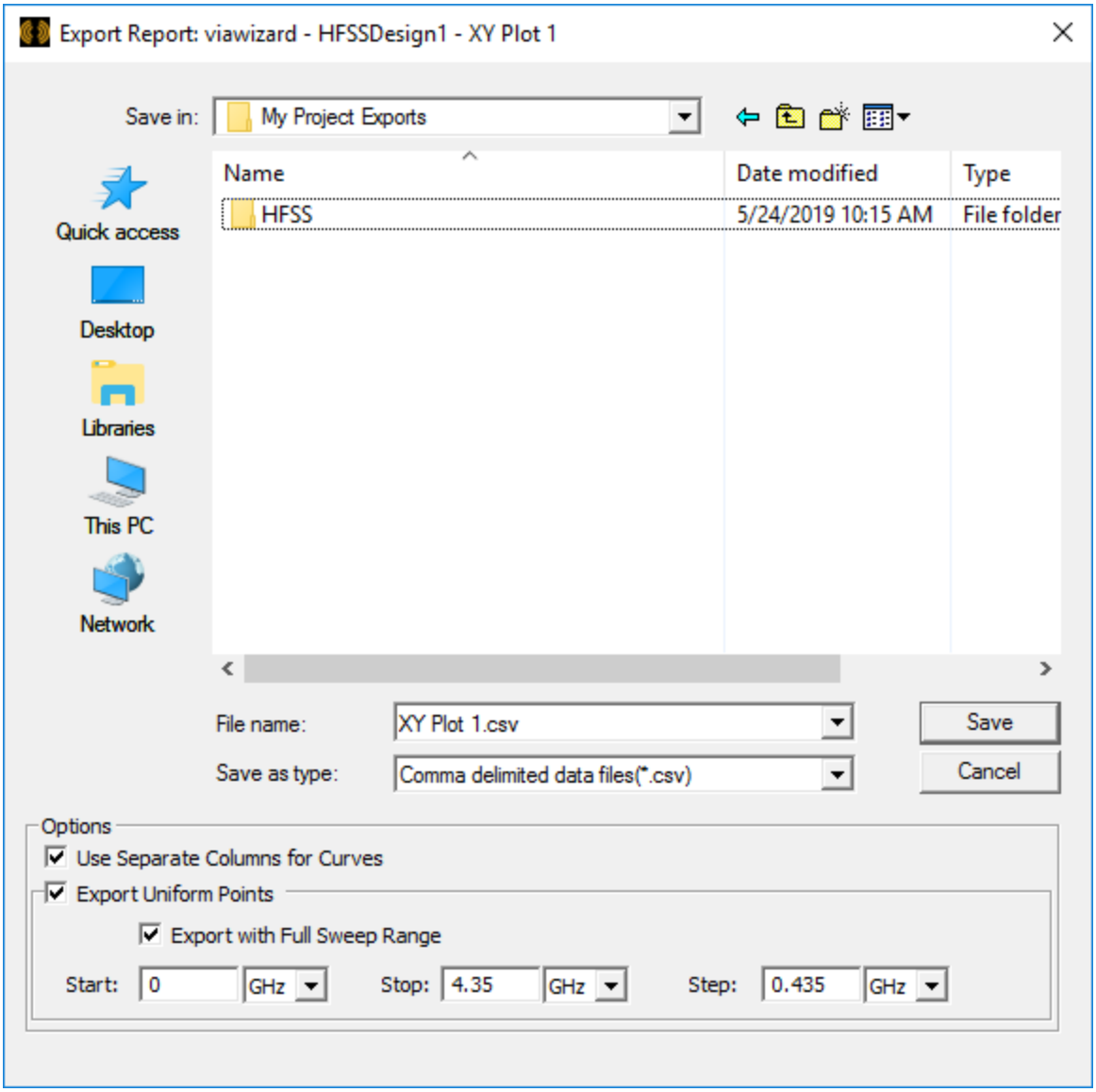
Exporting Ansoft Report Data Format Files

Ansoft Report Data format (*.rdat) files provide a way to export reports or report formats, which you can then import using **Reports> Create Report From File**. This can save repeated editing of properties (for example, the company name or color schemes) when you create other reports. You must have an existing plot open to see the **Report2D** menu.

1. Click **Report2D** > **Export...** or **Report 3D**> **Export...**

   The **Export Report** dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the **Save as type** drop-down menu to select **Ansoft Report Data Files (*.rdat)** format.
3. Browse to a destination folder and enter a name for your file in the **File name** field.
4. The **Export Uniform Points** option, if available for your report type, allows the *.rdat file to contain points for the start, stop, and step at the given frequencies. If you do not select this option, the file contains only the current file format, including any modifications you have applied.
5. Click **Save**.

   The file is exported to the specified location as an Ansoft Report Data file (*.rdat).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

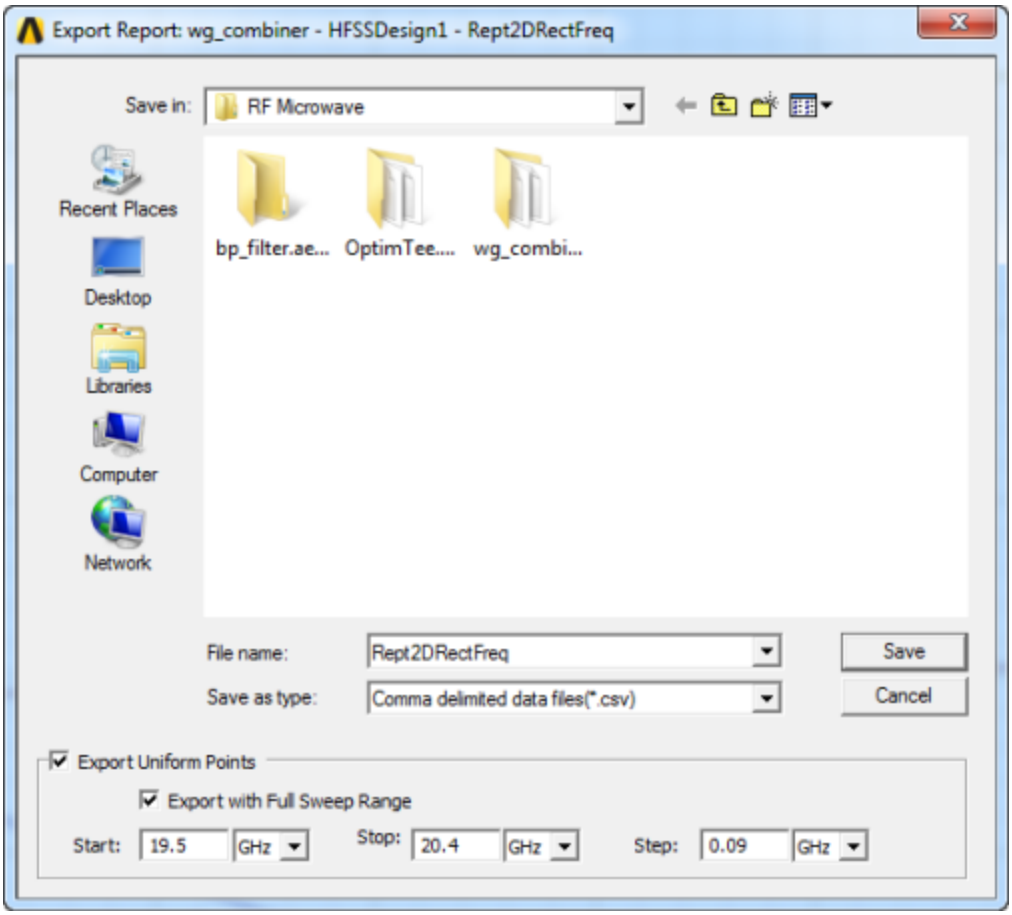
The file will then be available for import using Create Report from file.

Exporting Reports as Graphics

You can export reports as figures in several formats. You must have an existing plot open to see the **Report2D** or **Report 3D** menu.

1. With a report open, click **Report2D** > **Export...** or **Report3D** > **Export...**

   The **Export Report** dialog box appears:



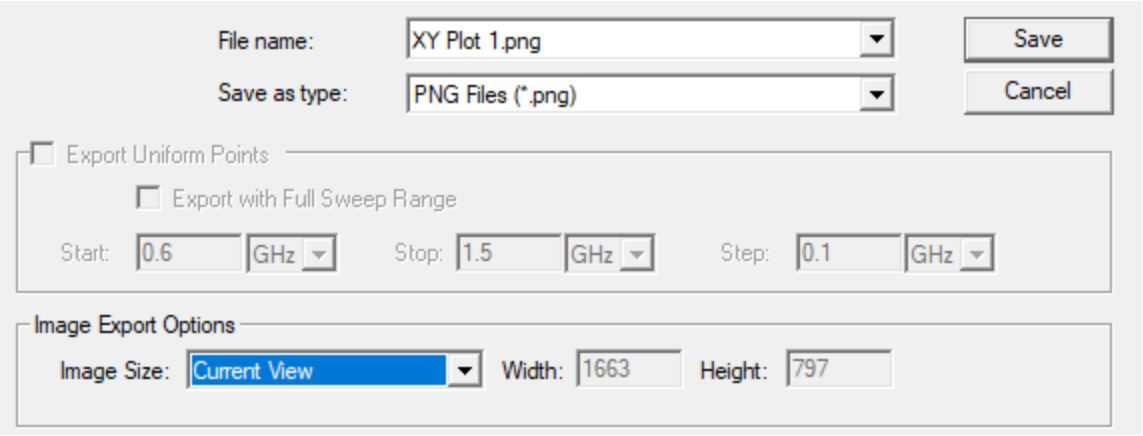
2. Click **Browse...** to open the **Export Report** browser window.
3. Specify the file location and name, and select a graphics format from the drop-down menu:

   - .bmp
   - .gif
   - .jpeg

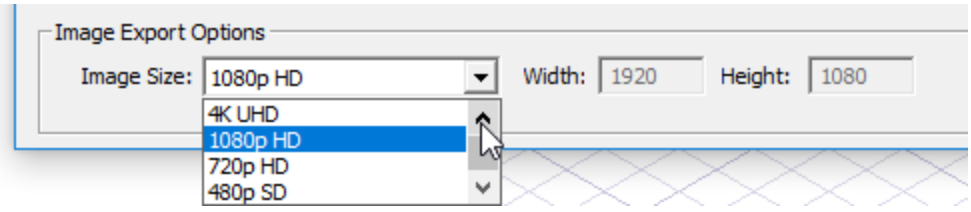
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- .png
- .tiff
- .wrl

4. When you select an image format, the **Image Export Options** appear at the bottom of the dialog box:



5. Specify the **Image Export Options** by selecting from the drop-down menu. You may need to scroll or click the up or down arrows to see all selections. These include: Current View, User Defined (which enables the Width and Height fields), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD. The resolution for the current selection is displayed in the Width and Height fields.



6. Click **Save** to close the browser window, and then **OK** to close the **Export** window.

To export an image file of a report to specified high resolution, use the scripting commands. The image can be created at a specified resolution. Fonts and line thickness is not scaled. Only the image. You will have to iteratively increase font sizes until you find a suitable output.

Creating Custom Report Templates and Defaults

You can edit properties from any report type and save it as a template or as the default. This can save repeated editing of properties (for example, the company name, or color schemes and plot attribute settings) when you create other reports. You can prepare a template by copy/pasting

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plots settings from one plot to another of the same display type. Once you create templates, you can access them from the **Results** > **Report Templates** > menu and the **Report2D** > **Report Templates** > **Apply Settings** menu.

See Modifying the Background Properties of a Report for a discussion of format changes you can make to any report.

To save an edited report as a template:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save...**. You can also click **Report2D** > **Report templates** > **Save...** or **Report3D** > **Save As Template...**

   This displays the **Save As Report Template** file browser. By default, the directory is your AnsysEM\<*productName*>\userlib\ReportTemplates directory. You can also save to the SysLib directory.

2. Typically, you accept the directory.
3. You must provide a file name, which will be given an *.rpt extension.

   It is good practice to give the template a descriptive name, showing both the kind of format you begin with (such as XY Plot or 3D Plot) and apt description of the distinguishing edits (such as for company name, or color scheme). Once, saved, this name will appear on the *PersonalLib* menu.

   The **Save As Type** field currently supports the Ansoft Report Format (*rpt) format.

4. Click **Save** to save the template to the PersonalLib menu.

All *.rpt templates in the userLib directory appear on the **Results> Report Templates> PersonalLib** menu. Selecting a report from the PersonalLib menu opens a report that you can then Modify to add traces or perform other edits. Templates in the SysLib directory appear on the **Report Templates** menu.

To save an edited report as a default:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save Settings as default**. You can also click **Report2D** > **Report templates** > **Save Settings as default** or **Report3D** > **Save Settings as default**.

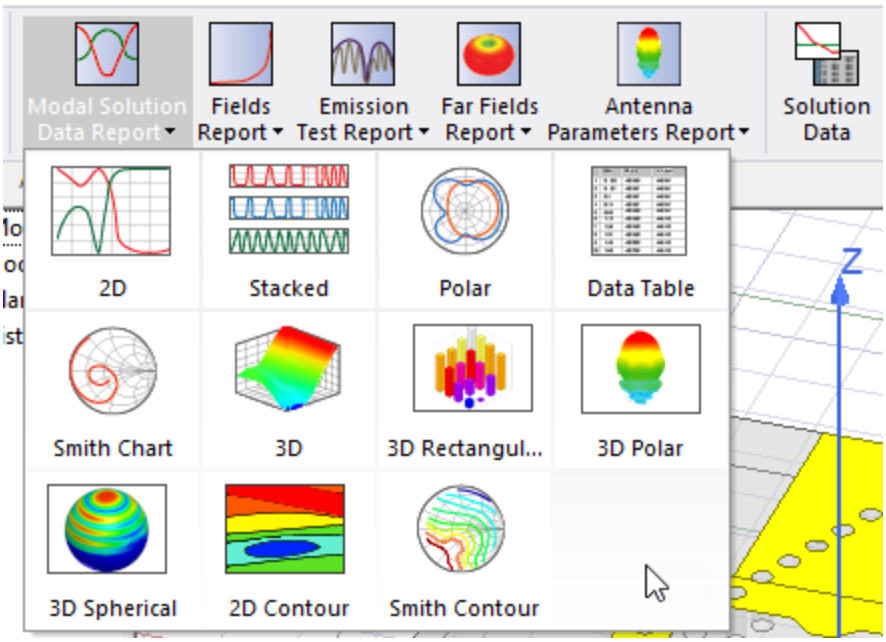
Selecting the Report Type

The **Report Types** available for creating a report depend on the simulation setup. The report category provides the default report name for the project tree and the text displayed in the report header. For example, S Parameters as the Category type causes the default report name to be S Parameter Plot *n*. Depending on the setup, you can make a selection from the following report types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modal Report	Results related to natural frequency of vibration (Modal) analyses, including Frequency, Mass Participation Factor, Effective Mass Ratio, and so on.
Modal Fields **Thermal Fields**	Basic or derived field quantities calculated on lines or integrated over surfaces or objects are available to plot. Displacement magnitude is available as a predefined modal field result. Predefined thermal field results include temperature, heat flux magnitude, and EM loss densities.

When you click **Product > Results** or right-click **Results** in the Project tree, or select the **Results** tab of the ribbon, one or more of the following report types and displays are available. *Available options depend on the nature of your installation and project(s)*.



Report Type	Display Type

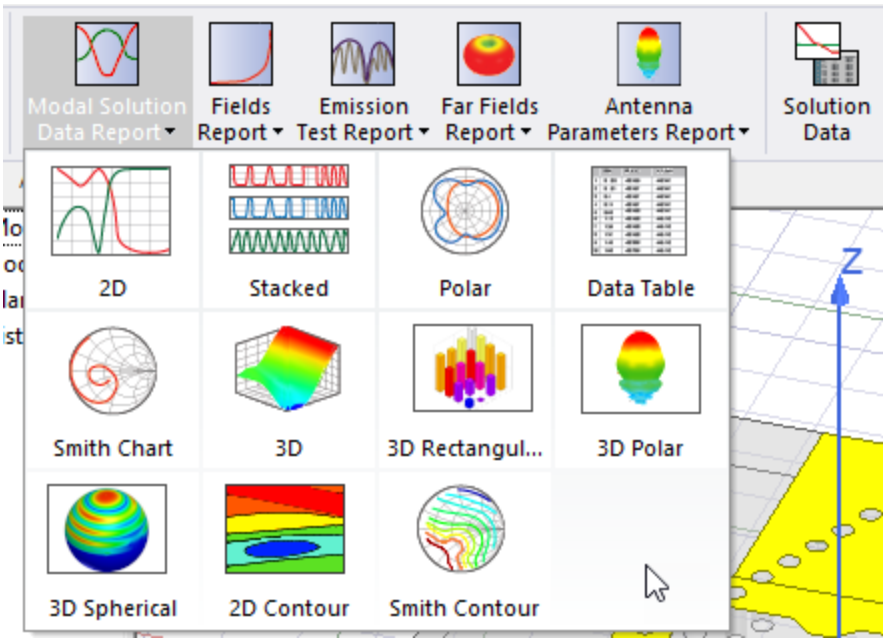
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Standard Report	A Standard report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Polar Plot • Smith Chart • Rectangular Stacked Plot • 3D Rectangular Plot • 3D Rectangular Bar Plot • 3D Polar Plot • 3D Spherical Plot • Rectangular Contour Plot • Smith Contour Plot
Create Eye Diagram Report	An Eye Diagram report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Statistical Eye Plot
Create Constellation Report	A Constellation report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Statistical Eye Report	A Statistical Eye report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Report From File	A Data File report can be created in any of the available display types using a previous report that has been saved to a file.

Selecting the Display Type

The information in a report can be displayed in several formats. When you select the **Results** tab in the ribbon, depending on reports available for a given design and Solution type, you see a selection of report type groups (for example, Modal Solution Data, or Fields Report, Emission Test, Antenna Parameters, Near Fields, Terminal Solution Report, Far Fields Report, Characteristic Mode Data, etc.), with drop-down menus that displays the kinds of reports available for the design in that group:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

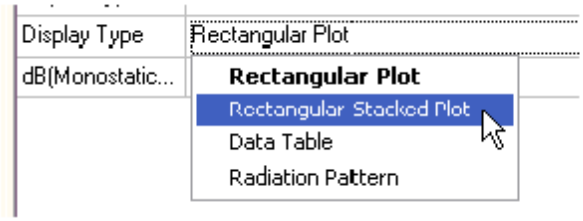


For the initial plot, you can also select from the following **Display Type** formats in the **Create<*type*> Report** submenu:

Rectangular Plot	A 2D rectangular (x-y) graph.
Rectangular Stacked Plot	This choice puts each trace into its own 2D rectangular plot, and stacks each plot, rather than overlaying the traces on the same plot.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
3D Rectangular Bar Plot	A 3D plot of rectangular bars.
Rectangular Contour Plot	A rectangular (x-y-z) graph. Contour plots are useful to visualize surfaces (e.g., Directivity as a function of phi/theta).
Data Table	A grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.

You can also modify the display type of an existing plot from the **Properties** dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the **Display Type** field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Plots

A rectangular plot is a 2D, x-y graph of results.

1. On the **Results** menu (under the **Mechanical** menu, or right-click **Results** in the Project Manager), click **Create < *type* > Report**, and select **Rectangular Plot**.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed, depend on the Solution type and the report <*type*> selected.
   c. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot. The category selected provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

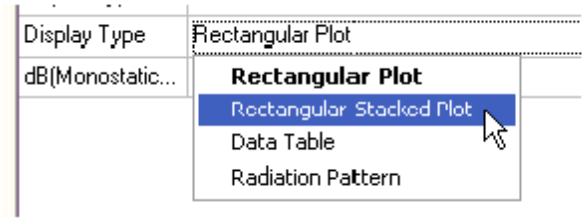
4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular sweep or sweeps, or all sweeps. The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plott *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

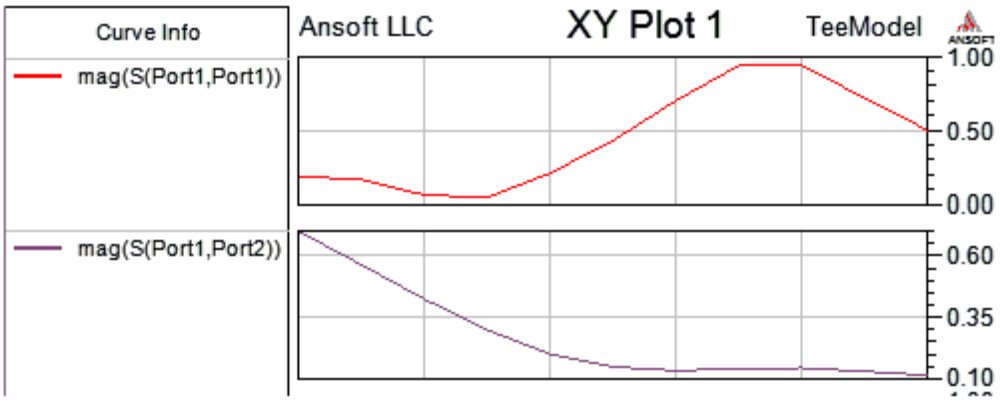


Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Stacked Plots

A rectangular stacked plot is a 2D, x-y graph of results, with each trace displayed on a separate plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. On the **Results** menu (under the **Mechanical** menu or right-click **Results** in the Project Manager), click **Create <*type*> Report**, and select **Rectangular Stacked Plot**.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop-down selection list. Whether this field appears, and the domains listed, depend on the Solution type and the report ***<type>*** selected.
   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
      b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

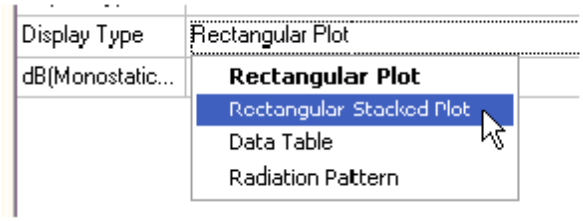
4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular sweep or sweeps, or all sweeps. The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
6. Click **New Report.**

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

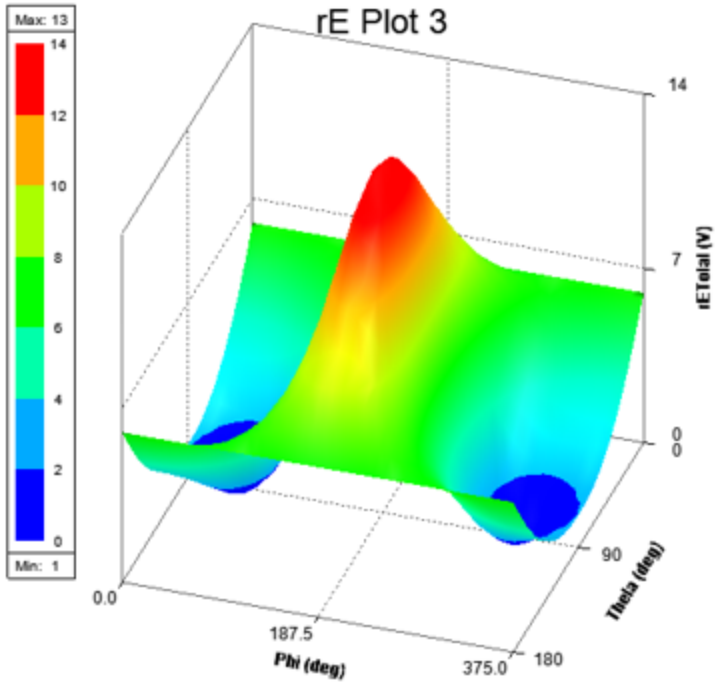


Once you make a selection, the plot display updates for the current selection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 3D Rectangular Plots

This is a 3D, x-y-z graph of results.



Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the Properties dialog box. The properties are grouped appropriately under various tabs, which correspond to plot entities:

- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title.
- Axis [X|Y|Z]: Properties related to the 3 axes
- Grid [XY|YZ|ZX]: Properties related to the 3 grids
- ColorKey: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Plot" on the next page.

Creating a 3D Rectangular Plot

1. On the **Results** menu (under the **Mechanical** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **3D Rectangular plot** from the report type menu.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab **Z** Component area, specify the information to plot along the z-axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the sweep variable to use from the drop-down list.
- If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

6. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
7. Click **New Report.**

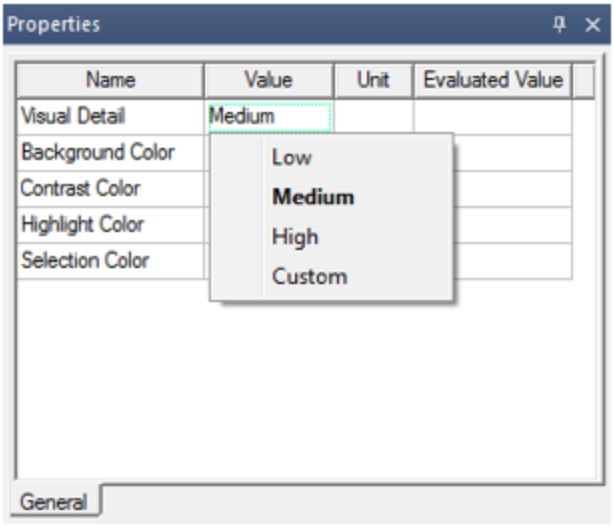
   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Controlling Visual Detail in a 3D Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

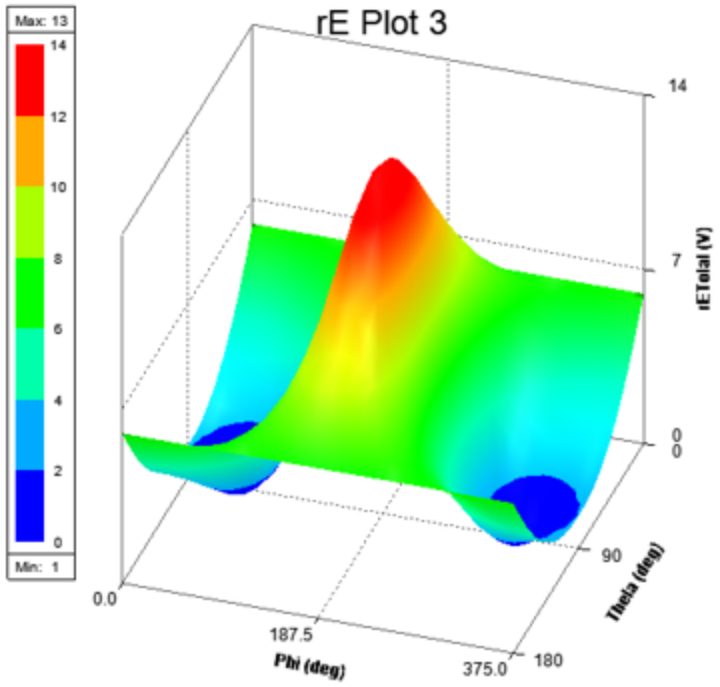


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

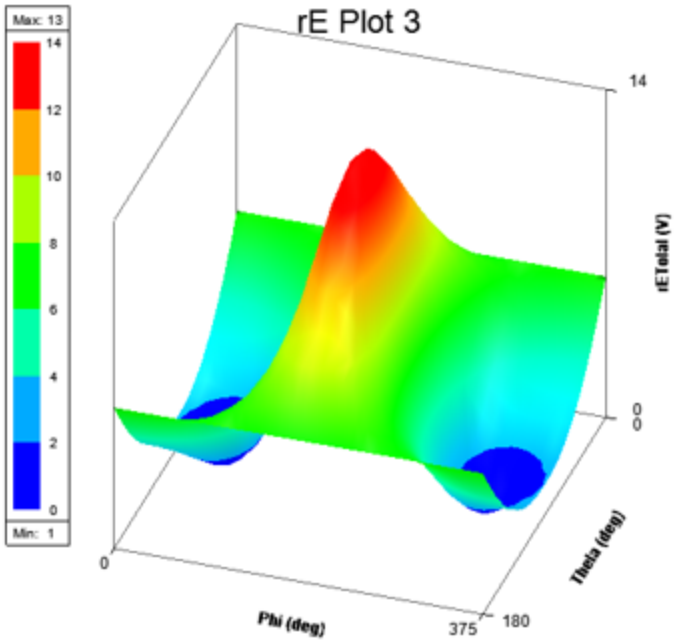
3D Rectangular Plot with Medium Visual Detail



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

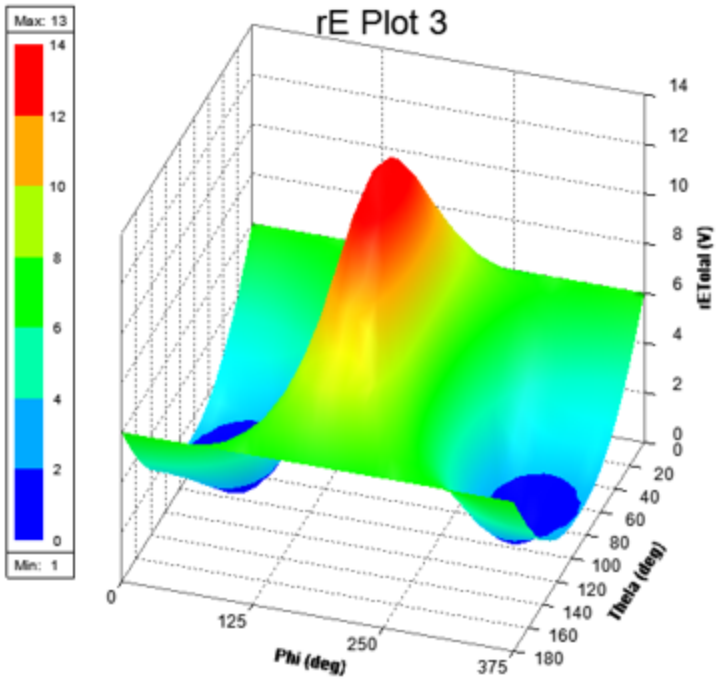
3D Rectangular Plot with Low Visual Detail

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Plot with High Visual Detail

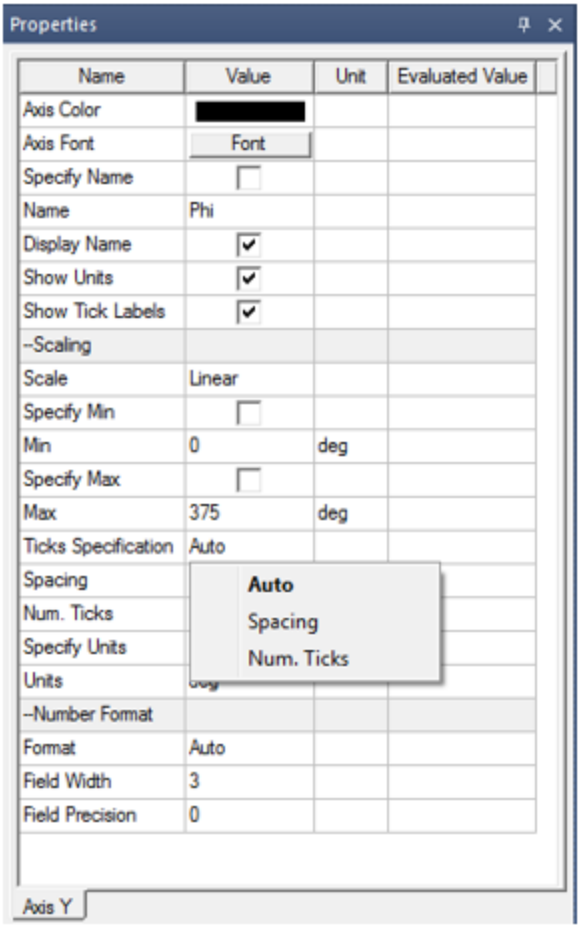


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--Scaling			
Scale	Linear		
Specify Min	☐		
Min	0	deg	
Specify Max	☐		
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Checked, Ticks Specification is set to Spacing.

Creating Rectangular Contour Plots

This is an x-y-z graph of results. Any data that you can currently plot in 3D (as 3D Cartesian or 3D polar) is a candidate for a contour plot.

1. On the **Results** menu (**Mechanical** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **Rectangular Contour plot** from the report type menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
    a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
    c. Domain field with a drop-down selection list.
3. Under the **Trace** tab **Z** Component area, specify the information to plot as contours:
    a. In the **Category** list, click the type of information to plot. The selected Category provides the default plot name.
    b. In the **Quantity** list, click the value to plot.
    c. In the **Function** list, click the mathematical function of the quantity to plot.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

    > **Note:**
    >
    > Color shows valid expression.

    e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
6. Click **New Report**.

    This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*). You

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can edit the plot names in the project tree and the plot header text in the report synchronizes.

The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

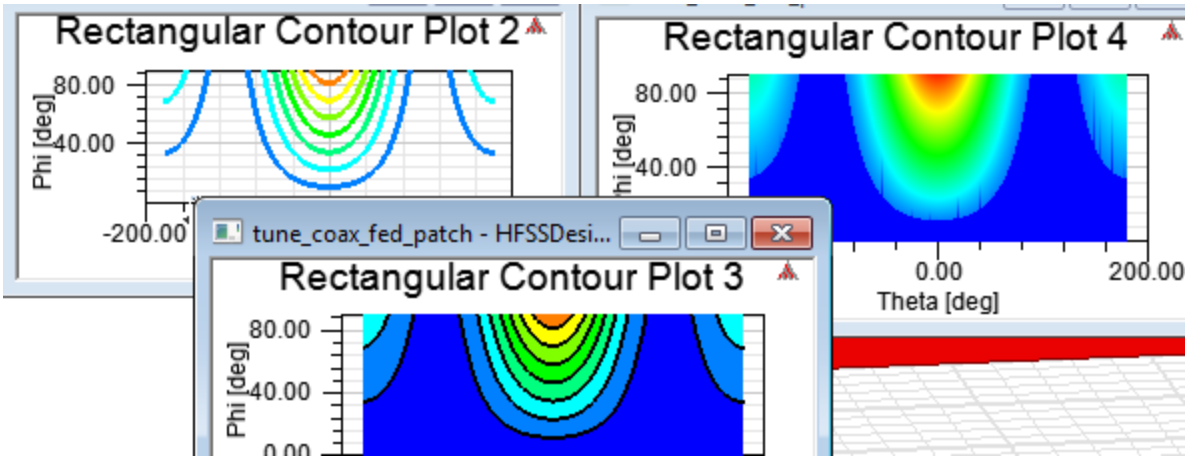
The Trace Properties for Cartesian plots include tabs for:

- Attributes tab, including Name, Line Style, Line Width, and Color.
- Contour tab, including Colormap Type, Color, Spectrum, IsoValType, whether to Overlay Contour Lines (on Fringe or Tone plots), Scaling parameters, and Number of contours and spacing.

Colormap Type can be Spectrum, Ramp, or Uniform.

Spectrum can be Rainbow, Temperature, Magenta, or Gray

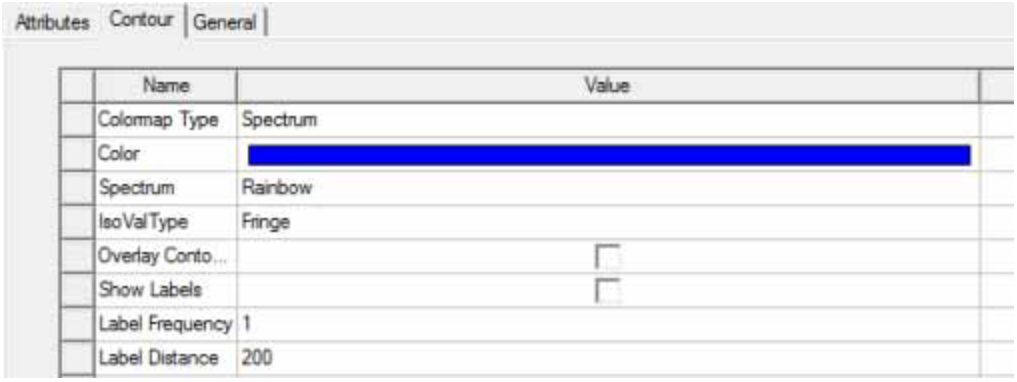
IsoValType can be Line, Fringe, or Tone.



General tab, including Back Color, Plot Area Color, Enable Y Axis Stripes, Field Width, Prevision, and whether to Use Scientific Notation.

You can also access these properties by double-clicking on the Contour plot and viewing the **Contour** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Creating Data Tables

A data table is a grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or other quantities.

1. Click **Mechanical**> **Results**> **Create *<type>* Report** or right-click **Results** in the Project Manager and click **Create *<type>* Report**.
2. In the **display type** menu, click **Data Table**.

   The *Report* dialog box appears.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed depend on the Solution type and the report ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.
   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
4. Under the **Trace** tab **Y** component section, select the quantity you are interested in and its associated function:
   a. On the **Category** drop-down menu, click the type of information to plot. The category selected provides the default name for the plot.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
   c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.

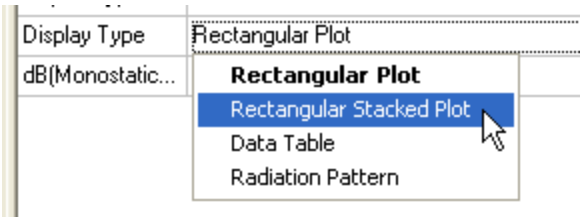
5. On the **Trace** tab **X (Primary sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The Y quantity will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

If you choose to print a data table:

- Selecting print "All" prints the whole table for current data page (if there is more than one data page).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Selecting print "Pages" prints user-specified pages.
- If the table is bigger than the screen view (that is, it has scroll bar), printing first scrolls right, prints until no more scrolling and then scroll down.
- The Page number appears at the bottom of the page, aligned at center.
- The table layout of each page follows the screen, but with no scroll bar will be printed, and no data page bar as on screen.

Delta Markers in 2D Reports

To view the difference between any two marker points in a report:

1. Set the first marker by clicking and holding the mouse button.
2. Move the mouse without releasing left button to another position, and then release the left button to create second marker.

In the marker text window, you see the difference between the two markers instead of the X, Y value of marker.

Working with Traces

A trace in a 2D or 3D report defines one or more curves on a graph. A trace in a data table defines part of the displayed matrix of text values.

The values used for a plot's axes (which may be X, Y, Z, phi, theta, or R depending on the display type) can be variables in the design, such as frequency, or functions and expressions based on the design's solutions. If you have solved one or more variables at several values, you can "sweep" over some or all of those values, resulting in a curve in 2D or 3D space.

A report can include any number of traces and, for rectangular graphs, up to 20 independent y-axes. Traces appear in the Project tree under their report. They can be selected, copied and pasted.

When you move a cursor over a trace in a report, the cursor changes to show that you can make a selection:

- For PC systems, the cursor changes to the color of the selectable trace.
- For Linux systems, the cursor changes to a solid black arrow, rather than the default black outline.

In general, to add a trace to a report:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a report in the Project window and right-click and select **Modify Report**.
2. In the *Report* dialog box, specify the Y component information.
   a. Specify the Category of information you want to plot from the drop-down menu.

      The Category drop-down menu lists the available categories for the Solution type and the current design. Selecting a category changes the Quantity and Function lists to represent what is available for that category.

   b. Specify the Quantity you want to plot by selecting from the Quantity list.

      The selected quantity appears in the Value field, operated on any selected function.

   c. Select the Function to apply to the specified quantity.
   d. The Value field shows the trace being readied for plotting on the Y-axis. This field is editable when the text cursor is present. You can modify the information to be plotted by typing the name of the quantity or sweep variable to plot along an axis directly in the text boxes.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies the currently specified Quantity and Function.
3. In the *Report* dialog box, specify the X axis information (for example Primary Sweep).
4. Click **Add Trace**.

   A trace is added to the traces list under its report icon in the Project tree. The trace represents the function of the quantity you selected and will be plotted against other quantities or swept variable values. Selecting a Trace in the Project tree displays the Properties window for that Trace. Selecting a trace in the report or legend displays the display Properties window for that trace.

   Trace icons can be selected, copied, and pasted for their definitions or their data. They can be selected and deleted from the Project Manager.

   By the default, the Trace name is the definition (the category, quantity and function). The trace will be visible in the report when you click **Add Trace**.

   Trace properties can be edited directly in the respective Properties windows or edited in the *Report* dialog box. To change the name or definition of a trace, see Editing Trace Properties. To edit other display properties of a trace, see Editing the Display Properties of Traces

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Trace Properties

To edit trace properties such as the name, Y Axis association, the component definition, the context, or the variables select the trace in the Project tree.

To edit a **trace name**:

1. Select the trace in the Project Manager.

   This displays a docked Properties window for the Trace.

2. Check the Specify Name box.

   This enables editing of either the Name field in the docked properties dialog box, or the Trace label text in the Project tree. Editing this name changes the display in the Legend and in the Project tree, but not the underlying Y-component definition.

> **Note:**
>
> To control the display of the Solution Name and Variation Key in the Legend, see Report 2D: Legend Tab.

To edit the Y Axis associated with the trace (2D Rectangular and Rectangular Contour plots):

1. Select the trace in the Project Manager.
2. In the docked properties window for the trace, select the Y Axis to be associated with the trace from the drop-down menu. Up to twenty independent Y axes can be added to a plot.

To edit a **trace component definition**:

1. Select the trace in the Project Manager.
2. In the docked Properties window for the trace, select the component field of interest, and select **Edit...** from the drop-down menu.

   This displays the edit Component field window from which you can edit the category, quantity and function.

3. Click **OK** to apply the changes and close the dialog box.

To edit a **trace Context**:

1. Select the trace in the Project Manager to display the docked properties window.
2. In properties window, click the Solution field or the Domain field. If other selections are possible, they can be selected from the drop-down menu.

To edit a **variable** for a trace:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the trace in the Project Manager to display the docked properties window.
2. Under the -Variables category, on the Families line, click the Edit button to display the Edit families dialog box.

   From this dialog box, you can select the Sweeps or Variations radio buttons. If other nominal values are available you can click the ellipsis button to select from a list.

Editing the Display Properties of Traces

Editing the display properties of traces differs for 2D and 3D reports. To edit the display properties of a trace for a 2D report:

1. Select a trace in an open **Report** window.
2. Click once on the trace to view settings in the docked *Properties* window, or double-click to open *Properties* dialog box.

   The display properties window for a 2D trace includes a **General** tab and an **Attributes** tab.

   The *General* tab properties apply to the general appearance of the plot. They include the Background color, Contrast color, Field width, and Whether to use Scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.)

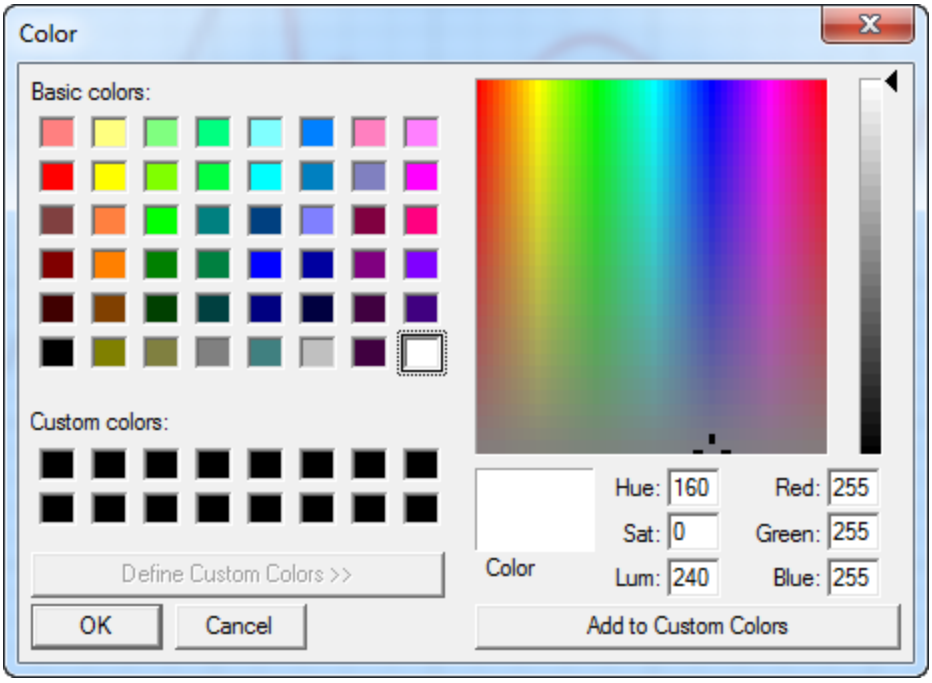
   The *Attributes* tab properties apply specifically to the trace. The defaults are set in the Report2D options. They include:

   - Name -- not editable by selecting the trace from the Report. It shows the characteristics of the trace as defined in the *Report* dialog box.

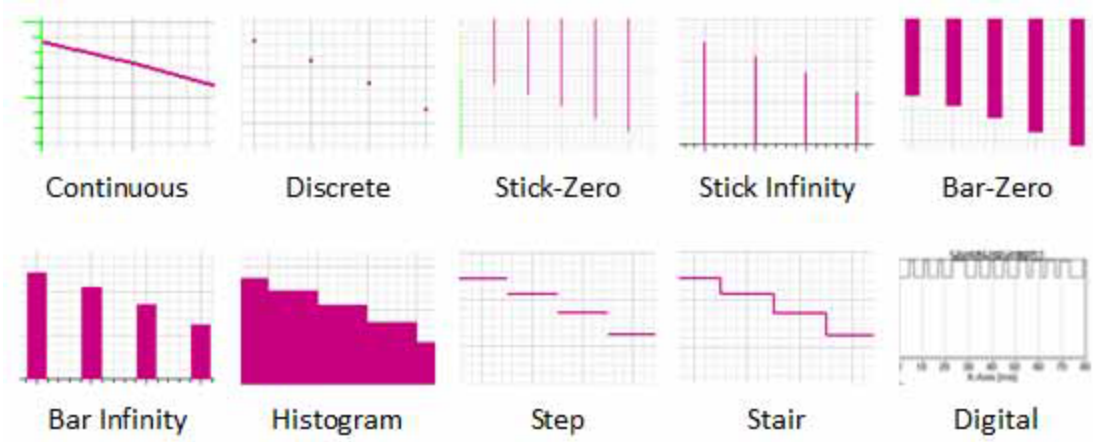
     To edit a trace name, see Editing Trace Properties.

   - Color -- shows the Trace color. Double-click to open a Color dialog box. You can select from Basic colors, or custom colors. You can define up to 16 custom colors by selecting or by editing the values for Hue, Saturation, Luminescence, and the Red, Green, and Blue.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



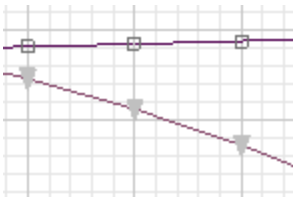
- Line style -- a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width -- a text field lets you edit the numeric value.
- Trace type -- the drop-down menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.



Notice the difference between Stair and Digital is that each Stair centers on a data point with transitions halfway between points, and Digital transitions from each data point to the next value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next four properties work together to define whether to show a symbol on data points, the symbol frequency, the symbol style, and whether to display the symbol as solid or hollow.



- Show Symbol -- whether to show a symbol at the data points on the line.
- Symbol Frequency -- how often to show symbols on the trace, based on the number of data points per symbol used. For example, specify 1 for one symbol per data point. Specify 10 for one symbol for every 10 data points.
- Symbol Style -- use a drop-down menu to select from box, circle, vertical ellipse, horizontal ellipse, vertical up triangle, vertical down triangle, horizontal left triangle, horizontal right triangle.
- Fill Symbol -- use the check box to set the symbol display as a solid or as hollow.
- Symbol Arrows -- use the check box to use arrows on the curve ends.

> **Note:**
>
> So that curves with single points always appear, Box is the default symbol.

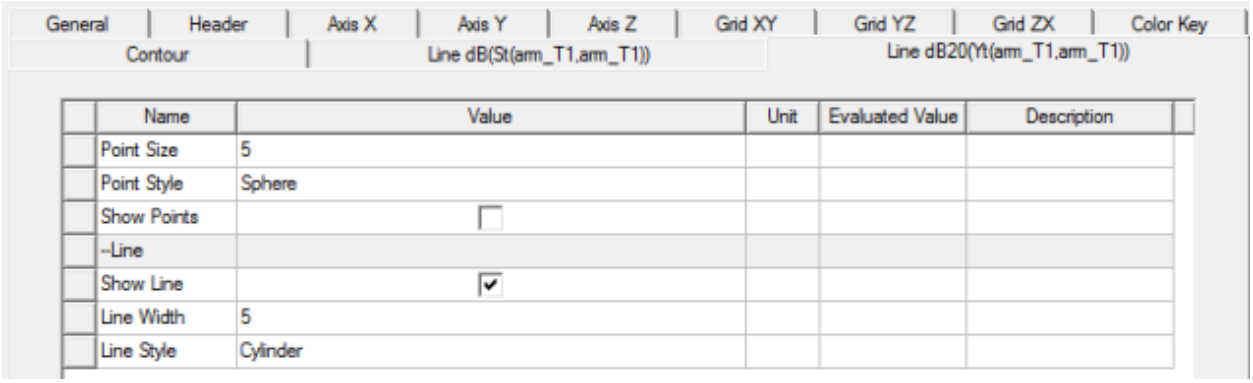
3. Edit the properties, if needed. Click OK to apply the changes and close the window.

To edit the display properties of a trace in a 3D report:

1. Click on the trace. This opens a Properties dialog for the plot with a tab named for the trace selected.
2. The editable properties include Point size, Point Style, whether to show points, whether to

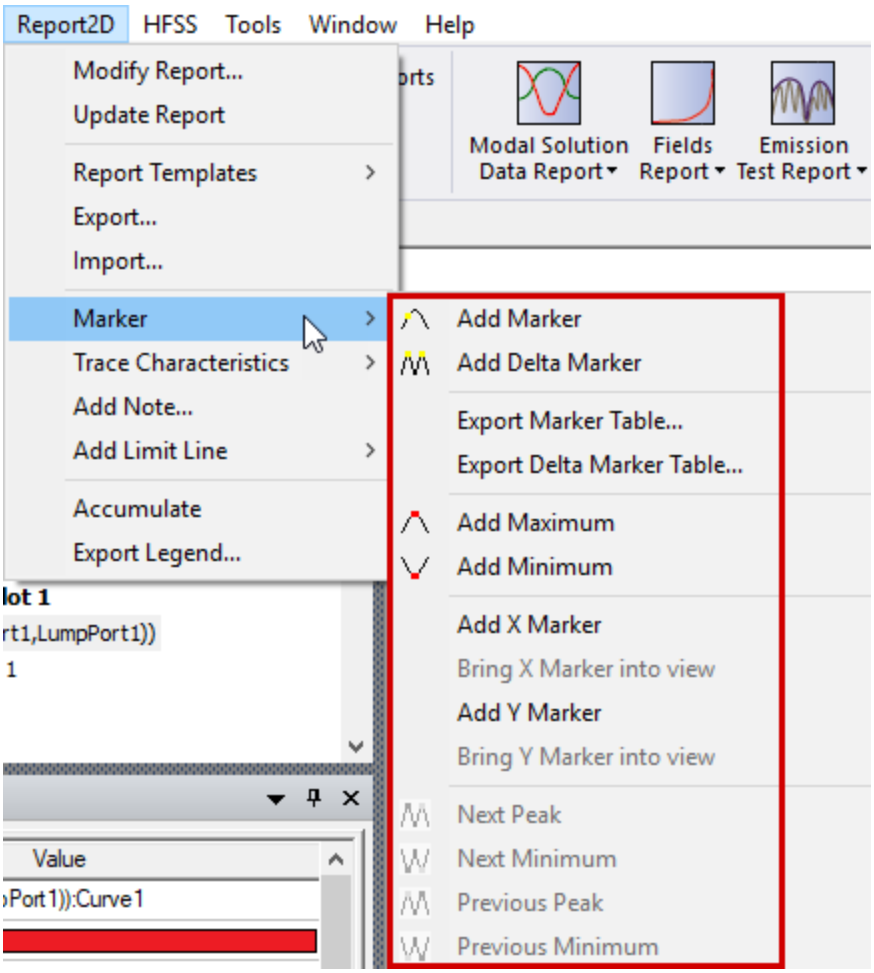
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

show line, line width, and line style.



Adding Data Markers to Traces

The Reporter includes **Report 2D** > **Marker** > menu commands and icons:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access these commands from the shortcut menu that appears when you right-click inside a report plot window.

These command let you add markers to traces. A marker appears as "m*N*" at the marked point, where *N* increments from 1 as you place additional markers. Each marker can be selected and has editable properties including name, font, background and color. As you place markers, one or more marker legends may be displayed, depending on the **View> Active View Visibility** settings for the legends. The main marker legend appears in the upper left of the plot, and lists the marker names and their X and Y values in a table. You can control the number format for the table values via the properties window, general tab. Under *Marker/Other Number Format*, you can specify field width, precision, and whether to use scientific notation. This value is independent of the Axis tab number properties. A separate marker legend appears for Delta Markers, as described for the **Delta Marker** command.

When you enter Marker mode, the cursor arrow is accompanied by an "m" while a circle on the selected trace shows the current position for a potential marker.

To end Marker mode, right-click to display the shortcut menu, and select **End Marker Mode**.

The available Marker mode commands and associated icons are the following:

- **Marker** – this command lets you place a marker at an arbitrary point on a selected trace.
- **X Marker** – this command adds up to 10 movable markers at the origin of the plot with a vertical line rising from the X axis. Each added marker has its own color and editable properties. To move an X marker, click on the X label and drag it to the desired location. The label at the bottom of the line gives the X coordinate, and flag on the vertical line identifies the Y coordinate on the trace. A trace property lets you lock the drag feature to leave the marker in place. The X markers are cleared by the **Clear All** command.
- **Bring X Marker into view** – this command is enabled if an X Marker is not visible in the plot. It allows you to select from a list of existing X Markers to bring into view.
- **Y Marker** – this commands adds up to 10 Y Markers with a horizontal line extending from the Y axis. For more detail on Y Markers and their use, see Y Markers in stacked XY plots.
- **Bring Y Marker into view** – this command is enabled if a Y Marker is not visible in the plot. It allows you to select from a list of existing Y Markers to bring into view.
- **Maximum** – places a marker at the Maximum value on the selected trace.
- **Minimum** – places a marker at the Minimum value on the selected trace.
- **Delta Marker** – enters delta marker mode, placing a circle on the selected trace. Clicking on the trace sets an initial point and subsequent clicks on arbitrary points on the trace place additional markers until you leave marker mode. These markers have their own legend, which includes the following information for each pair of markers specified:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Delta(X)	Delta(Y)	Slope(Y)	InvSlope(Y)
d(m2,m3)	0.4700	1.8319	3.8976	0.2566

- **Next Peak** – moves a selected marker on the next peak on a trace. You must exit marker mode and select a marker to enable this command.
- **Next Minimum** – moves a selected marker to the next minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Peak** – moves a selected marker on the previous peak on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Minimum** – places a marker on the previous minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Go to Start** (Right arrow) – moves a selected trace marker to the first data point. Enabled by leaving marker mode and selecting a marker.
- **Go to Previous** (Left arrow) – moves a selected trace marker to the previous data point.
- **Go to Next** – moves a selected trace marker to the next data point.
- **Go to End** – moves a selected trace marker to the last data point.
- **Next Curve** – selects the next curve in the report, based on the order in the trace legend.
- **Previous Curve** – selects the previous curve in the report, based on the order in the trace legend.
- **Clear All** – clears all markers on a report.

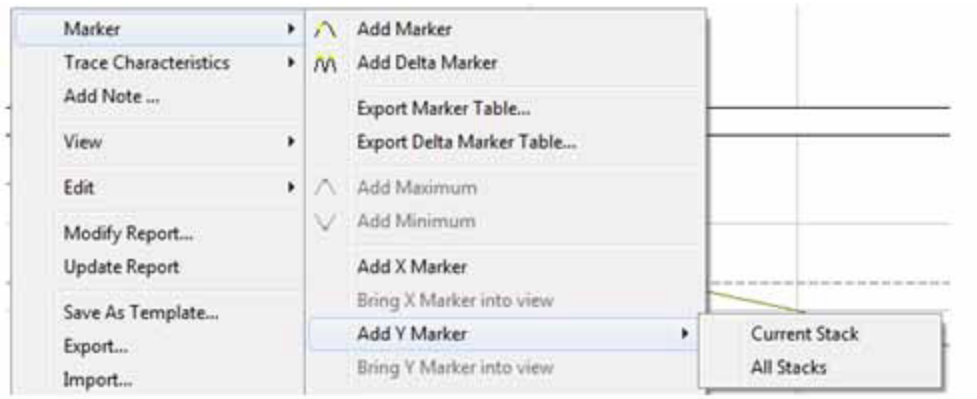
Y Markers in Stacked XY Plots

Y Markers allows for easy analysis and comparison of curves at a particular y-coordinate. Y Markers can be used to compare stacked curves.

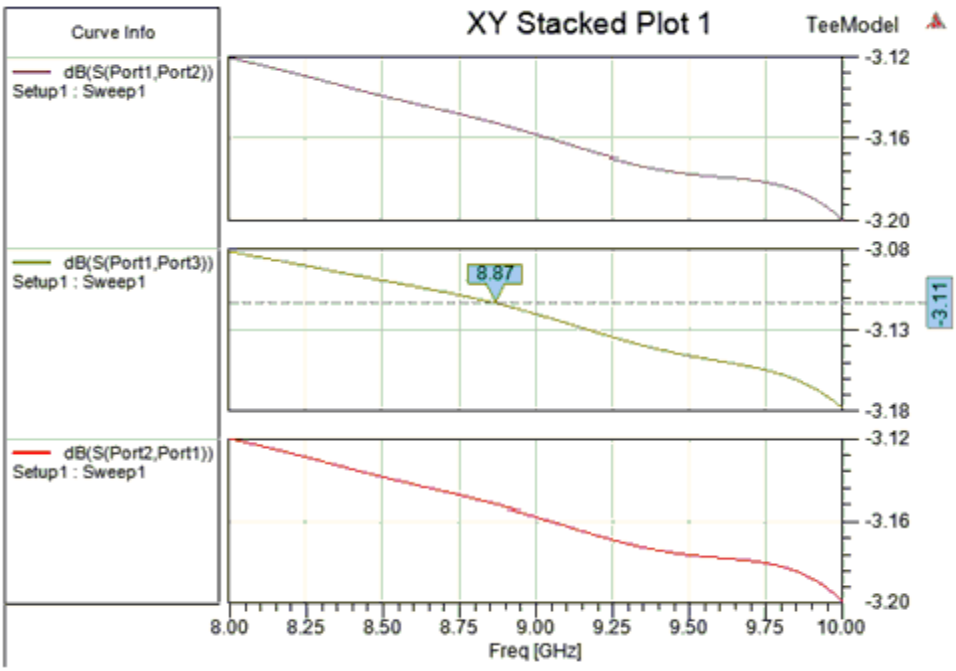
Creating Y Markers in Stacked Plots

There are two ways to create Y Markers in Stacked Plots. You can create a Y Marker for a particular stack or for all stacks. Right-clicking on any stack shows the following shortcut menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

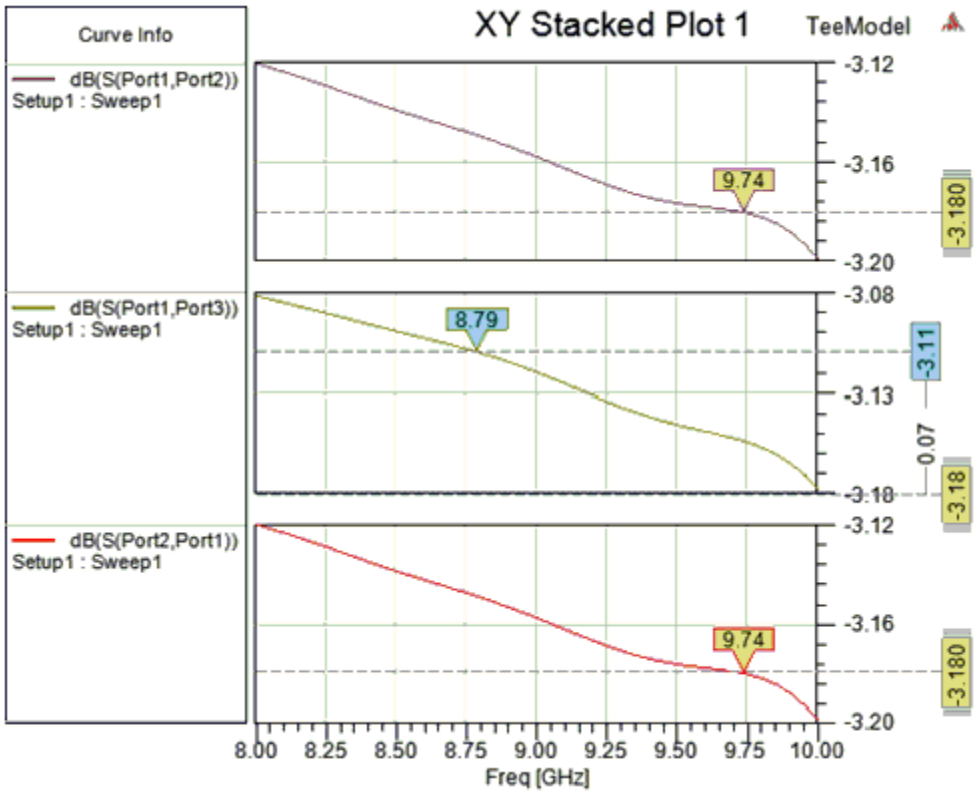


Add Y Marker > **Current Stack** creates a Y marker for the stack on which user performed right mouse button click. The following figure shows that a Y Marker was added to second stack only:



Add Y Marker > **All Stacks** creates one Y marker in each stack with same value. Initially this value is the minimum Y value of the Y ranges in all the stacks. This is shown is figure below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

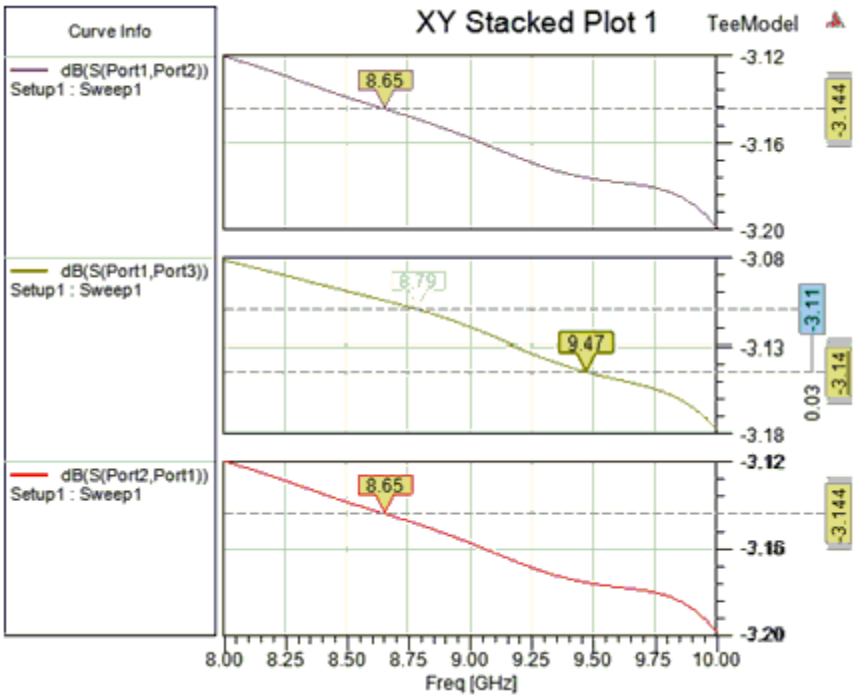


Notice that the Y Marker for All Stacks has a different appearance than the Y Marker for a particular stack, that is, it has double parallel lines above and below the Y Marker textbox.

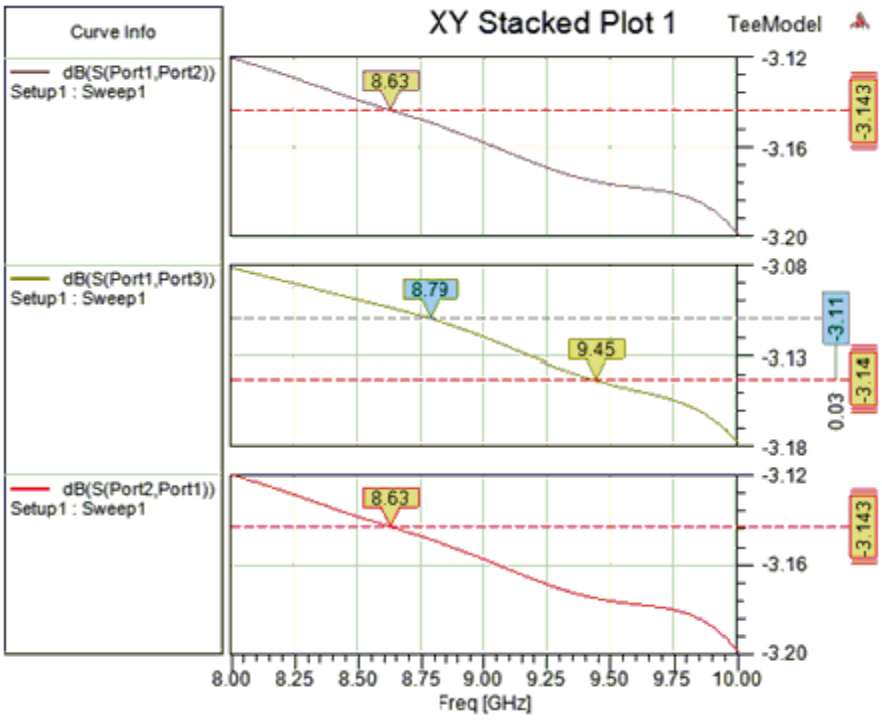
Synchronized Y Markers

All the "same" Y markers for all stacks are synchronized, that is to say that if one Y marker is dragged or it's value is changed, all the "same" Y markers in all the stacks will change their position too. The following figure shows that when Y marker in bottom stack was dragged, Y marker in top stacks moved as well:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



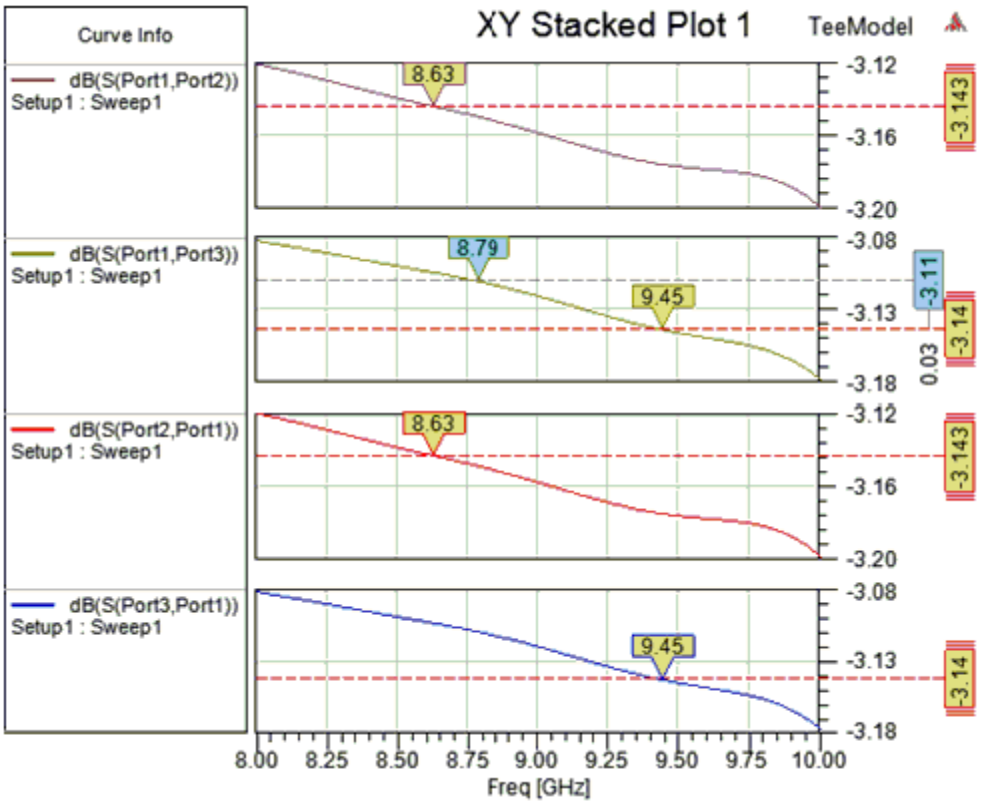
Also if a property of any one Y Marker is changed, all the "same" Y Markers show the change in property as well. For example the following figure shows that when the line color of a Y Marker in the top stack was changed to red color, a Y Marker in bottom stack show the same line color as well:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Y Markers for the New Stack

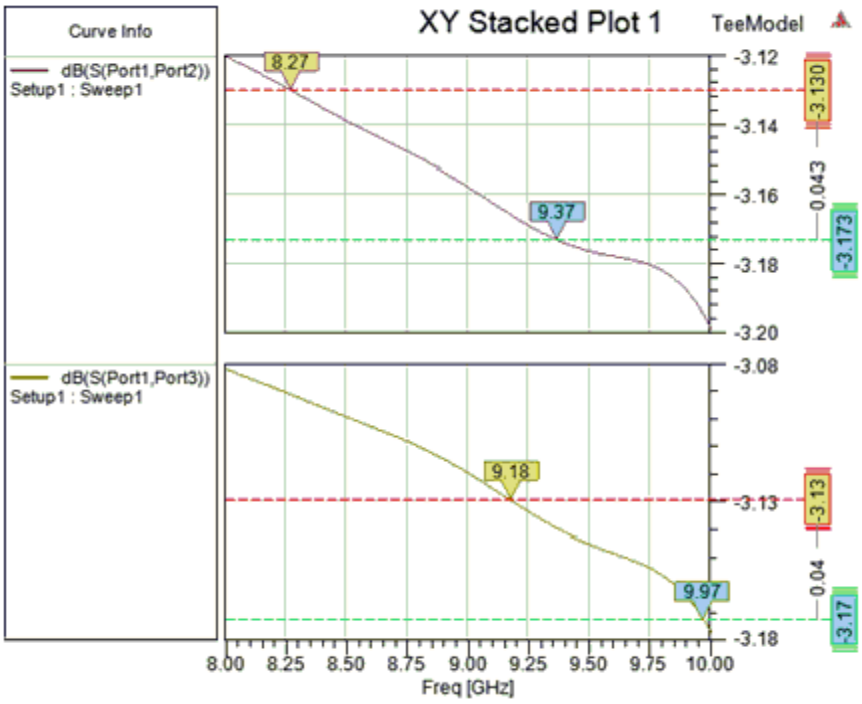
When a new curve is added to the plot, it gets all the Y Markers for all stacks in other stacks, excluding the Y Marker for particular stacks. The following figure shows that when the new curve "dB(S(Port3, Port1)" was added, a Y Marker was added to it with value -3.14 and it has all the same properties as other "same" Y Markers in other stacks:



Y Marker Delta Annotations

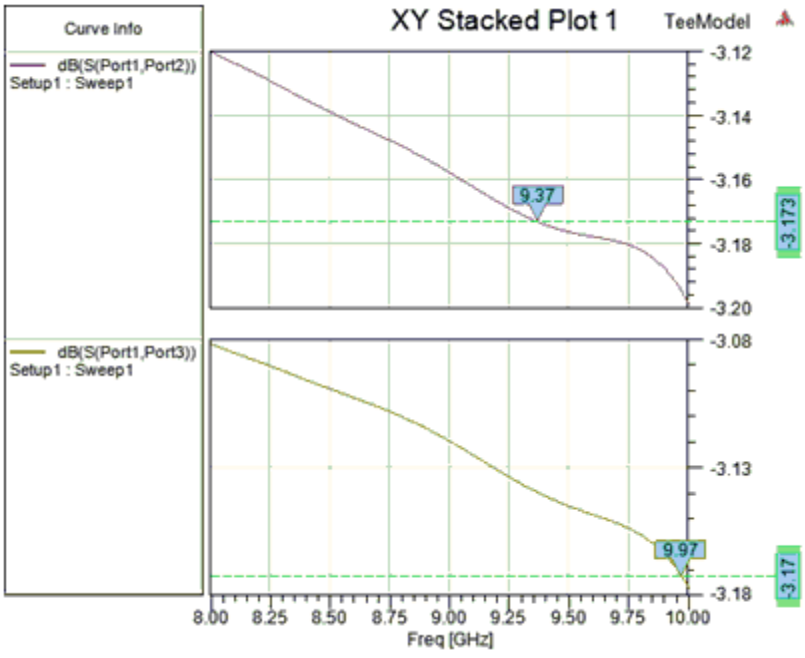
When two more Y Markers are present in a Stacked Eye Diagram, then delta annotations are shown between a pair of adjacent Y Markers in all the stacks, as shown in figure below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Deleting a Y Marker

To delete a Y Marker, select a Y Marker in any stack and press the Delete key. This action will also delete all the corresponding Y Markers in all the stacks. For example, when the Y Marker with value -3.13 (red Y Marker) was deleted from the bottom most stack, all of the corresponding Y Markers were also deleted:

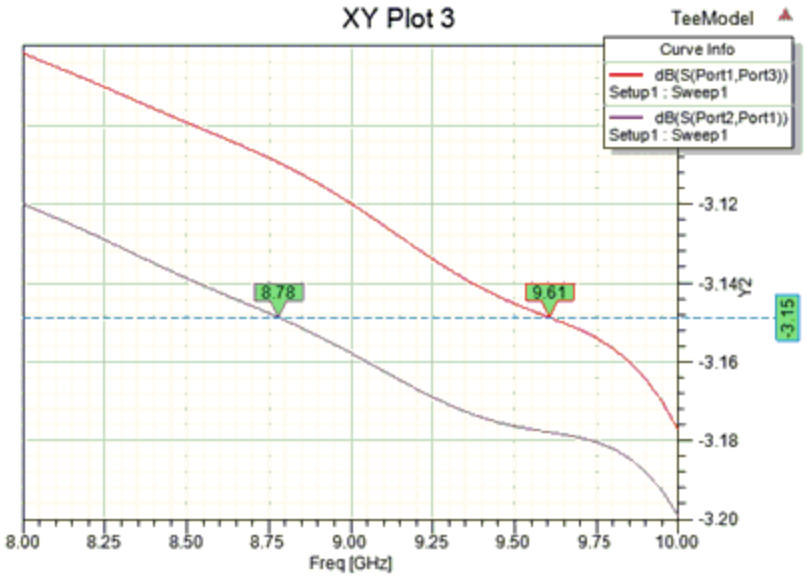


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

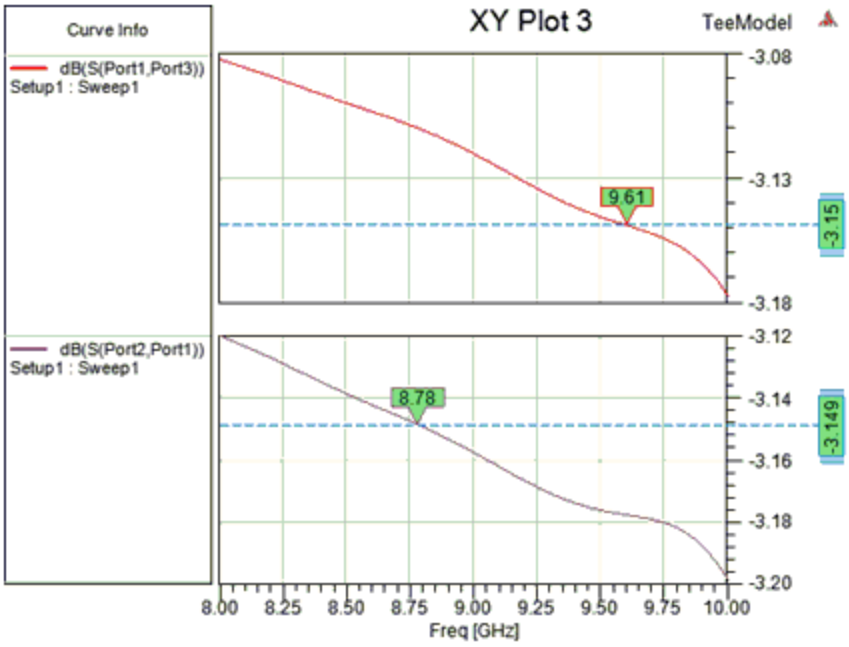
Note that on deleting a stack, Y Markers in other stacks are not affected.

Converting Rectangular XY Plot to Rectangular Stacked XY Plot

The following figure shows a Rectangular XY Plot with two curves and a Y Marker with value -3.15 (blue Y Marker):



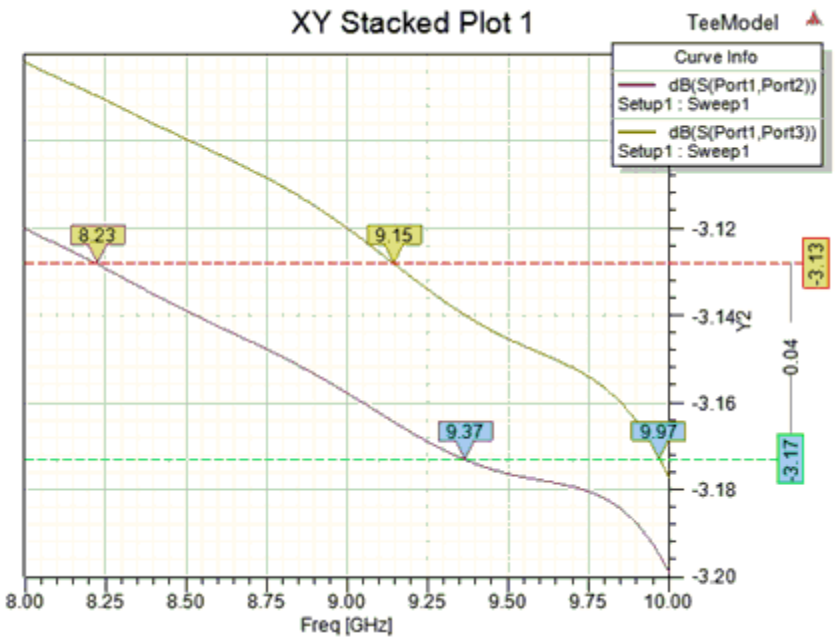
If you change the Display Type property of this plot to Rectangular Stacked Plot then a Rectangular Stacked XY Plot is created with each curve in its own stack and a Y Marker is shown in each stack with value -3.15 (blue Y Marker):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Similarly when you change a Rectangular Stacked XY Plot to a Rectangular XY Plot then all the "same" Y Markers in all the stacks are shown as a single Y Marker in Rectangular XY Plot as shown in following figures:

The Rectangular Stacked XY Plot in the previous figure, when converted to Rectangular XY Plot, looks like the following figure:



Discarding Report Values Below a Specified Threshold

To prevent real small numbers from skewing a plot, you can discard small values (below a specifiable threshold).

1. Double-click the X or Y axis of interest on an open plot display.

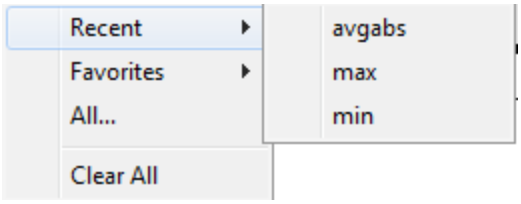
   This opens the **Properties** window for the Axis.

2. Under the **Axis** tab, use the scroll bar to find the **Specify Discard Values** property.
3. Click the check box to enable the property.
4. Enter a value in the **Discard Below** field. Units specified elsewhere in the Axis property are applied to this value. The Discard Below text box is inactive if the Specify Discard Values check box is not enabled.
5. Click **OK** to apply the Discard Values to the report.

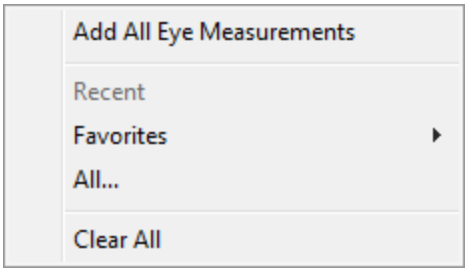
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding Characteristics to a Trace

There are several options for adding characteristics to a trace. When you click **Report2D** > **Trace Characteristics**, or right-click a selected trace, the shortcut menu is displayed. The following example shows the menu with expanded **Recent** selections.



The following shows the shortcut menu for Eye Measurements, which includes the **Add All Eye Measurements** option. You can use this option to add all eye measurements at once.



Adding a Recently Used Trace Characteristic

If you recently used a characteristic, you can add it to a selected trace by selecting from a list of recently used characteristics. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a recently used characteristic to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **Recent**, and then select the function you want. The specified characteristic is added to the trace.

Adding a Trace Characteristic from Favorites

You can add a trace characteristic to a selected trace by selecting from a list of favorites. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a favorite characteristic to a selected trace:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **Favorites** and then select the function you want. The specified characteristic is added to the trace.

Adding Trace Characteristics to your Favorites

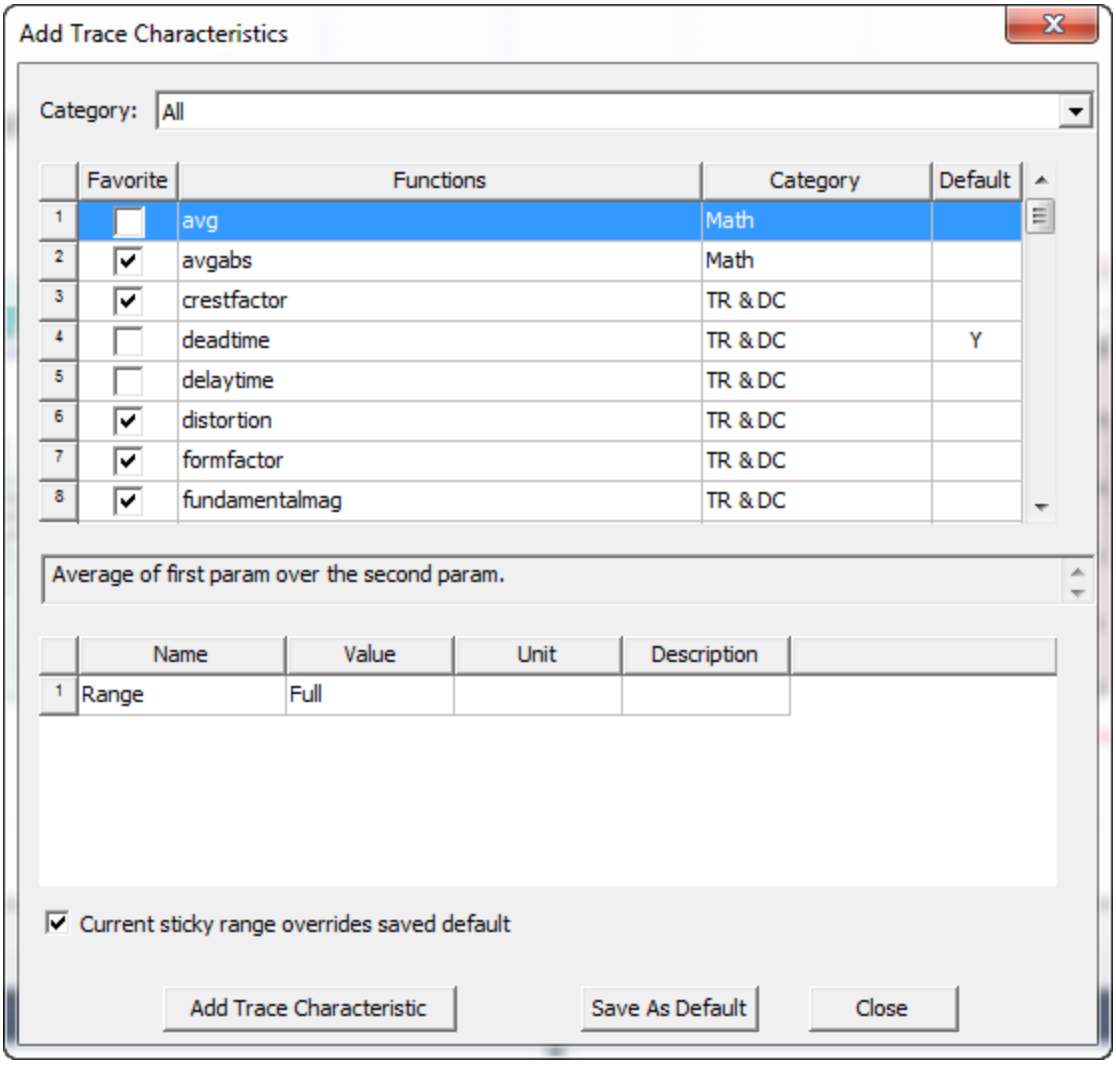
You can add trace characteristics to your list of favorites.

To add characteristics to your list of favorites:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the short cut menu.
3. Select **All**.

   The *Add Trace Characteristics* dialog box displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click the **Favorite** check box in front of any function you want to add to your Favorites. You can define as many favorites as you need, but no more than ten are displayed in the menu, and they are displayed in alphabetical order.
5. Click **Close**. You can view the current favorites by selecting Favorites in the Category drop-down menu.

> **Note:**
>
> You can remove favorites by clearing the **Favorite** check box for one or more functions, and clicking **Close**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

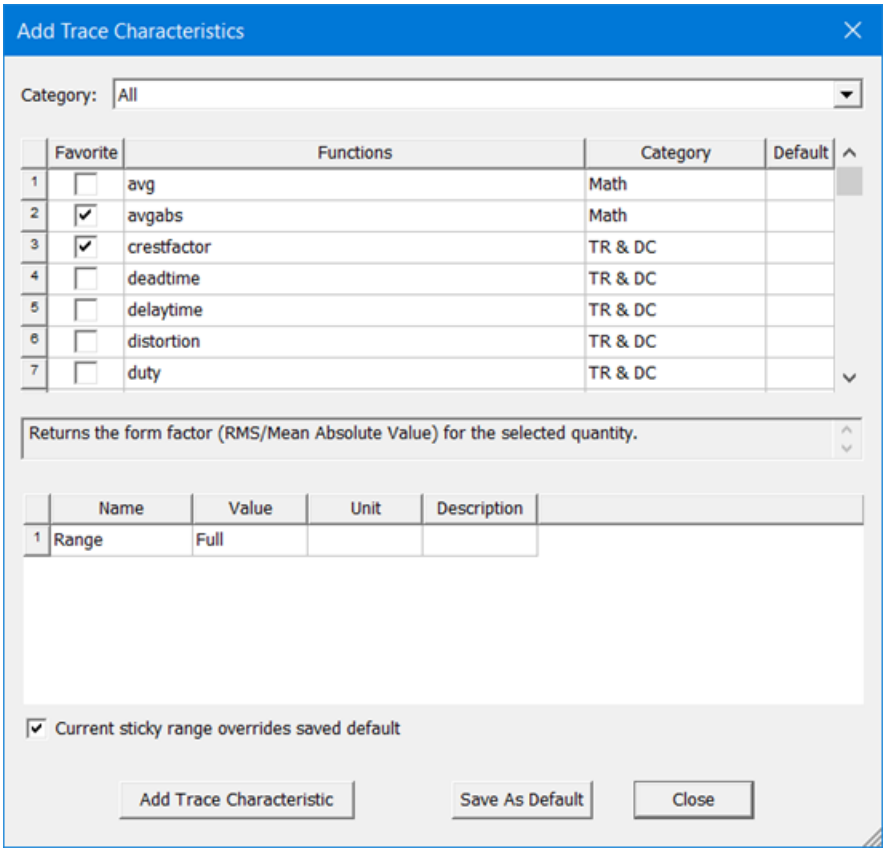
Adding Characteristics Using the Add Trace Characteristics Window

You can add characteristics to a selected trace by selecting from the **Add Trace Characteristics** window.

To add additional characteristics to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **All**.

   The **Add Trace Characteristics** window displays:



4. Select the desired **Category**.

   The available categories depend on the plot, and the selecting of a category displays its associated functions.

> **Note: Categories** and **Functions** only appear if they are available for that plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Functions
Recent	Displays the most recent functions used, sorted by the time they were added.
Favorites	Displays all favorites. The defaults are avg, max, min, and pk2pk.
All	Displays all available functions.
Math	avg, avgabs, integ, integabs, max, mean, min, pk2pk, pkavg, ripple, rms, rmsAC, stddev, sum, variance, XatYMax, XatYMin, XatYVal, XWidthAtYVal, YatXMax, YatXMin, YatXVal
PulseWidth	pulsefall9010, pulsefront1090, pulsefront3090, pulsemax, pulsemaxtime, pulsemin, pulsemintime, pulsetail50, pulsewidth5050, pw_minus, pw_minus_avg, pw_minus_max, pw_minus_min, pw_minus_rms, pw_plus, pw_plus_avg, pw_plus_max, pw_plus_min, pw_plus_rms
Overshoot/ Undershoot	overshoot, overshootAbs, undershoot, undershootAbs
TR & DC	crestfactor, deadtime, delaytime, distortion, formfactor, fundamentalmag, risetime, settlingtime
Error	iae, ise, itae, itse
Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq
Radiation	lSidelobeY, rSidelobeY, lSidelobeX, rSidelobeX, xdb10Beamwidth, xdb20Beamwidth
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime, MinEyeWidth, MinEyeHeight
TDR	Shunt_C_in_pF, Series_L_in_nH

For a selected function, the *Add Trace Characteristics* dialog box displays the function's purpose in a text field.

5. Some categories and functions call for you to specify one or two additional values in a table. You can save these values using the **Save as Default** button. The Default column shows a Y if there is a saved default value for the function.
6. Select the **Current sticky range overrides saved default** check box if you do not want the range value in the table to be changed when the function selection is changed: the current range value becomes the "sticky range." If the check box is not checked, the range value is updated from the saved default values and becomes a new sticky range.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click the **AddTrace Characteristic** button to add the specified characteristics to the trace.
8. Click **Close**.

Copy and Paste of Report and Trace Definitions

You can copy and paste report and individual trace definitions within a single design or across designs. The report or trace definition will be evaluated within the context of the target design or report.

> **Note:**
>
> - If the report or trace definition contains properties that do not exist in the target design (for example, a port name) an error will be posted that indicates a solution does not exist for this trace
> - You must copy and paste trace definitions between the same report types. For example, you cannot copy a trace from a Modal Solution Data report and paste it in a Far Fields report.

To copy a Report Definition:

Right-click the report name in the Project Manager and select **Copy Definition** from the shortcut menu.

To paste the Report Definition:

Right-click Results in the Project Manager of the target design and select **Paste**.

A new report is created and it contains the copied definitions.

To copy an individual Trace Definition(s):

Right-click the trace or traces under a report name in the project tree and select **Copy Definition**.

To paste the Trace Definition(s):

Right-click the report in the target design to which you would like to copy the trace or traces and select **Paste**.

A new trace(s) is added to the report and it contains the copied trace definition(s).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you copy and paste a report or trace definition to a design that contains a definition with the same name, then an incremented number is appended to the pasted report or trace name.

Copy and Paste of Report and Trace Data

You can copy and paste report and individual trace data within a single design or across designs. The report and trace definitions and all underlying data within the report or trace are copied and pasted to the target design or report.

To copy all data from a report:

Right-click the report name in the project tree and select **Copy Data**, or use the menu bar **Edit** > **Copy Data**, or right-click within a plot to display a shortcut menu with **Copy Data**.

To paste copied report data:

Right-click Results in the project tree of the target design and select **Paste**.

To copy data from an individual trace(s) in a report:

Right-click the trace or traces under a report name in the project tree and select **Copy Data**.

To paste copied trace data:

Right-click the report in the target design to which you would like to copy the trace data and select **Paste**.

> **Note:**
>
> If you copy and paste report or trace data which contains the same name definition as a report or trace in the target design then an incremented number will be appended to the pasted name.

Removing Traces

You can remove traces from the traces list in the following ways:

To *remove one trace* from the report:

- Select the trace you want to remove from the Project tree, and then click **Delete**.

To *remove all traces* from the report:

- Select all the traces and click **Delete**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Removing All Trace Characteristics

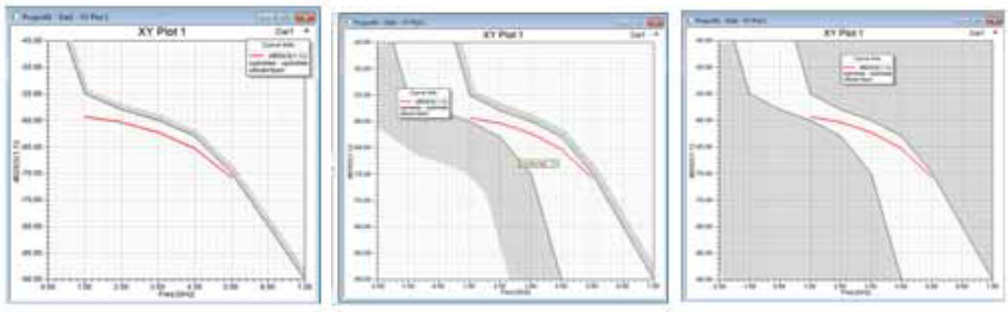
1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the shortcut menu.
3. Select **Trace Characteristics** > **Clear All**.

Trace characteristics are cleared from the selected trace.

Limit Lines in Cartesian Plots

Limit lines are simple graphical representation of constraints on XY plots. These are modeled as a sequence of XY point pairs, or as offsets from a selected curve. You can designate a single limit line to delineate an upper limit, or two lines to delineate upper and lower limits, or upper and lower offset lines simultaneously.

You can control the display properties of the line including color and hatch width in pixels. These lines are available only on XY plots (and not on the XY-like plots: bode, stacked etc)

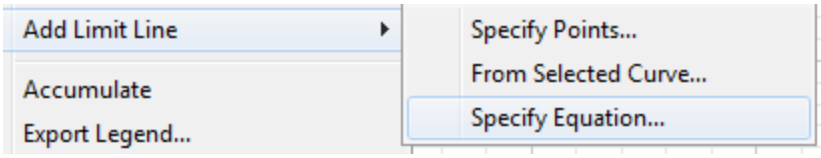


> **Note:**
>
> Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked. On 2D Plots, the axes extents are based on the extents of the curves.

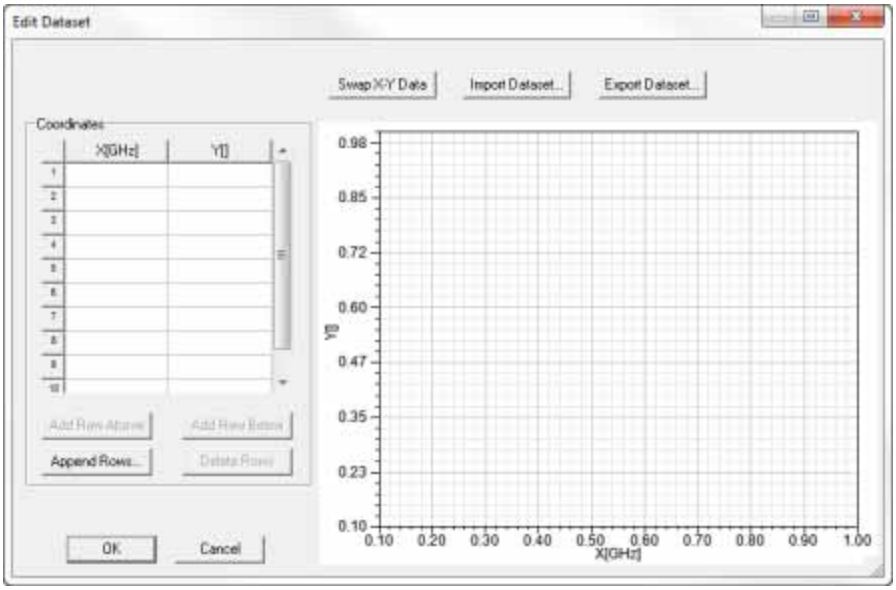
To create a limit line:

1. Click **Report2D** > **Add Limit Line** or right-click an XY plot and select **Add Limit Line...** from the Context menu. You then select whether to **Specify Points**, or **From a Selected Curve**, or **Specify Equation**.

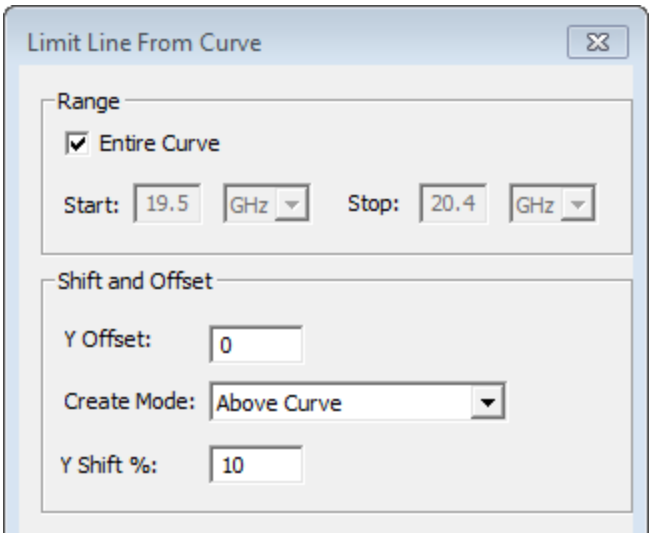
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select **Specify Points** to open an *Edit Dataset* dialog box so that you can specify points.



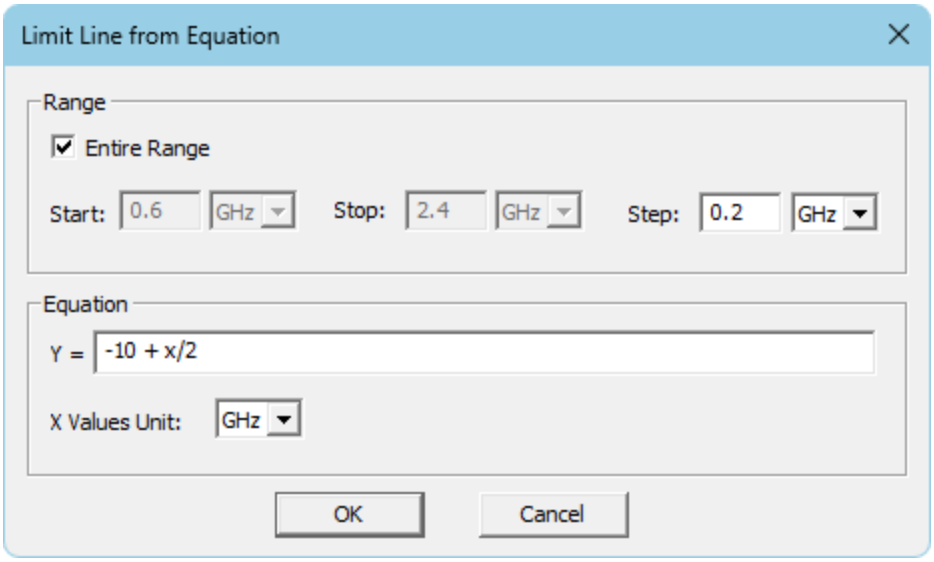
Select **From Selected Curve** to open the *Limit Line From Curve* dialog box.



Select **Specify Equation** to open the *Limit Line from Equation* dialog box. Specify the desired **Equation** (y as a function of x). Choose the **X Values Unit** from the drop-down

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu. The available units depend on the type of sweep quantity (frequency, distance, and so on). The default choice is the current X axis unit for the plot.



2. You can use the *Edit Dataset* dialog box to
   - enter the EY values directly.
   - import XY values from a .tab file.
   - export Dataset to a file.

   If you require additional data points, you can use the buttons to **Append Rows** to the Coordinates table. If you select a row in the Coordinates table, you can then use the buttons to **Add Row Above**, **Add Row Below** the selected rows, or **Delete Rows**.

   You can use Shift+click to select multiple adjacent rows, or Ctrl+click to select any rows for deletion.

   > **Note:**
   >
   > Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale and so on). This Y axis association defaults to the first available Y axis when the limit line is created. However, if the plot contains multiple Y axes, it can be associated with a different Y axis later via its properties tab.

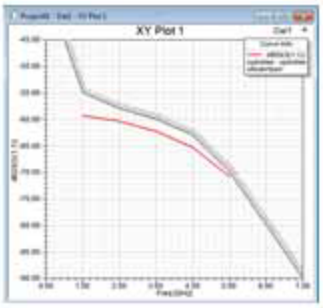
3. You can use the *Limit Line From Curve* dialog box to create a limit line with a:
   - Range using the Entire Curve, or a specified Start and Stop.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you uncheck “Entire Curve,” the Start and Stop fields are enabled and initialized based on the zoom level.

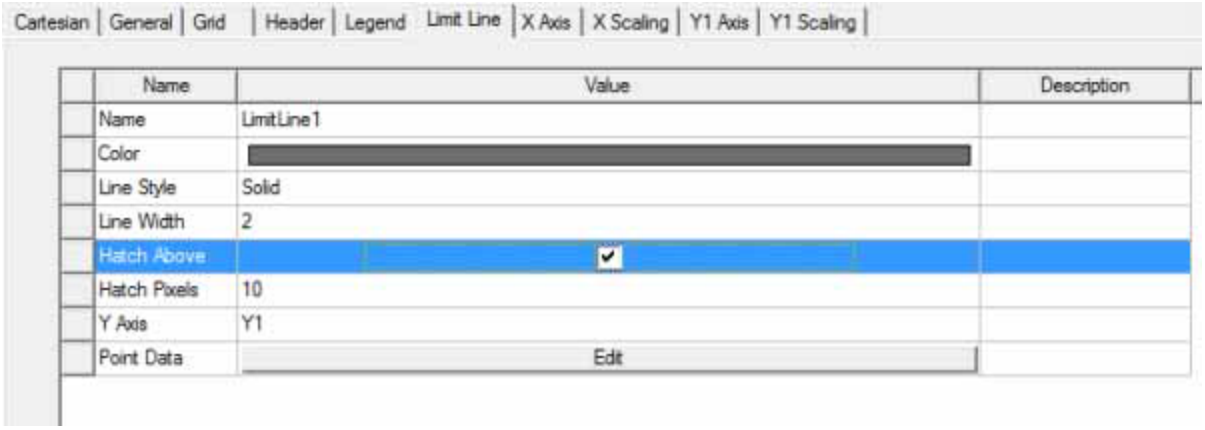
- Shift and Offset relative to the Y, either as a Y Offset value or as a Y Shift %.
- Create Mode as Above Curve, Below Curve, or Above and Below Curve.

4. You can use the *Limit Line from Equation* dialog box to create a limit line based on an equation in the form Y = f(x).
    - Only x variable is allowed
    - x value is linearly sampled and f(x) is evaluated to get y. By default, the entire range for plot is covered. You can uncheck **Entire Range** to specify Start and Stop. You can always edit Step and unit values for x.
    - If y results in NaN, an error is indicated and limit line is shown.
    - If y results in Inf value, then it maps to the MapInfValue specified on its associated y axis.
    - Selecting the limit line added in such a way should show: Start, Stop, Step, and Equation in limit line property window. You can edit these values in the Properties window.
5. Once you click **OK**, the limit line, or lines, you define are added to the plot. Each limit line divides the plot into two regions within the context of its length. By default, the upper region is hatched to designate constraint violation.



6. You can select the limit line in the plot to edit its properties via the **Limit Line** tab of the plot properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

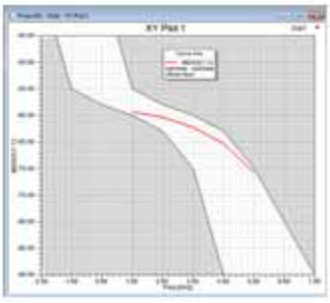


- Line properties: Color, Style and Width
- Y axis association
- Hatch properties
- Hatch width in pixels
- Hatch direction (hatch above or below the Limit line)
- The point definition of the limit line itself.

For a limit line specified as an Equation, the Properties also include:

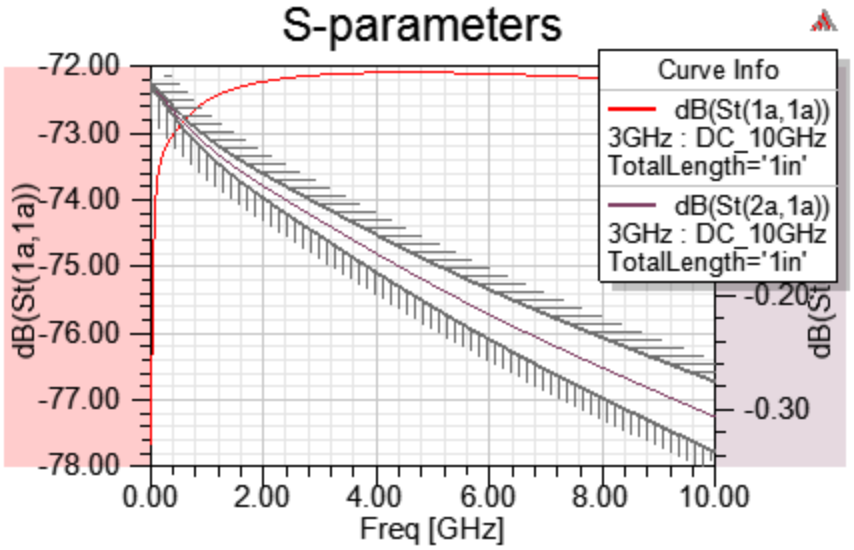
- Start, Stop, Step, and Equation values.

7. If you add a second limit line, you can designate it as hatch below by unchecking **Hatch Above** to produce a tunnel marking upper and lower constraints.



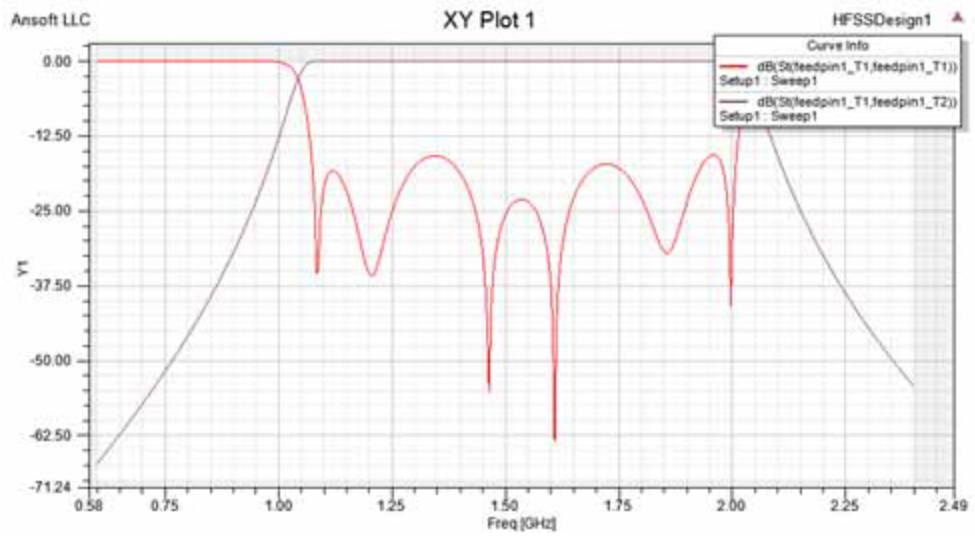
The following example shows the a limit line from curve plot, where the Hatch Above property for the lower limit line has been unchecked.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



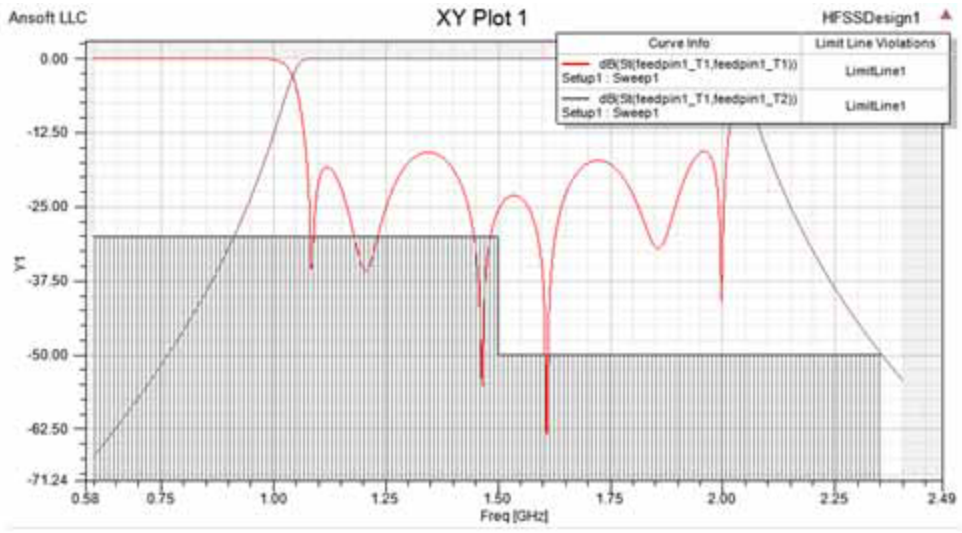
Limit Line Violations

You can use a plotting feature to help you discern whether a curve violates a limit line or not. Consider following plot which shows two curves:

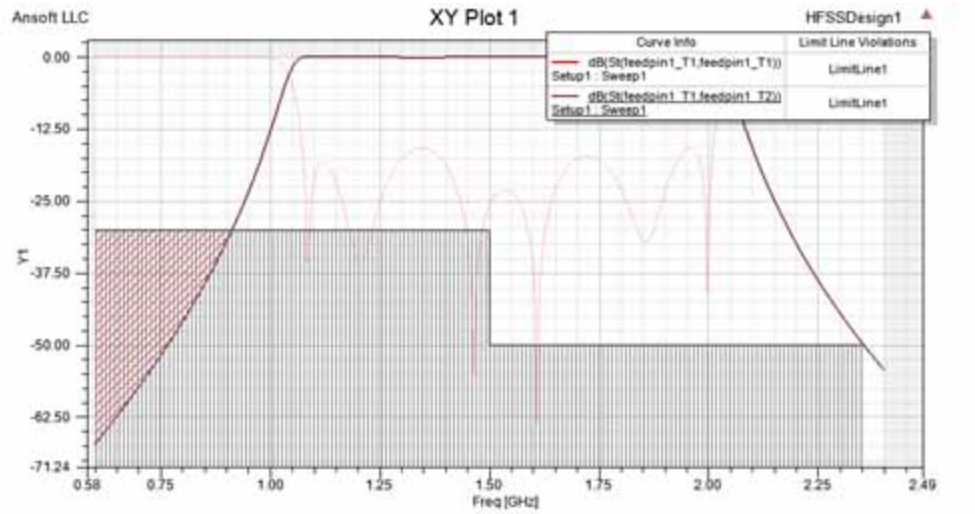


Suppose that the response cannot be below -30 dB until 1.5 GHz, and cannot be below -50 dB at 1.5 GHz or higher. You add a limit line for this requirement in the plot using 'Add a Limit Line' functionality. The plot automatically calculates whether a curve violates this requirement, that is, the limit line and show it in the legends window, as shown in following figure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



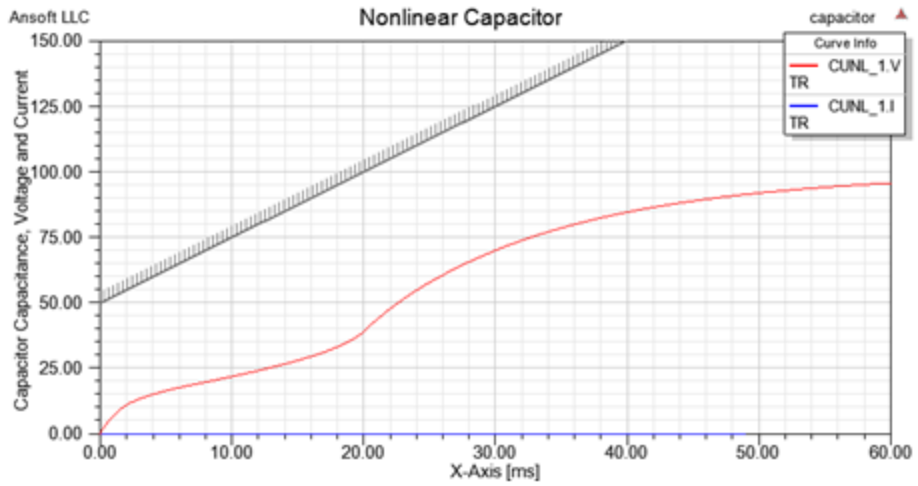
If a curve is selected, then the plot shows the region of the curve that violates the limit line (shaded with slanted red lines), as shown in following figures:



Note:

As shown in the following figure, if no curve violates a limit line, then the limit line hatching is restrained to 10 pixels in length. The minimal hatching keeps the focus on the curve traces rather than the limit lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Error Handling

If an error is encountered while calculating Limit Line Violations for a particular plot trace, the *Curve Info* window shows "NaN (<*Limit Line name*>)" under the *Limit Line Violations* column for that curve. For example, if a value is not available or goes to infinity at a particular point, the program is unable to evaluate that point. One of the coordinates is "Not-a-Number" (NaN).

Sweeping Values Across a Distance

1. If you are plotting a field quantity along a line, define a polyline object in the problem region.
2. In the *Report* dialog box, click the line geometry of interest in the **Geometry** list.
3. Specify the quantities you want to plot along the axes.
4. For the **X (Primary Sweep)**, select the **Distance** variable.

   The values at which the selected quantity or quantities will be plotted are listed to the right. By default, a post-processing polyline object is divided into 100 equally spaced points.

5. For Near field, to plot the selected quantity or quantities at every point on the line, select **All Values**.

   For Near field, to plot the selected quantity, or quantities, at specific points on the line, clear the **All Values** option, and then select the point values on which you want to plot.

> **Note:**
>
> All maximum near-field data calculated by Mechanical is at their maximum over the selected line object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Function for a Plot

The value of a quantity being plotted depends upon its mathematical function, which you select from the **Trace** tab **Function** list in the *Report* dialog box. The available, valid functions depend on the type of quantity (real or complex) that is being plotted. The function is applied to the quantity which is implicitly defined by all the swept and current variables. For example, "S(11)" is the value of the S-parameter for every swept combination of variables (e.g., "height", "frequency"). (A smaller set of functions appears for the Function list in the Output Variables dialog.)

These functions can also be applied to previously specified Quantities and Functions as **Range Functions** when using the Set Range Function dialog box.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the following functions in the **Trace** tab **Function** list or type them directly into the Yor X field, if necessary.

abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
ang	Magnitude of an angle.
ang_deg	Angle (phase) of a complex number, cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates ang_deg expressions in radians. See ang_deg_val.
ang_deg_val	Angle (phase) of a complex number in unitless degree values, and suitable for use in Optimetrics, which works in SI values. Returns simple numbers.
ang_rad	Angle in radians.
arg	Argument of a complex number. It is the angle the complex number makes with the positive x axis. Same as **ang_deg**.
asin	Arc sine (the inverse function of sine).
asinh	Inverse hyperbolic sine.
atan	Arc tangent (the inverse function of a tan).
atanh	Inverse hyperbolic tan.
atan2	Two argument function. For non-0 x,y, the function returns the angle between the + x-axis and the given x,y coordinates.
avg	Returns the average of the values of the selected quantity.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
cang_deg	Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep variable). Returns a double precision value cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates cang_deg expressions in radians. See cang_deg_val.
cang_deg_val	Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values and suitable for use in Optimetrics, which works in SI values. . Returns simple numbers.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable). Returns a double precision value.
cmplx(*re*, *im*)	A complex number, where *re* is the real part and *im* is the imaginary part.
conjg	Conjugate of the complex number.
cos	Cosine.
cosh	Hyperbolic cosine.
crestfactor	Returns the crest factor (peak/RMS) for the selected quantity.
cum_integ	The cumulative integral function returns a set of values that have the same length as the original set of points (the first element will always be zero). Element *I* of the set returned by cum_integ is the integral of elements 1 through *I* of the original data set.
cum_sum	The cumulative sum function returns a data set that has the same length as the original set of points. Element *I* of the set returned by cum_sum is the sum of elements 1 through *I* of the original data set.
dB(x)	20*log10(|x|) to base 10.
dBc	Decibels relative to the carrier. It is the power ratio of the signal to a carrier signal. Gives the relative signal strength.
dBm(x)	10*log10(|x|) +30.
dBm	(for electric field quantities) is computed as: 20.0*log10(x) + 60.0
dBu	(for electric field quantities) is computed as: 20.0*log10(x) + 120.0
dBW(x)	10*log10(|x|).
dB10	10*log(|x|) to base 10.
dB10normalize	10*log [normalize(mag(x))].

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dB20	20*log(x) to base 10.
dB20normalize	20*log [normalize(mag(x))].
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero.
delaytime	Obtains the time from zero to 50% of the target point.
degel	Conversion from degrees electrical to seconds with respect to Hz.
deriv	Derivative of a given parameter.
distortion	Returns the total distortion for the selected simulation quantity and an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
even	Returns 1 if integer part of the number is even; returns 0 otherwise.
exp	Exponential function (the natural anti-logarithm) of the simulation quantity.
fmod	nReturns the double precision remainder of x/y.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.
fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.
if	if(cond_exp,true_exp, false_exp).
im	Imaginary part of the complex number.
int	Truncated integer function.
integ	Integral of the selected quantity. Uses trapezoidal area.
integabs	Absolute value of integral.
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	this function, you need to open the Add Trace Characteristics dialog and select the Error category.
j0	Bessel function of the first kind (0^{th} order).
j1	Bessel function of the first kind (1^{st} order).
jn	Bessel function of the first kind (nth order).
ln	Natural logarithm.
log	Natural logarithm (same as ln).
log10	Logarithm base 10.
lowercutoff	Returns the lower 3dB frequency of the selected simulation channel in Hertz.
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.
mag	Magnitude of the complex number.
max	Returns maximum value of the simulation quantity.
max_swp	Returns maximum value of a sweep.
max2	Maximum value of the two simulation quantities. For example, **max2 (a,b)** will plot maximum of **a** and **b** for a particular instance.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
min_swp	Returns the minimum value of a sweep.
min2	Minimum value of the two simulation quantities. For example, **min2 (a,b)** will plot minimum of **a** and **b** for a particular instance.
mod	Returns the double precision modulus (x - floor(x/y)*y). Not supported in solvers.
nint	Nearest integer.
none	Returns null value.
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x)).
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.
overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.
polar	Coverts the complex number in rectangular co-ordinates to polar co-

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	ordinates.
per	Returns the period of a simulation quantity.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180 degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
pk2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected simulation quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pmax	Maximum period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
pow	Raises x to the power of y; pow(x,y).
prms	Period Root Mean Square.
pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
pwl	Piecewise Linear.
pwl_periodic	Piecewise Linear for periodic extrapolation on x.
pwlx	Piecewise Linear x with linear extrapolation on x.
pw_minus	Pulse width of the first negative pulse.
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.
pw_minus_rms	RMS of the negative pulse width input stream.
pw_plus	Pulse width of the first positive pulse.
pw_plus_avg	Average of the positive pulse width input stream.
pw_plus_max	Max. Pulse width of the positive pulse of input stream.
pw_plus_min	Min. Pulse width of the positive pulse of input stream.
pw_plus_rms	RMS of the positive pulse width input stream.
re	Real part of the complex number.
rect	Converts the complex number in polar to rectangular co-ordinates.
rem	Returns the fractional part of a decimal number such that rem(x) = x-int (x) Syntax: rem(x)
ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
risetime	Obtains the time taken to go from 10% to 90% of target point.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
root	nth root function.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional args.
sgn	Sign extraction.
sin	Sine.
sinh	Hyperbolic sine.
slidingmean	Returns the moving average value of the selected simulation quantity (specified by the first argument). The average is calculated over a period (specified by the second argument).
slidingrms	Returns the moving RMS value of the selected simulation quantity (specified by the first argument). The RMS value is calculated over a period (specified by the second argument).
sqr	Square of the selected simulation quantity.
sqrt	Square root of the selected simulation quantity.
stddev	Returns the standard deviation of given values.
sum	Returns the sum of the given values.
tan	Tangent.
tanh	Hyperbolic tangent.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.
uppercutoff	Returns the upper 3dB frequency of the selected simulation channel in Hz.
variance	Calculates the variance of the given values.
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value.
XWidthAtYVal	Returns the X width between the first 2 occurrence of Y value.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
YAtXMax	Returns the X value at maximum value of Y.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.
y0	Bessel function of the second kind (0^{th} order).
y1	Bessel function of the second kind (1^{st} order).
yn	Bessel function of the second kind (nth order).

Selecting Solution Quantities to Plot

When you create a report, each trace in the report includes a quantity that is plotted along an axis. The quantity being plotted can be a value that was calculated by Mechanical, a value from a calculated expression, or an intrinsic variable value (such as frequency or theta).

> **Important:**
>
> Available solution quantity categories depend on the design type, solution type, setup parameters, and plot domain. The selected category provides the default name for the plot, such as "S Parameter Plot *n*". You can edit plot names in the project tree and plot header text in the Properties window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select solution quantities to plot:

1. Click **Results** > **Standard Report** and select the type of report you would like to generate.

   The **Report** window appears.

2. On the **Trace** tab, select a **Category** from the list.

Category	Description
Variables	Intrinsic variables (such as frequency or theta), or user-defined project variables (such as the length of a quarter-wave transformer).
Output Variables	User-defined expressions applied to derive quantities from the original field solution.
S-Parameter	S-parameters from the S-matrix. For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S_{11} and S_{21} represent the extracted reflection and transmission coefficients, respectively.
Y-Parameter	Admittance matrix parameters computed from the S-parameters and port impedances.
Z-Parameter	Impedance matrix parameters computed from the S-parameters and port impedances.
Group Delay	Quantity calculated as rate of change of the total phase shift with respect to angular frequency, $\frac{d(\Phi)}{d(\omega)}$
VSWR	Voltage standing wave ratio, calculated from the equation: $\frac{1+\lvert S_{ij}\rvert}{1-\lvert S_{ij}\rvert}$
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
Matched S Parameter	Matched S is the S-matrix produced by solver. It becomes useful to view when passive ports exist and the matched S-matrix is rectangular instead of square. When passive ports exist, S is the square matrix of the active ports and the passive ports represent a loss mechanism. Matched S-matrix provides coupling information to the passive ports. In Transient, we show Matched S entries while the simulation is in process which is not clean for regular S when port post processing exists.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Short Circuit Z	Short circuit impedance, calculated as: $\mathbf{ShortCircuitZ\,(P1)} = \mathbf{mag}\left(\frac{1}{\mathbf{Y(P1,\ P1)}}\right)$
Loop Inductance	LoopL computes with all other ports shorted, even ports on other nets. These short circuits permit induced currents to flow in nearby nets, meaning LoopL will always be lower than a more rigorous loop inductance computation. $\mathbf{LoopL\,(P1)} = \frac{\mathbf{im}\left(\frac{1}{\mathbf{Y(P1,\ P1)}}\right)}{(2 \times \pi \times \mathbf{freq})}$
Loop Resistance	Calculated as: $\mathbf{LoopR\,(P1)} = \mathbf{re}\left(\frac{1}{\mathbf{Y(P1,\ P1)}}\right)$
Capacitance	This capacitance calculation leaves other nets open-circuited, whereas a rigorous capacitance calculation would short them to ground, meaning "Capacitance" computed here will always be too low. $\mathbf{C\,(P1)} = \frac{\mathbf{im}\left(\frac{1}{\mathbf{Z(P1,\ P1)}}\right)}{(2 \times \pi \times \mathbf{freq})}$
Lambda	Guided wavelength.
Epsilon	Effective permittivity.
TDR Impedance	Time-Domain Reflectometry (TDR) impedance for non-terminal problems. The idea is to excite a structure with a step function, and inspect the reflections as a function of time. This quantity is available only when the **Time** domain is selected. Selecting the TDR Impedance category displays the TDRZ of every terminal or mode in the ports. The list of available Functions includes those that can operate on the TDRZ values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active S-Parameter	Active S parameters represent the reflection coefficient of one port when another port is excited. HFSS offers passive S-parameters as well as active S-Parameters. If you plot Active S-Parameters, HFSS includes the effects of mutual coupling with other ports or elements. One use is in Optimization, to view how S-parameters change as design parameters change. In this case you may plot Active S Parameters. In such cases they allow you to forecast design behavior in the presence of an ideal feed network. A 0w value is a matched load, but 1w values take that into account. Active S-parameters are given as a linear combination of the regular S-parameters, with some weights 'a', based on the matrix equation $\mathbf{b} = \mathbf{S} \times \mathbf{a}$. In the post-processor, the active S-parameter for port*n* is displayed to the user as ActiveS (P*n*:*m*) where "P*n*" indicates the port number and ":*m*" indicates the mode number. Active S, Y, Z, VSWR are supported only for driven modal projects. Power is proportional to the square of the fields. As a consequence, active S-parameters, which are field related, are proportional to the square root of what you set under Edit Sources. You can check this by setting all power to 0 except for one port of interest where you want to see the Active S-parameter. Then for a neighboring port, change the input power in steps, and monitor how the active S-parameter changes. Keep in mind that this involves complex numbers which complicates things. Given a driven model project with a total of *N*port modes, let a_k denote the complex excitation for the *k*-th mode specified in the Edit Sources dialog box. Also let S denote the computed *N* x *N*scattering matrix. If $a_m \neq 0$ define active-S_m, *m* = 1..., *N* by $$active\text{-}S_m \equiv \sum_{n=1}^{N} S_{mn} \frac{a_n}{a_m} \qquad m = 1, ..., N$$ If *S* is renormalized or deembedded, the interpretation is that the stimulations are applied to the renormalized external transmission lines at the plane of deembedding. The other relative active quantities are simply transformations on the active S_m.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active Y-Parameter	$Y_0^{(M)}$ is the port admittance of the *m*-th mode. $$active\text{-}Y_m \equiv Y_0{}^{(m)} \frac{1 - active\text{-}S_m}{1 + active\text{-}S_m} \qquad m = 1, ..., N$$
Active Z-Parameter	$Z_0^{(m)}$ is the impedance of the *m*-th mode. $$active\text{-}Z_m \equiv Z_0{}^{(m)} \frac{1 + active\text{-}S_m}{1 - active\text{-}S_m} \qquad m = 1, ..., N$$
Active VSWR	Active Voltage standing wave ratio is supported only for driven modal projects. $$active\text{-}VSWR_m \equiv \frac{1 + \lvert active\text{-}S_m \rvert}{1 - \lvert active\text{-}S_m \rvert} \qquad m = 1, ..., N$$
Passivity	A scalar quantity based on the matrix Q = I - conjugate (transpose(S)) * S For every frequency, the value must be no larger than 1. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.
Power	For Composite Excitation Solution types, selecting Power as Category lists Quantities for Accepted (Pacc), Reflected (Pref), and Incident (Pinc) powers/port based on your Edit Sources settings.
Design	Enables you to plot or tabulate properties of objects, such as their volumes.
Expression Cache	The values of expressions listed in the Expression cache of the Solution setup can be plotted, for example, as a function of adaptive pass to monitor their convergence.
Expression Convergence	This is intended to plot convergence, as a function of adaptive pass, of expressions in the Expression cache of the Solution setup. In defining the report, for "Context", select Solution: SetupN:Adaptive Pass. ExprDelta will show the change in the value of the expression as a function of adaptive pass, while ExprGoal will show, for comparison, the convergence goal for this expression, as defined in the Analysis Setup under the panel Expression Cache.

3. Select a **Quantity** from the list.
4. If desired, select a **Function**.
5. Click **Close**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was automatically calculated by Mechanical such as the magnitude of S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or phase.

To select a field quantity to plot:

1. When you create the report, specify the Report Type as "Fields" and the plot type (for example, radiation pattern.)
2. In the *Report* dialog box, select Geometry for the Context, unless you are plotting scalar (for example, integration).
3. In the *Report* dialog box, select one of the following categories. The selected category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User defined expressions applied to derive quantities from the original field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by Mechanical, as well as derived field quantities that are defined by calculated expressions you set up in the Fields Calculator.

4. Select a quantity to plot from the **Quantity** list. The available quantities will depend upon the selected category and the setup of the design. See Field Quantities list for definitions.

Plotting Imported Solution Data

1. In the **Solution** drop-down menu of the *Report* dialog box, click the imported data you want to plot.
2. Follow the procedure for creating a report.

Setting a Range Function

To apply a range function to the Y, Z, or Mag component of a trace:

1. Click **Range Function** in the *Report* dialog box.

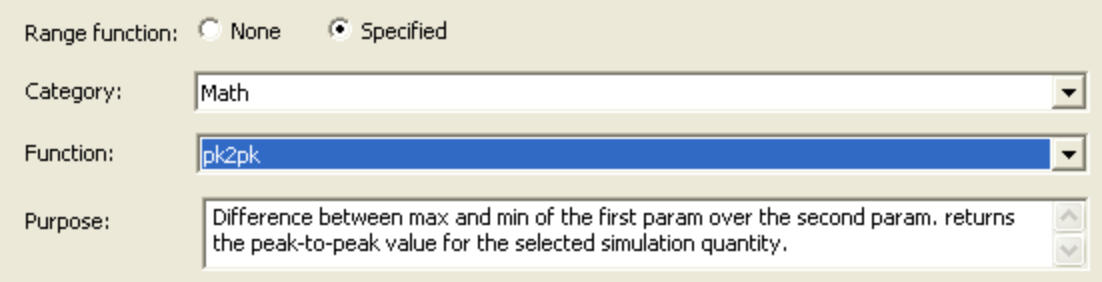
   This opens the *Set Range Function* dialog box. The functions available are the same as described in the Selecting a Function section, with the exception of those for the Eye Measurements category.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To enable Range function selection, click the Specified radio button.

   Selecting the None radio button disables the Range Function fields.

3. Select the **Category**, and then an associated Function to apply. The available categories depend on the plot, and Category enables the display of associated functions.



   Given a selected Function, and Category, the **Set Range Function** dialog displays a text field that explains the Purpose of the function. See figure above.

   Selecting a function causes the display of a description in the **Purpose** field. If the function requires a value (such as the XatYVal Math function or the pw_minus_max Pulse Width function), the following table the function field displays the name, editable value field, unit, and description.

4. Use the **Over Sweep** drop-down menu to select from available sweeps.
5. To select from available Sweeps, or to edit them, use the ellipsis [**...**] button and uncheck **Use All Sweeps**.

   This enables a list of the sweeps. The sweep(s) you select is displayed on the Over Sweep line. You can use the buttons to **Clear All Selections** or **Select All** sweeps.

6. Select the Sweeps **Default** or **Edited** radio buttons to specify whether to accept the default or edited sweeps.
7. To edit the sweeps further, select the ellipsis button to display an *Edit Sweep* dialog box.

   For frequency variables, this lets you specify a single value, linear step, linear count, decade count, octave count, or exponential count. You can **Add** legal values to the list of sweep values, **Update** the list for changes, or **Delete** selected entries.

8. Click **OK** to apply the range function.

Range Functions

The following table shows the **Functions** according to their **Categories**. The most commonly used range categories are **Math** and **Radiation**. Other functions could be used if needed. Use the category links to navigate to tables with definitions of functions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Functions
Math	max, min, pk2pk, rms, sum, mean, variance, stddev, integabs, avgabs, rmsAC, ripple, pkavg, XatYMin, XatYMax, YAtXMin, YAtXMax, XAtYVal, YAtXVal, XWidthAtYVal
PulseWidth	pulsemin, pulsemax, pulsemintime, pulsemaxtime, pulsefall9010, pulsefront1090, pulsefront3090, pulsetail50, pulsewidth5050, pw_plus, pw_minus, pw_plus_avg, pw_minus_avg, pw_plus_max, pw_minus_max, pw_plus_min, pw_minus_min, pw_plus_rms, pw_minus_rms
Overshoot, Undershoot	overshoot, undershoot.
TR & DC	crestfactor, formfactor, distortion, fundamentalmag, delaytime, risetime, deadtime, settlingtime
Error	iae, ise, itae, itse
Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq.
Radiation	xdb10bandwidth, xdb20bandwidth, lSidelobeX, lSidelobeY, rSidelobeX, rSidelobeY

Math Functions	
***avg**	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
integabs	Absolute value of integral.
max	Returns maximum value of the simulation quantity.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pkp2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected simulation quantity.
sum	Returns the sum of the given values.
stddev	Returns the standard deviation of given values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Math Functions	
variance	Calculates the variance of the given values.
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value.
XWidthAtYVal	Returns the X width between the first 2 occurrences of Y value.
YAtXMax	Returns the X value at maximum value of Y.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.

Radiation Functions	
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default) To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.

Pulse Width Functions

> **Note:**
>
> In this table, the functions with the asterisk (*) do not appear on the Range Function drop-down menu. They can still be used via text entry.

pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
***pwl**	Piecewise Linear.
***pwl_periodic**	Piecewise Linear for periodic extrapolation on x.
***pwlx**	Piecewise Linear x with linear extrapolation on x.
pw_minus	Pulse width of the first negative pulse.
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.
pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.
pw_minus_rms	Returns the rms of the negative pulse width input stream.
pw_plus	Returns the pulse width of the first postive pulse.
pw_plus_avg	Returns the average of the positive pulse width input stream.
pw_plus_max	Returns the maximum pulse width of the positive pulse of input stream.
pw_plus_min	Returns the minimum pulse width of the positive pulse of input stream.
pw_plus_rms	Returns the rms of the positive pulse width input stream.

Overshoot/Undershoot	
Overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
Undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TR & DC Functions	
crestfactor	Returns the crest factor (peak/RMS) for the selected simulation quantity.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.
distortion	Returns the total distortion for the selected simulation quantity and an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
delaytime	Obtains the time from zero to 50% of the target point.
risetime	Obtains the time taken to go from 10% to 90% of target point.
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional arguments.

Error Functions	
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.

Periodic Functions	
per	Returns the period of a simulation quantity.
pmax	Max period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
prms	Period Root Mean Square.

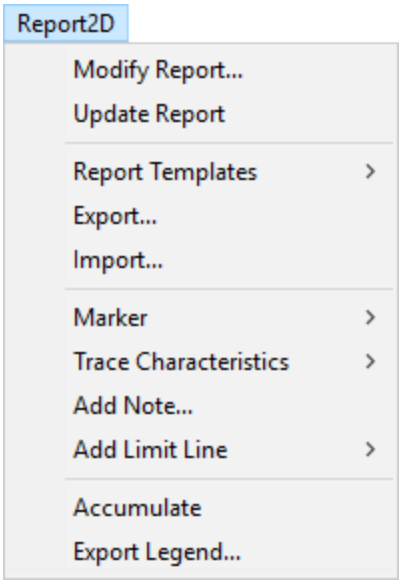
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC Functions	
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180 degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
lowercutoff	Returns the lower 3dB frequency of the selected simulation channel in Hz.
uppercutoff	Returns the upper 3dB frequency of the selected simulation channel in Hz.
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.

Importing Plot Data

When a report is open in Ansys Electronics Desktop, the **Report2D** menu appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> You *must* have a report open in order to import plot data.

The **Report2D**> **Import Data** command lets you import plot data from comma delimited files (*.csv), tab delimited files (*.tab), Ansoft PlotData files (*.dat), post-processor files (*.txt), or Ansoft ReportData files (*.rdat).

1. From the top menu bar, click **Report2D** > **Import**.

   The **Open** window appears.

2. Browse for and select the desired file.
3. Click **Open** to import the file into the current report.

   The imported traces appear in the Project tree under the current report.

   > **Note:**
   >
   > For a report trace where the primary sweep is NOT the same as the x-component, an export report then import report may not produce the same curve. In this circumstance, only *.rdat import the same curve/trace. For other formats, the import produces two separated traces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

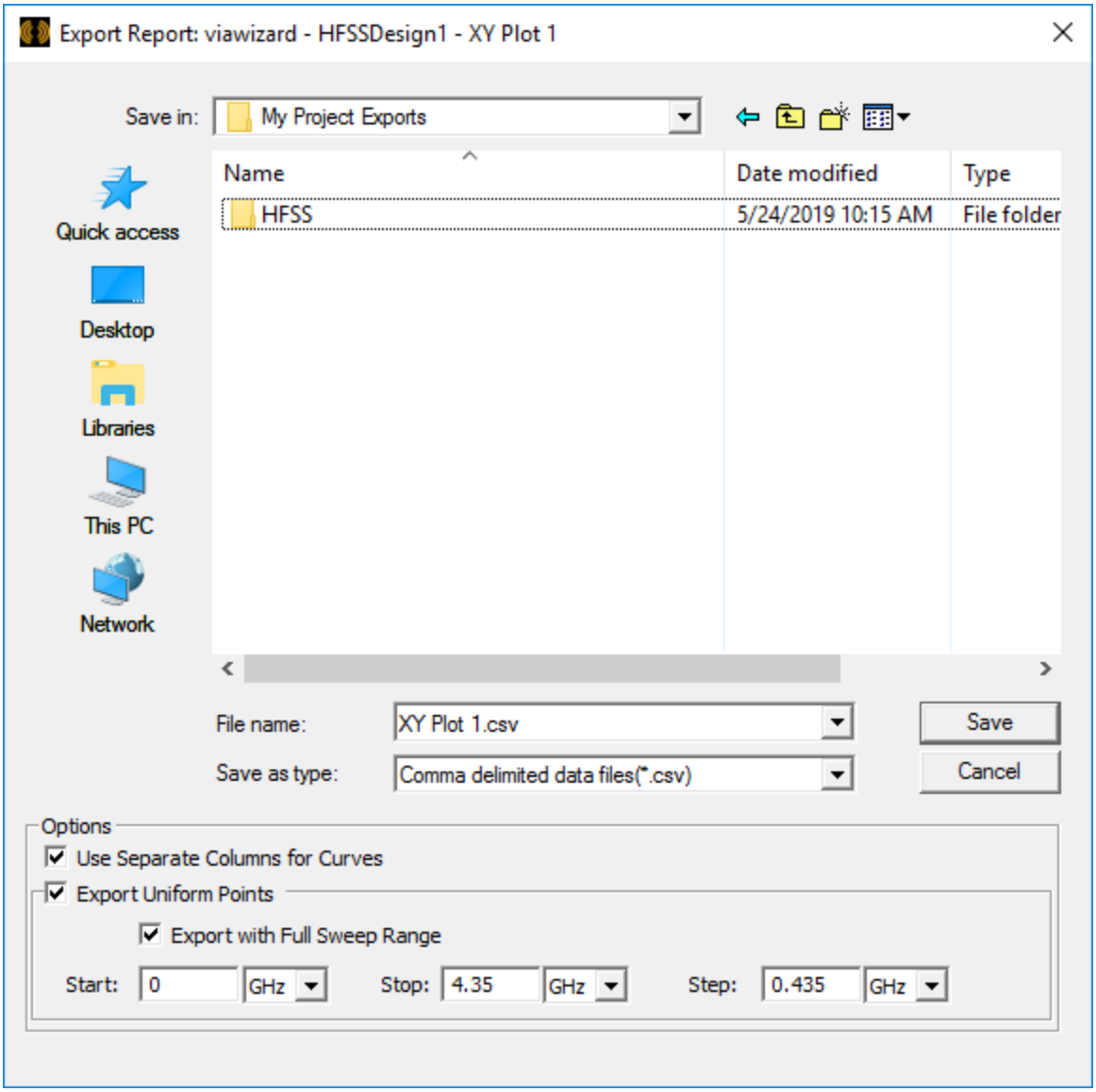
Exporting Reports

You can export reports in a variety of text and graphic formats. You must have an existing plot open to see the corresponding **Report2D** or **Report3D** menu.

1. Click **Report2D**> **Export** or **Report3D**> **Export**.

   Alternatively, right-click the plot and select **Export**.

   The *Export Report* dialog box appears.



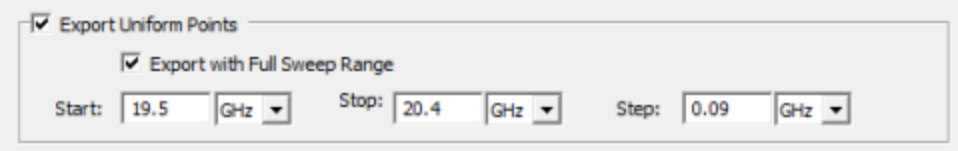
2. Use the file browser to find the directory where you want to save the file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the name of the file in the **File name** field.
4. Select one of the following file formats from the **Save as type** drop-down menu:

Extension	Contents
.csv	Comma-delimited data file
.tab	Tab-separated file
.dat	Ansoft plot data file
.txt	Post processor format file
.rdat	Ansoft report data file
.emf	Microsoft EMF files
.gif	GIF files
.bmp	BMP files
.wrl	VRML files
.tif, **.tiff**	TIFF files
.jpg, **.jpeg**	JPEG files

   - When exporting to *.csv or .tab files for an XY Plot, Eye Diagram or Data Table, a **Separate Columns for Curves** check box will be shown. Otherwise the check box will be hidden.
   - When it is checked the data is exported with one column per curve, and variable values appear in the column's header cell.
   - When it is unchecked the data is exported with one column per a trace, each variable has its own column.
   - When the design has many variables, uncheck the check box may keep the column header from becoming too long and hard to read.
   - When trace has more curves than number of sweep values, uncheck the check box to prevent the number of columns from being larger than the number of rows.
   - When a report only has one trace or all trace have same variations, this option is unchecked by default. Otherwise, this feature is checked.
   - For 2D reports, you can select **Export Uniform Points** and specify a full sweep range by editing start, stop, and step values.



5. Click **Save**. The report is exported to the specified location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For a report trace where the primary sweep is NOT the same as the x-component, exporting and then importing a report may not produce the same curve. In this circumstance, only *.rdat format imports the same curve/trace. For other formats, the import produces two separated traces.

Deleting Reports

To use **Delete All Reports**:

1. Click **Mechanical**> **Results**> **Delete All Reports.** You can also right-click on the Results folder in the Project Manager, to display the shortcut menu, and click **Delete All Reports.**

   All items under the Results folder in the Project Manager are removed.

To use **Delete** for a selected report:

1. Right-click a report icon in the Project Manager to display the shortcut menu.
2. Click **Delete** to remove this report.

Warning:

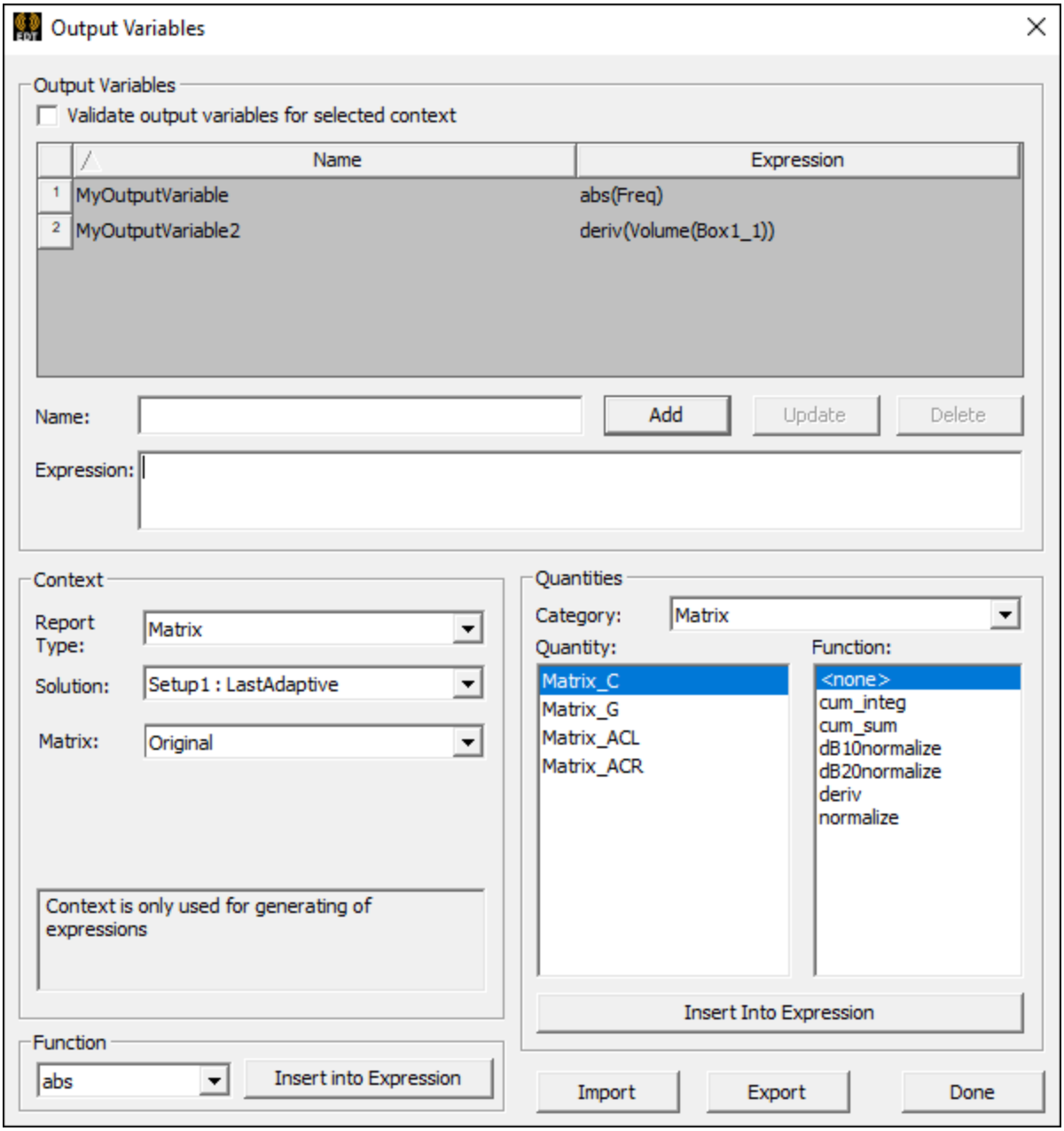
Solution data that has been deleted cannot be recovered!

Specifying Output Variables

Access the **Output Variables** window one of several ways:

- Click **Mechanical** > **Results** > **Output Variables**.
- In the Project Manager, right-click **Results** and select **Output Variables** from the shortcut menu.
- Click the **Output Variables** button in the **Report** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Output Variables** window contains four sections:

- **Output Variables** – contains a list of existing output variables, and allows you to specify the name and expression for a new output variable. The **Validate output variable...** check box allows you to validate an expression before adding it. Valid expressions appear in blue and invalid expressions appear in red.
- **Context** – specifies the Report Type and Solution. Depending on the type of report selected, additional fields may appear. All selections affect the functions and quantities

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

listed.

> **Note:**
>
> The context is used only for generating expressions.

- **Quantities** – allows you to insert quantities into the Expression area of the **Output Variables** section.
- **Function** – allows you to insert completed expressions into the Expression area of the **Output Variables** section.

To add an output variable:

1. Enter a **Name** for the variable.
2. Specify the **Context**. This impacts the available quantities and functions.
3. Specify the **Quantities** (required) and **Functions** (optional) using the **Insert Into Expression** buttons.
4. Review the **Expression**. When it is ready, click **Add**.

   The new output variable appears in the **Output Variables** list.

To delete an output variable:

1. Remove all references to the output variable from the project.
2. Save the project to erase the command history.
3. Open the **Output Variables** window using one of the methods above.
4. Select the variable from the **Output Variables** list and click **Delete**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. For example, if you want to plot the polar angle of a complex simulation result S_{11}, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity, but in degree and radian units respectively.

When using non-SI units in expressions, surprising outcomes might result. For example, the expression 1+ang_deg(S_{11}) represents an angle and the number 1 is treated as 1 rad. The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat 1 as degrees, make it explicit and use 1deg + ang_deg(S_{11}) instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. For example, if the complex S_{11} lies on the positive Y axis, ang_deg_val(S_{11}) would be 90 and 1 + ang_deg_val(S_{11}) would be 91.

Function List for Output Variables

The **Output Variables** window includes a second function list containing functions to enter directly into the Expression field. These functions can also be applied to previously specified Quantities and Functions.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the functions in the **Output Variables** dialog box's **Function** list or type them directly into the Expression field, if necessary.

Plotting Field Overlays in Mechanical

Field overlays are representations of basic or derived field quantities on surfaces or objects for the current design variation. You must have selected **Save Fields** in the solution setup in order to enable the **Plot Fields** selections. An exception to this rule is for *Transient Thermal* solutions, where the first and last time steps' field results are always saved. In this case, selecting the **Save Fields** option is necessary only when you wish to plot intermediate time step field results.

You must select applicable geometry to create a plot.

You have more control over the display via Modify Plot Attributes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Much of the content within this branch of the help is general in nature and applicable to multiple design types (HFSS, Q3D, Icepak, and so on). For **Mechanical** designs, the types of reports and overlays that you can generate are dependent on the Solution Type (*Modal, Structural,* or *Thermal*). Mechanical-specific field overlays content is covered within the *Modal Solutions, Structural Solutions,* and *Thermal Solutions* branches of the Mechanical help (sink links below). Use the following links to access the Mechanical-specific filed overlays content of interest:

Modal Solutions:

- Results (Modal)
- Field Overlay and Animation

Thermal Solutions: (Steady-State or Transient)

- Results (Thermal)
- Field Overlay and Animation
- Creating a Fields Summary

Structural Solutions:

- Results (Structural)
- Field Overlay and Animation
- Creating a Fields Summary

Modifying Field Plots

1. Click **Mechanical**> **Fields**> **Modify Plots** , or in the Project Manager, select the **Field Overlays** icon, right-click, and select **Modify Plots**, or press the **M** hotkey.

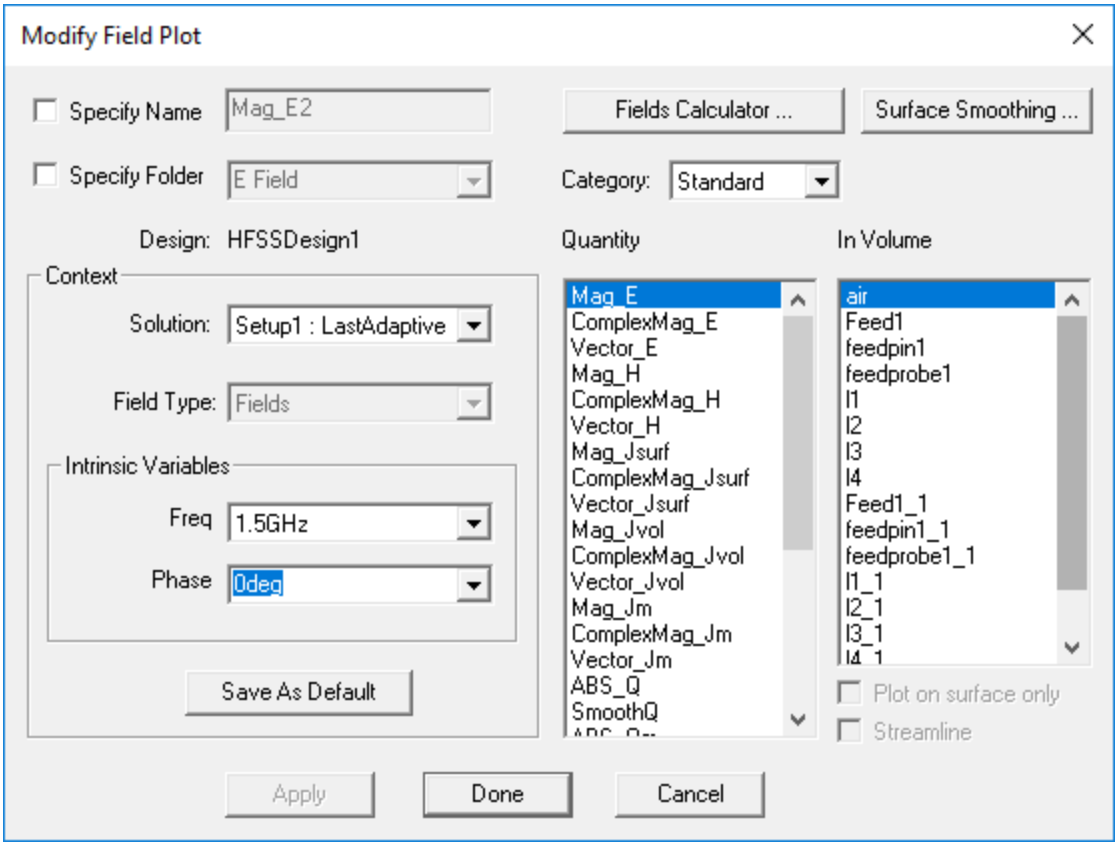
   The **Select Field Plot(s)** dialog box appears.

   Optionally, you can right-click an existing plot listed under the Field Overlays in the Project tree, bypass the **Select Field Plot** dialog box and go to step 3.

2. Select the plot you want to modify in the **Select** column, and then click **OK**.

   The **Modify Field Plot** dialog box opens.

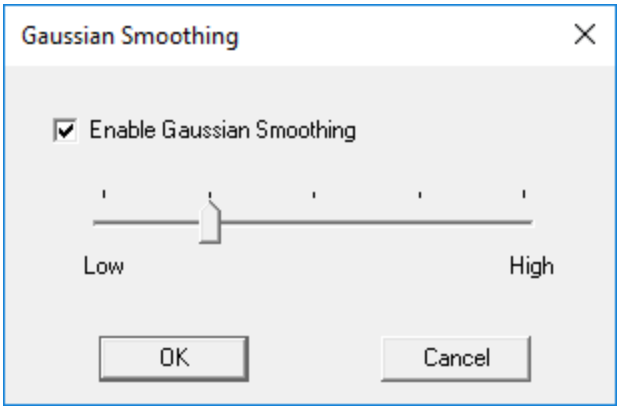
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Optionally, select the **Specify Name** check box to enable the name field.
- Optionally, select the **Specify Folder** check box to enable the drop-down menu.
- Optionally, select a different **Solution** from the drop-down menu.
- Optionally, select a different **Field Type**, if available.

3. Under **Intrinsic Variables**, specify the frequency and phase at which the field quantity will be evaluated.
4. Optionally, select a different field quantity to plot from the **Quantity** list.
   - To choose a calculated expression, select **Calculator** from the **Category** drop-down menu. If you choose Calculator, click the Fields Calculator button to display the calculator.
   - To choose a predefined field quantity, select **Standard** from the **Category** drop-down menu. Select from the Quantity list.
   - For scalar quantities plotted on the surface of a geometry, the **Surface Smoothing** button is enabled. For vector quantities, the **Surface Smoothing** button is disabled. Surface Smoothing applies a selected level of smoothing based on the Gaussian Blur algorithm. Clicking Surface Smoothing opens the Gaussian Smoothing dialog box.

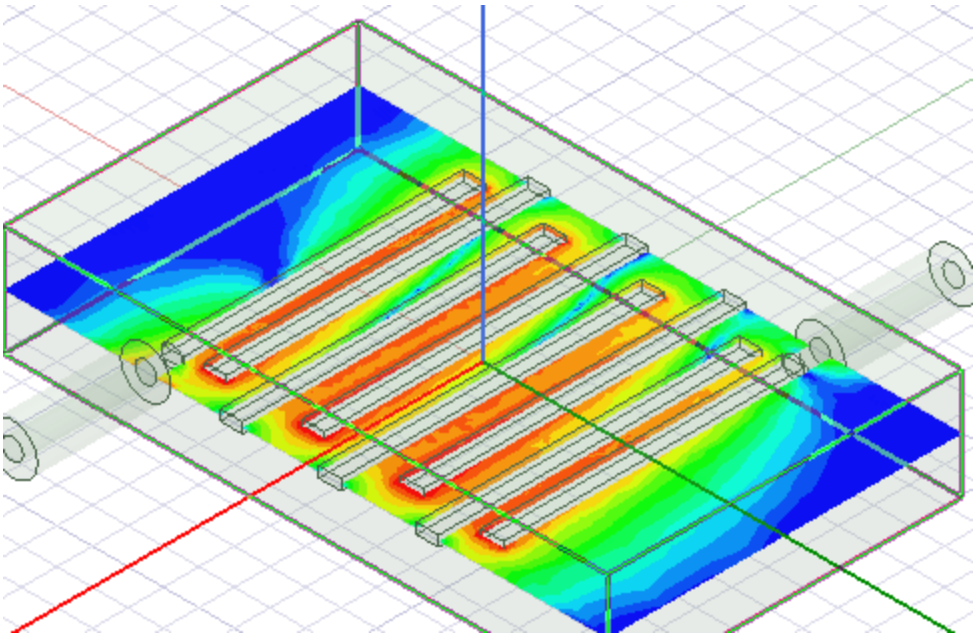
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the check box to enable the feature and use the slider to set the degree of smoothing.

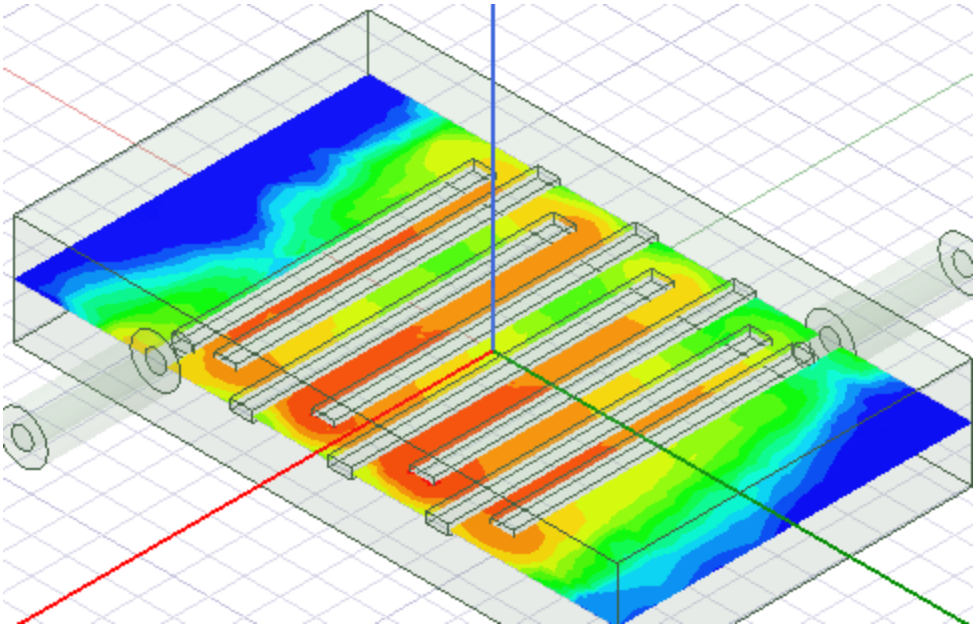
Important: You *must* click **Apply** in the *Modify Field Plots* dialog box to see the changes.

For example, this is a Mag_E fringe plot before applying the smoothing:



This is a Mag_E fringe plot after applying High smoothing:

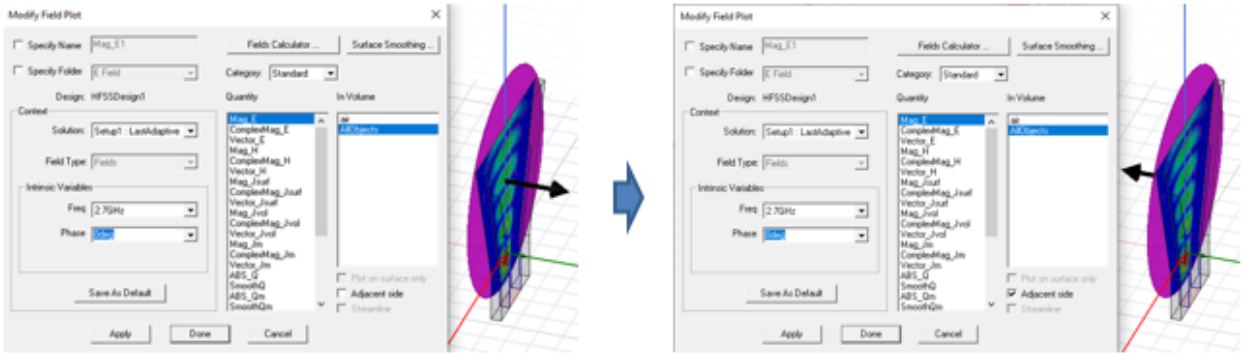
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.

6. If you are modifying a plot on a sheet object, the **Adjacent** check box allows you to generate field overlays on a sheet object using data from either side of the sheet. An arrow indicates where the data is coming from. While the data is usually the same on the two sides of a sheet object, in a situation where the sheet object is assigned to be a shell element, the field may be different on its two sides. (Shell elements are an option for Layered Impedance Boundaries and Finite Conductivity Boundaries with DC conductivity and two-sided selected.) In such cases, you can visualize the field data on the side you want.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **Apply** to make the changes and leave the dialog box open, or click **Done** to apply the changes and close the dialog box.

   The field quantity is plotted on the surfaces or within the objects you selected. The modified plot is listed in the specified plot folder in the project tree.

   The plot uses the attributes specified in the **Plot Attributes** dialog box. For details see Setting Field Plot Attributes.

Modifying Markers on Point Plots

For scalar point plots, a marker is used to represent a field quantity at a selected point (for vector point plots, arrows are used).

Modify the shape and size of markers in the plot attributes window:

1. Click **Mechanical**> **Fields**> **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Marker/Arrow** tab in the plot attributes window.
4. Under **Marker options**, select one of the marker types to represent the field quantity at the point:
   - **Sphere**
   - **Box**
   - **Tetrahedron**
   - **Octahedron**
5. Use the **Size** slider to increase or decrease the size of the marker.
6. Select **Map size** to scale the size of the marker to the magnitude of the quantity being plotted.
7. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

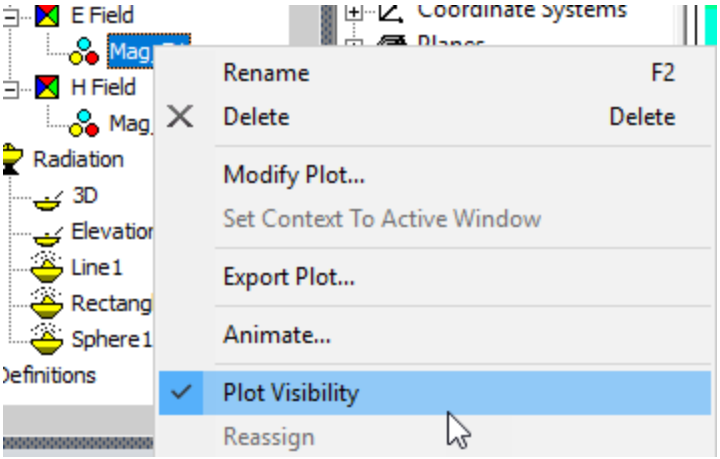
8. Click **Close**.

Setting a Plot's Visibility

You can control the visibility of Field Overlays by means of the **View> Visibility> Active View Visibility** dialog box, **Fields Reporter** tab (shown below) or by right-clicking on the plot in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

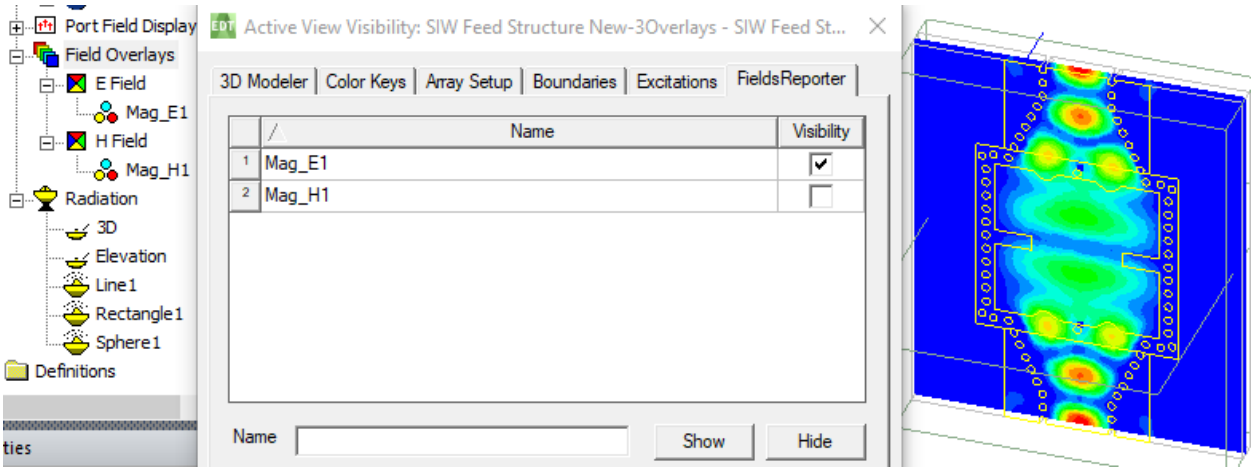
Project Manager and checking or unchecking **Plot Visibility**.



To display or hide a field overlay or mesh plot from view in the *3D Modeler* window:

1. Click **View**> **Visibility**> **Active View Visibility**. Alternatively, you can select the **Active View Visibility** icon from the **View, Draw,** or **Model** ribbon tabs.

   The *Active View Visibility* dialog box appears.

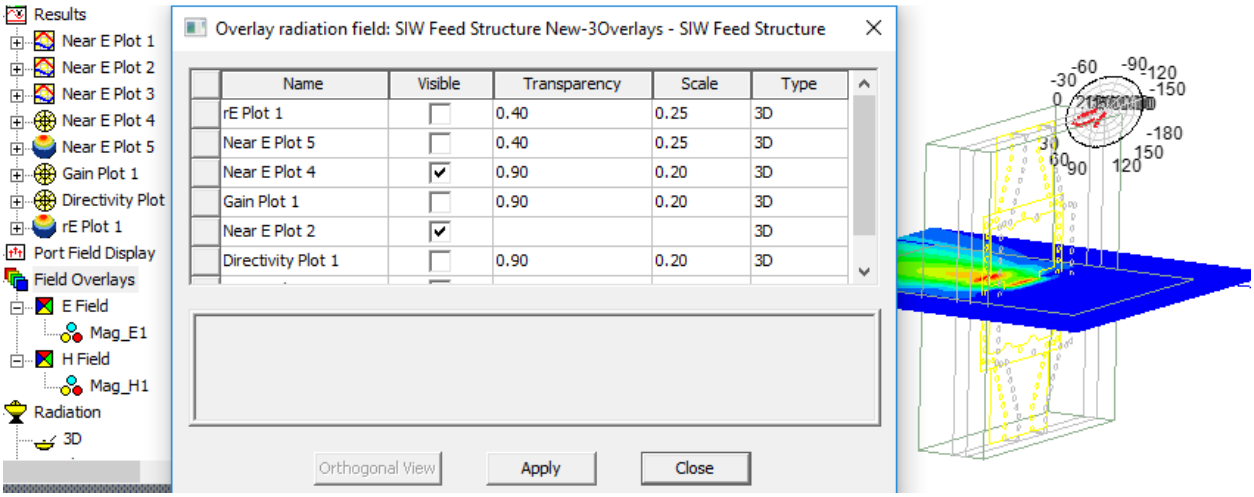


2. Click the **FieldsReporter** tab.
3. In the **Visibility** column, select the field overlay or mesh plots you want to display. Clear the plots you want to hide from view.

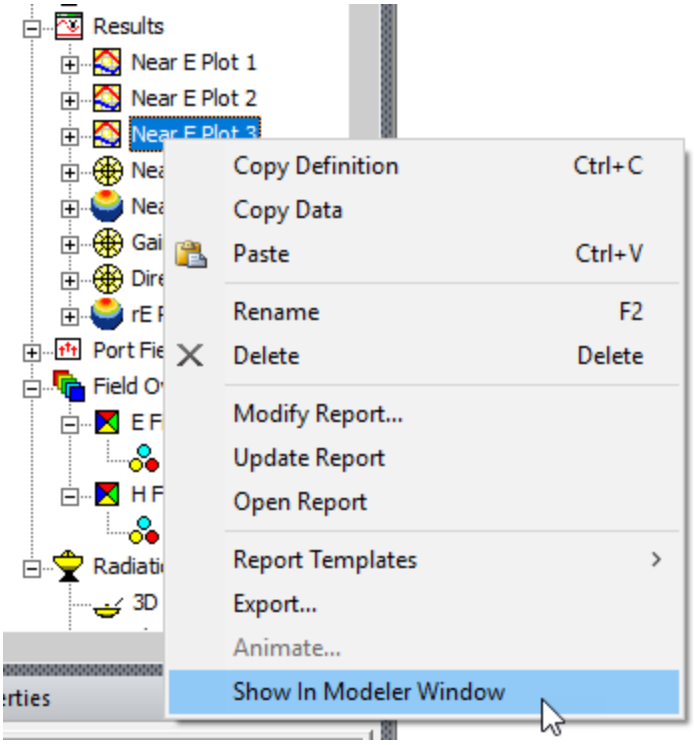
   Only the selected plots will be visible. Multiple plots can be shown at once.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation plots and antenna parameter tables available for overlay can be viewed by right-clicking **Field Overlays** in the *History Tree*, and selecting **Plot Fields> Radiation Field...**. This displays an **Overlay radiation field** dialog that lists each Results plot available for overlay.



You can also control the visibility of **Results** plots available for **Overlay radiation field** by right-clicking on the report and selecting **Show in Modeler Window**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Key Settings

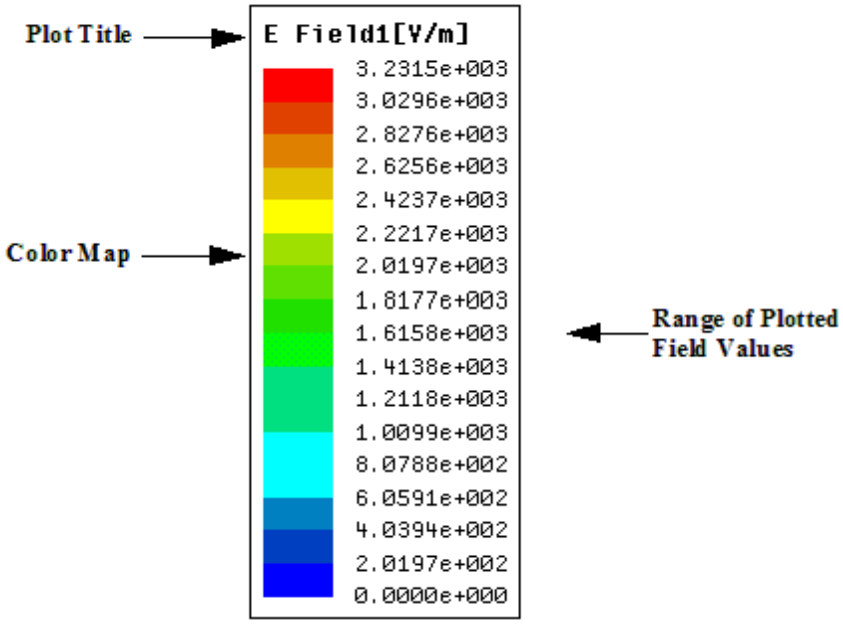
The color key (shown below) displays the range of plotted field values for a field overlay plot. It displays the colors that correspond to the range of field values on the plot. You can click and drag the color key to move it. In the case of no field data, the color key does not appear.

1. Click **View**> **Visibility**> **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Color Keys** tab.
3. In the **Visibility** column, select the field overlay or mesh plots in which you want to display the color key. Clear the plots in which you want to hide the color key from view.
4. Click **Done** to dismiss the dialog box.

Alternatively, to hide the color key, right-click on the color key in the active view window and choose **Hide** from the shortcut menu.



Color keys will only be visible for the plots that are selected in the **Active View Visibility** dialog box. If a hidden color key is shared by two or more overlay plots (such as a Heat Flux key, which is shared by heat flux magnitude and heat flux vector plots), then the color key will be hidden in all plots that share it. The **Active View Visibility** dialog box only lists a single incidence of a particular key, regardless of how many overlay plots share it. Use this dialog box to restore visibility of hidden color keys.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Field Overlay Attributes

After creating a mesh or field overlay on a surface or volume, you can modify its appearance by changing the settings in the **Plot Attributes** dialog box. You will modify the settings for a plot folder and all plots in that folder will use the same attributes.

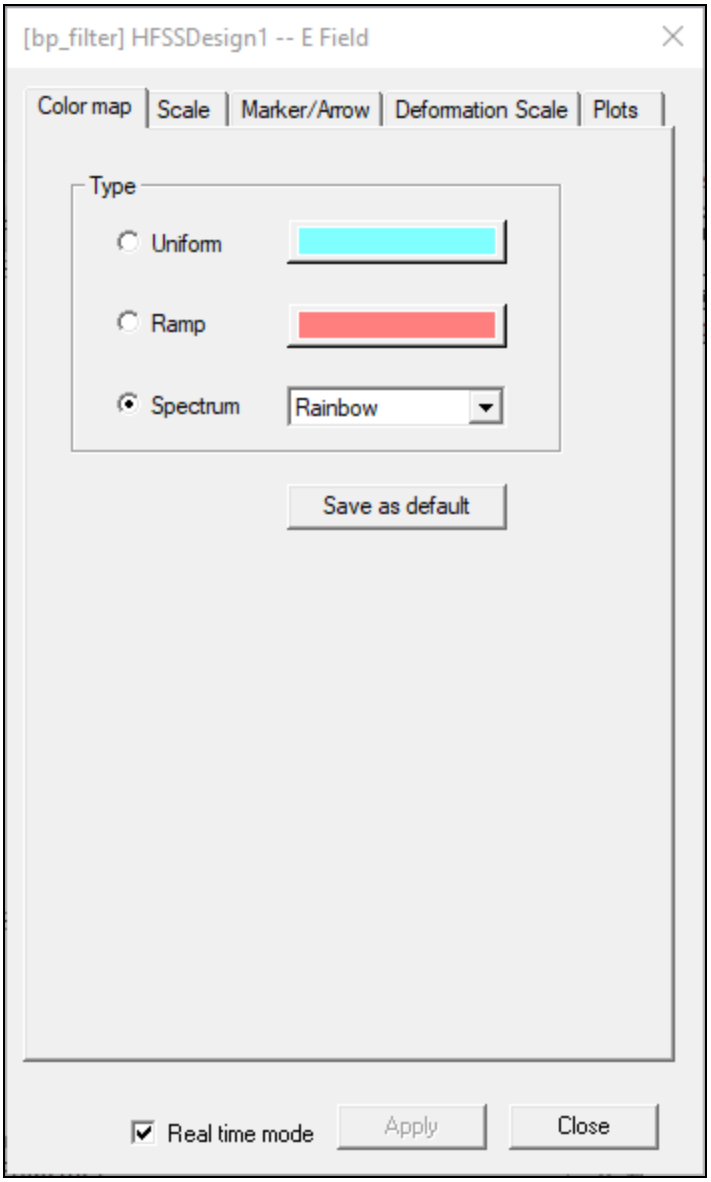
1. Click **Mechanical** > **Fields** > **Modify Plot Attributes**. Alternatively, right-click **Field Overlays** in the Project Manager and select **Modify Attributes** from the shortcut menu.

   The **Select Plot Folder** dialog box appears.

2. In the **Select Plot Folder** dialog box, select the plot type that you want to modify and click **OK**. You can also right-click the specific plot in the Project Manager and choose **Modify Attributes** from the shortcut menu.

   A dialog box with attribute settings for the selected plot type (whether a field result or a mesh plot) appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. For field result overlays (not mesh plots), the available plot attributes under each tab in the dialog box are listed in the following table:
    - **Color map:**

      The number of colors used and how they are displayed. The field data must be available for the color key to appear.

    - **Scale tab:**

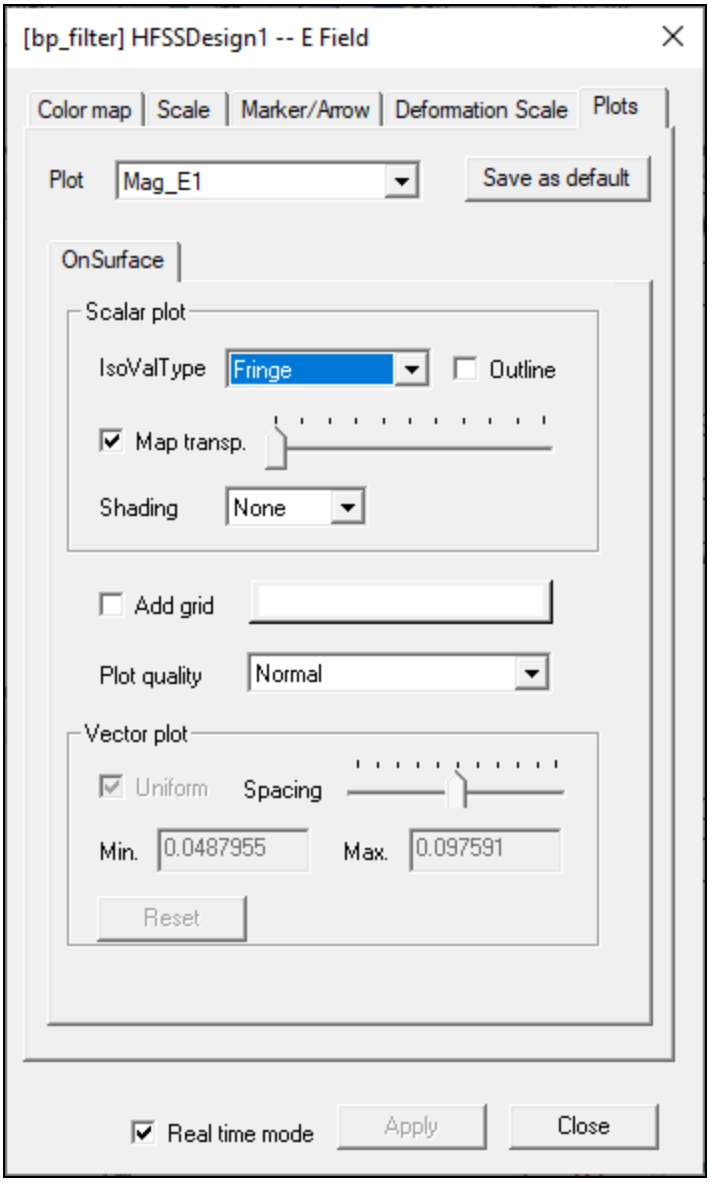
      The scale of field quantities, including the number of divisions in the scale, whether to use dB as the units, whether to use a linear or log scale, auto schall options, and plot number format.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Marker/Arrow**
  - The appearance of points (for scalar point plots).
  - The appearance of arrows (for vector plots).
  - Magnitude filtering (for vector plots). That is, you uncheck Map size, and specify a Min and Max Magnitude, or use a slider to set the Min threshold.
- **Deformation Scale:**

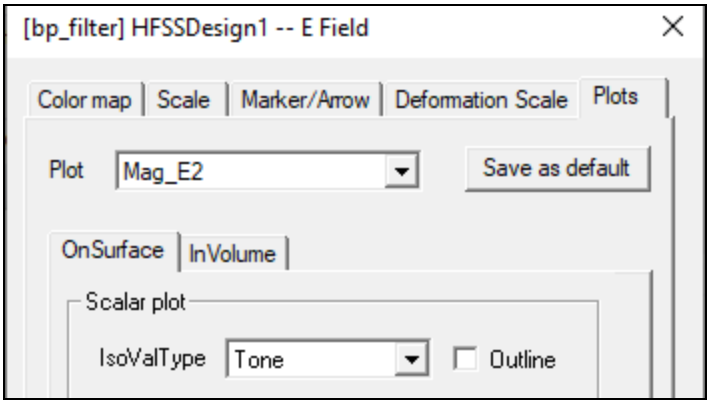
  This is for use with plots that include Stress feedback from Ansys Workbench Integration.
- **Plots (if not vector or streamline):**
  - The **Plot** selected. If multiple plots are available, you can select from a drop down menu. Plots can be OnSurface or InVolume. If both types are available, the dialog includes tabs for each.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



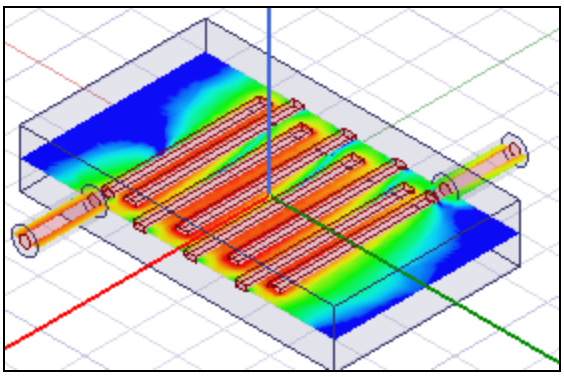
- Plots can be **OnSurface** or **InVolume**. If both types are available, the dialog includes tabs for each.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



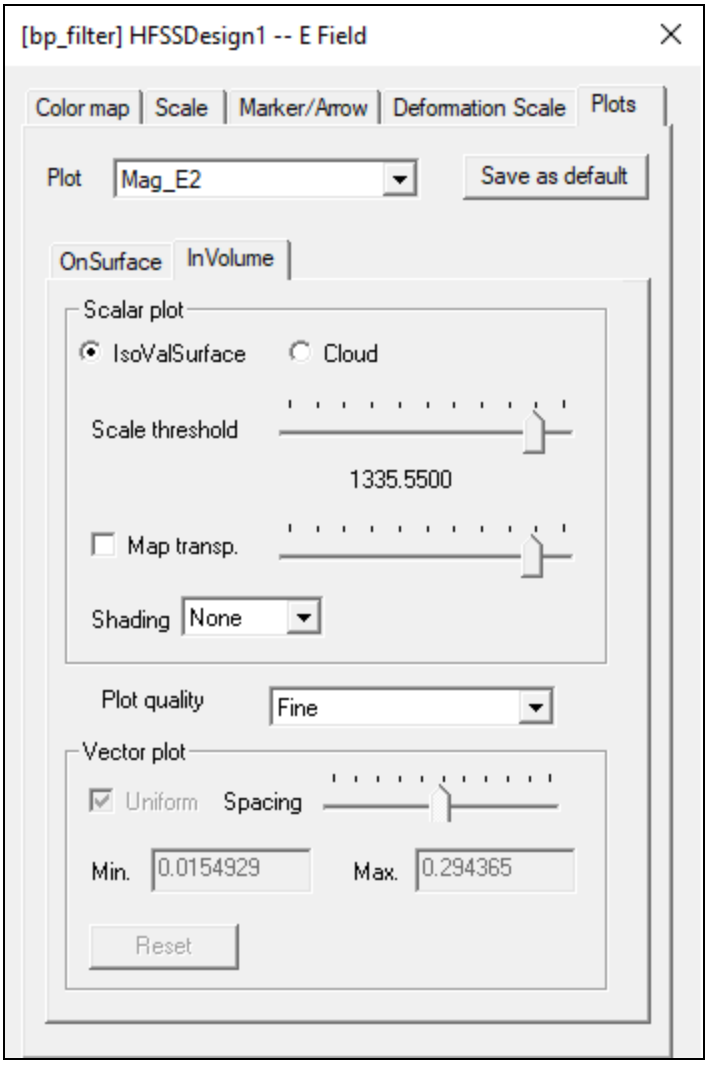
- The type of isovalue display (for scalar plots.) For *Line, Fringe* and *Tone* **IsoValType**, *Outine* is enabled. For *Gourad*, it is disabled. The *IsoValue* display selection also affects *InVolume*, which also has other options discussed below.
- Whether to use Shading (for scalar plots), if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting."
- The transparency based on solution value.
- Whether to add a grid (that is, a mesh overlay), and to set the grid color.
- Specify the plot resolution as **Coarse**, **Normal**, **Fine**, or **Very Fine**.

  This affects the use of memory for animating plots. For large plots with more frames to animate, use **Coarse** or **Normal** to reduce memory requirements and improve performance. For smaller plots with few frames, if higher resolution is required, use **Fine** or **Very Fine**.



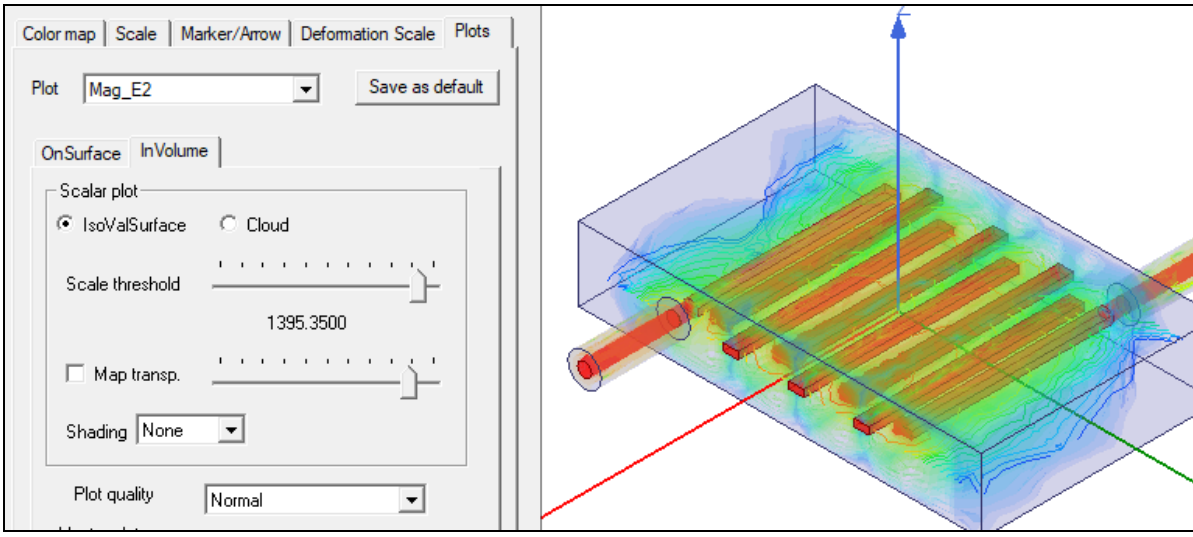
If an InVolume plot is available, and you select the tab, additional display options are available to allow visibility of internal layers and objects, particularly those with high field values. For *IsoValSurface* for an *InVolume* Scalar plot, the choices are for **Scale threshold** and **Map transp**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

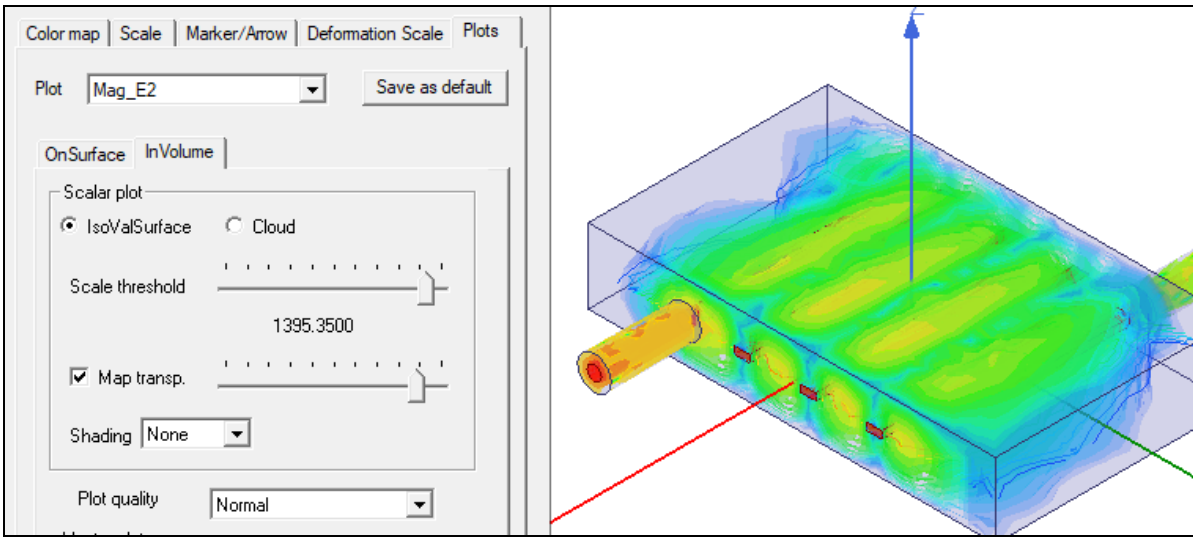


- The "**Scale threshold**" slider bar allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



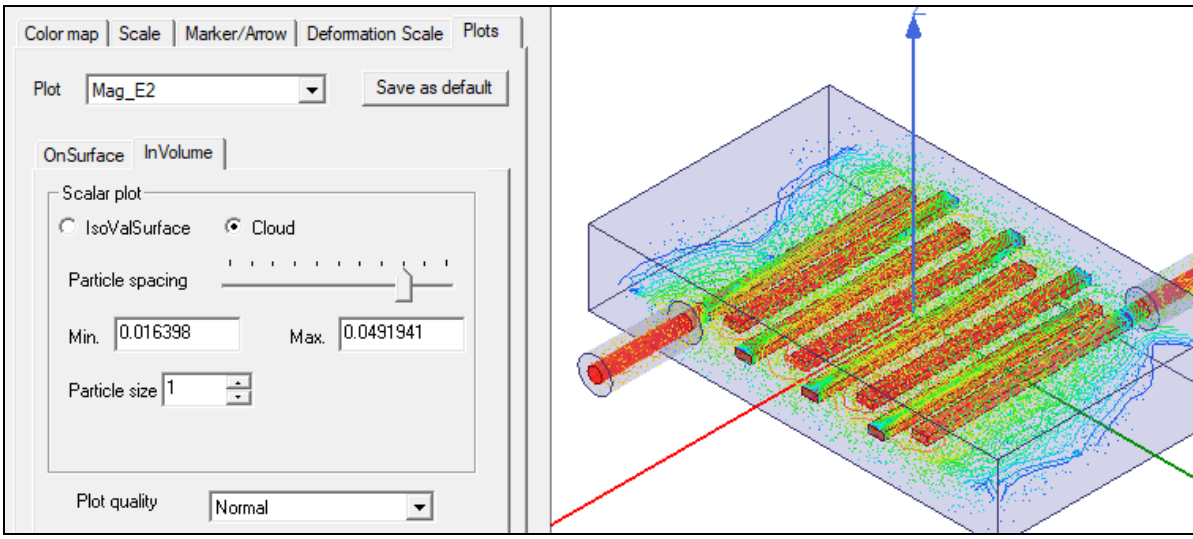
- Iso surfaces with field value higher than the threshold will show as opaque. The range of the "**Scale threshold**" is the same as the color key range from *Min* to *Max*.
- When "**Map transp.**" is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When the **Map transp.** slider bar is at the left most position, transparency is 0. This feature is essentially disabled, and the "**Scale threshold**" slider bar will be disabled. Also, high fidelity transparency should be enabled for best results.

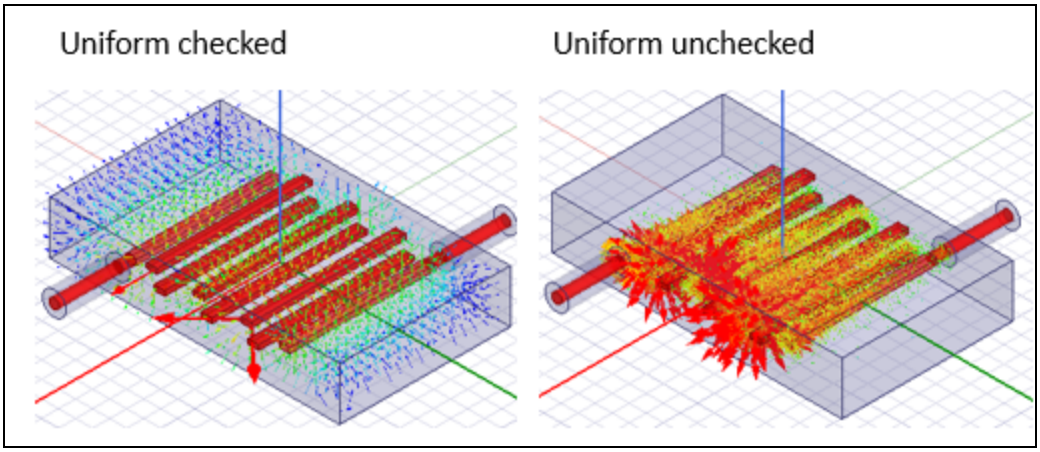
- For *Cloud* plots, field values are represented by points that illustrate the spatial distribution of the solution. You can use the **Particle spacing** slider bar to adjust the plot. The higher the solution value, the greater the cloud density. You can also specify **Min** and **Max** values (numeric values based on the active model length unit) and adjust **Particle size**.



- **Plots (for vector plots):**
  - The **Plot** selected.
  - The **Min** and **Max** spacing of arrows (for vector plots). These values are based on the model's active length unit and are only relevant when the **Uniform** option is selected.
  - **Plot quality**, as *Normal, Coarse, Fine,* or *Very Fine*.
  - **Uniform** vector spacing, based on the *Spacing* slider position and the *Min* and *Max* values.

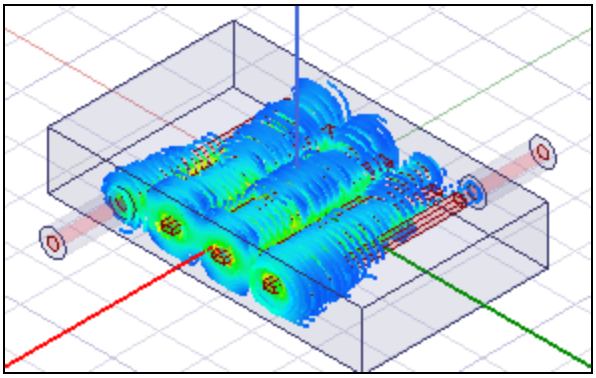
    When Uniform is *not* selected, a vector is rendered at every node (both corner and mid-side nodes of the elements).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Reset** button resets a good initial spacing (and thus Min, Max spacing range) without the need to recreate the field plot.

- **Plots (if streamline is checked):**
  - The **Plot** selected
  - The **Line style** as *solid* or *cylinder* from drop-down menu.
  - **Line width**, specified using a slider.
  - Whether to show *Marker* on *Streamline*.
  - **Seeds density** spacing. This affects the number of stream lines used to represent the quantity in the plot. Moving the slider to the left decreases the spacing and increases the number of stream lines. Moving the slider to the right increases the spacing and decreases the number of lines used to represent the quantity.
  - **Min**. and **Max**. values represented (in *Magnitude filtering* settings).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Under each tab, click **Save as default** if you want the tab's settings to apply to field overlay plots created after this point.
b. Select **Real time mode** if you want the changes to take effect immediately in the view window.
c. If this option is cleared, click **Apply** when you want to see the changes.

Modifying the Field Plot Scale

To change how field quantities are scaled on the field overlay plot:

1. Click **Mechanical** > **Fields** > **Modify Plot Attributes** or, in the Project Manager, right-click Field Overlays and select **Modify Plots** from the shortcut menu (or use the "**m**" hotkey).

   The **Select Plot Folder** dialog box appears.

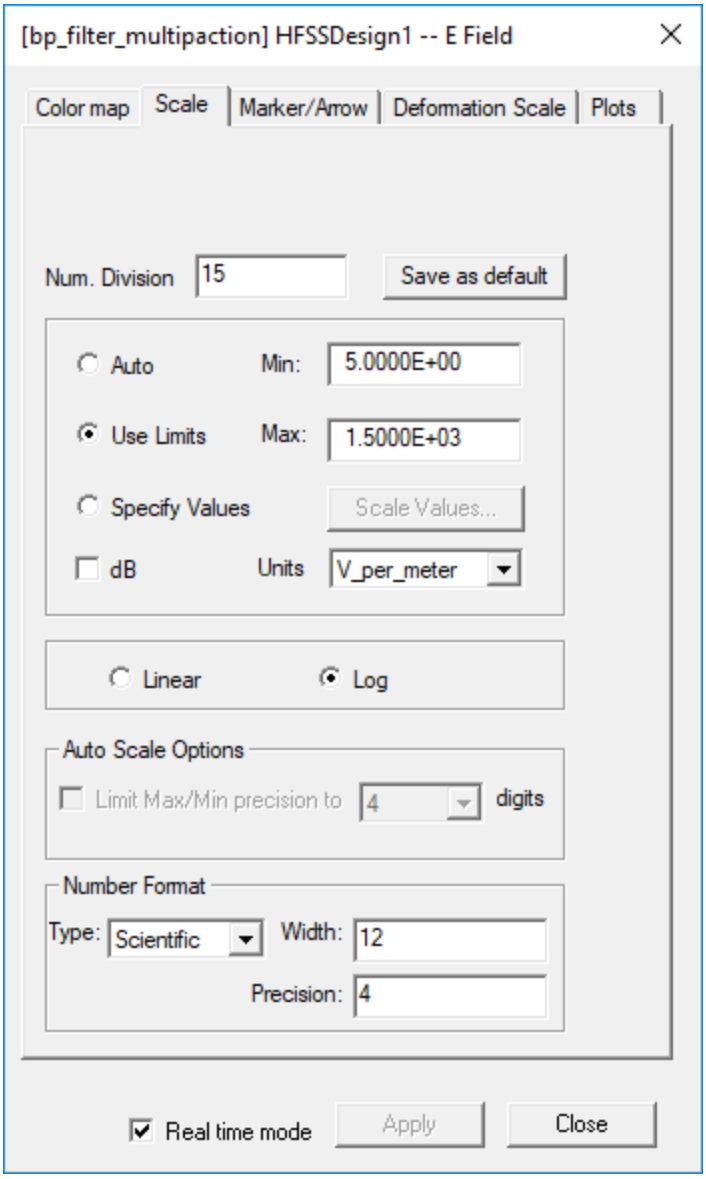
2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Scale** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, to change the number of divisions in the field plot scale, set the **Num. Division** field to a new value. You can click **Save as Default**, if desired.
5. Select one of the following scale options:

Auto The full range of field values will be plotted on the selected surface or volume. Selecting **Auto** enables the **Auto Scale Options** and disables the **Min** and **Max** fields. By default, precision is not limited and auto-min is the actual computed min on the plotted geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Limits	Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively. Selecting **Use Limits** enables the **Min** and **Max** fields and disables the **Auto Scale Options**. Field values have a precision of at most 6 decimal places (field solution files are saved in floating precision), so Min/Max numbers are displayed to this precision.
Specify Values	This enables a **Scale Values** button.

Optionally, when dB is checked, dB scale is used for the plot. It disables the Units field, Linear and Log and have "Linear" selected.

6. Optionally, use the **Units** drop-down menu to select the default unit of measure for the plot.

   The units specified here appear on the Color map for the fields plot, and for the properties dialog for the field quantities.

7. If you selected **Use Limits**, enter the lowest field value to be plotted in the **Min.** text box and the highest field value to be plotted in the **Max.** text box.

   If you selected **Auto** or **Use Limits** without having dB checked, the **Auto Scale Options** are enabled. You should only changed for cases where auto-min is a small number. Use the 'Limits Max/Min precision to' check box to enable setting the drop-down menu for the precision limit.

   The auto-min is the greater of the following:

   - Actual computed Min
   - Max/pow(10, num digits of field precision)

   If you selected Specify Values, you can click the **Scale Values** button. This opens a dialog with an editable, scrollable list of the current scale values. To apply the changes you make, click **OK**. To close the dialog box without making changes, click **Cancel**.

8. If you selected **Auto** or use **Limits** or **dB**, you can select one of the following options:

Linear	Field values are plotted on a linear scale.
Log	Field values are plotted on a logarithmic scale. If field plots have negative and positive values and when auto-scale is selected, the log-scale choice automatically sets the Min value as the Max/Min Ratio. (If field plots have all negative values, **Log** is not allowed.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Specify the Number Format for the plot as Auto, Scientific Notation, or Decimal. You can also specify Width and Precision for the plot.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.
11. Optionally, you can use the **Save As Default** button to save the following to the registry:
    - Whether to limit field precision
    - The number of digits of field precision
    - Whether to use log/linear scale
    - Whether to use dB

    Auto scale is the default for new plots. For scalar-in-volume plots, iso-surface (rather than cloud) is the default display
12. Click **Close** to dismiss the dialog box.

Modifying Field Plot Colors

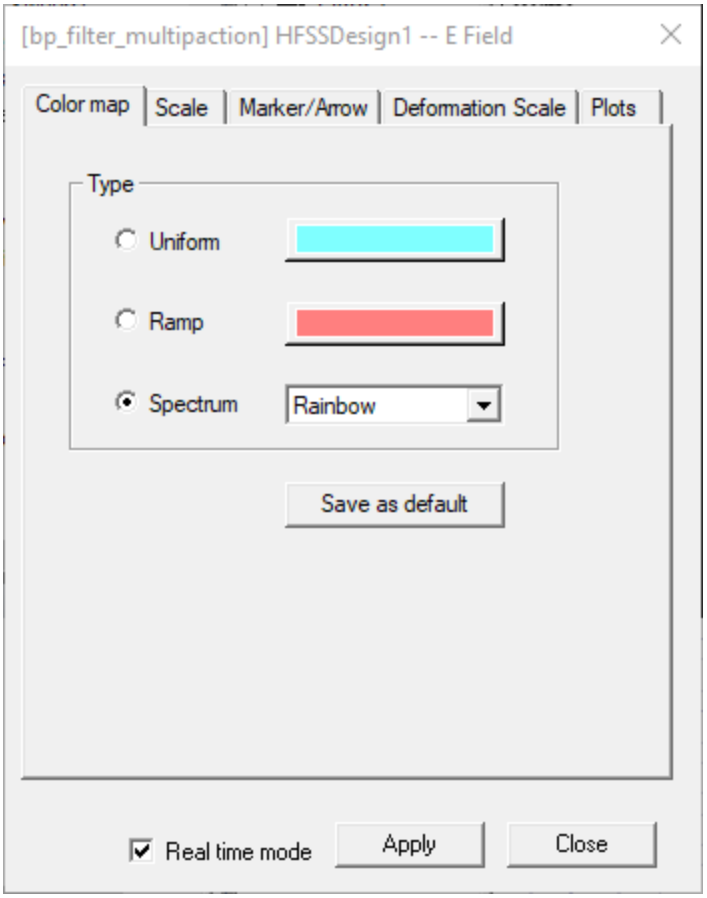
1. Click **Mechanical** > **Fields**> **Modify Plot Attributes** or, in the Project Manager, right-click **Field Overlays** and select **Modify Plots** from the short-cut menu (or use the "m" hotkey).

   The **Select Plot Folder** dialog box appears.
2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

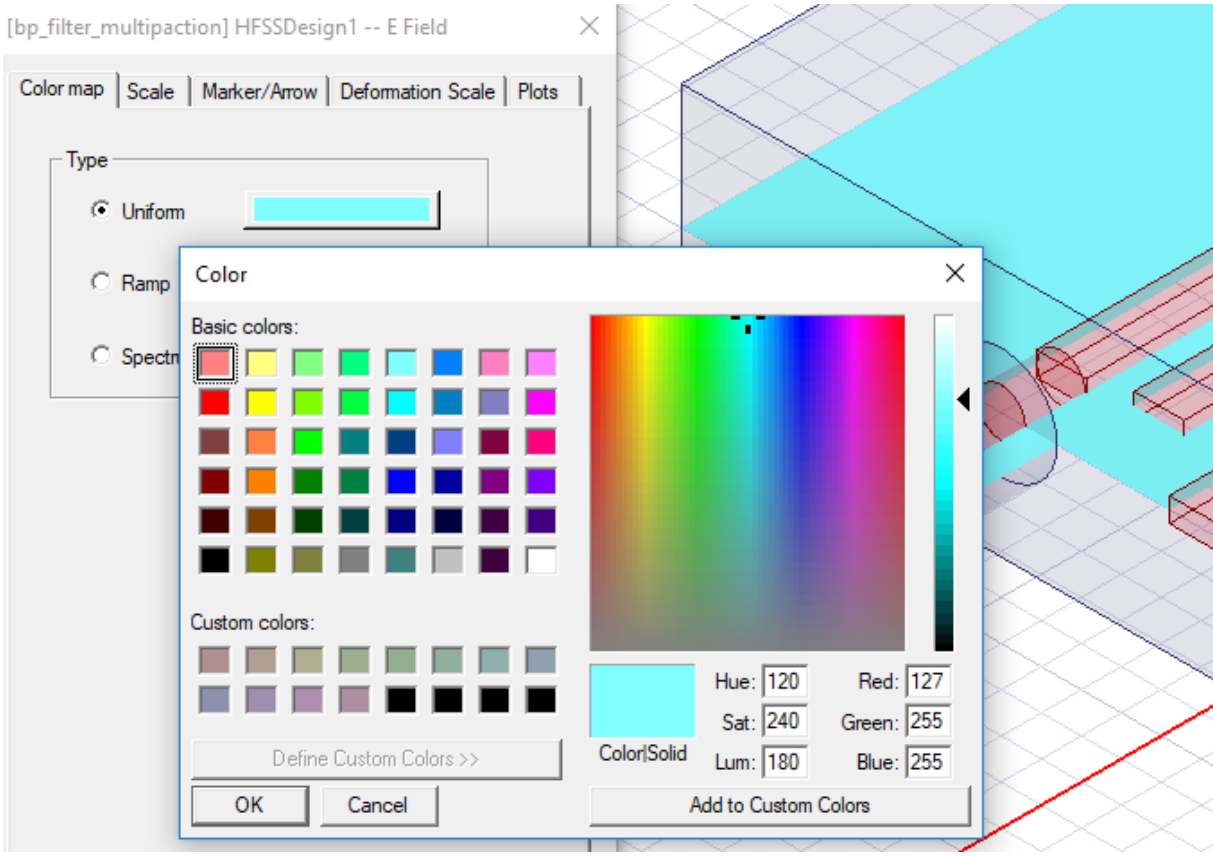
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the **Color Map** tab.
4. Select one of the following color types:

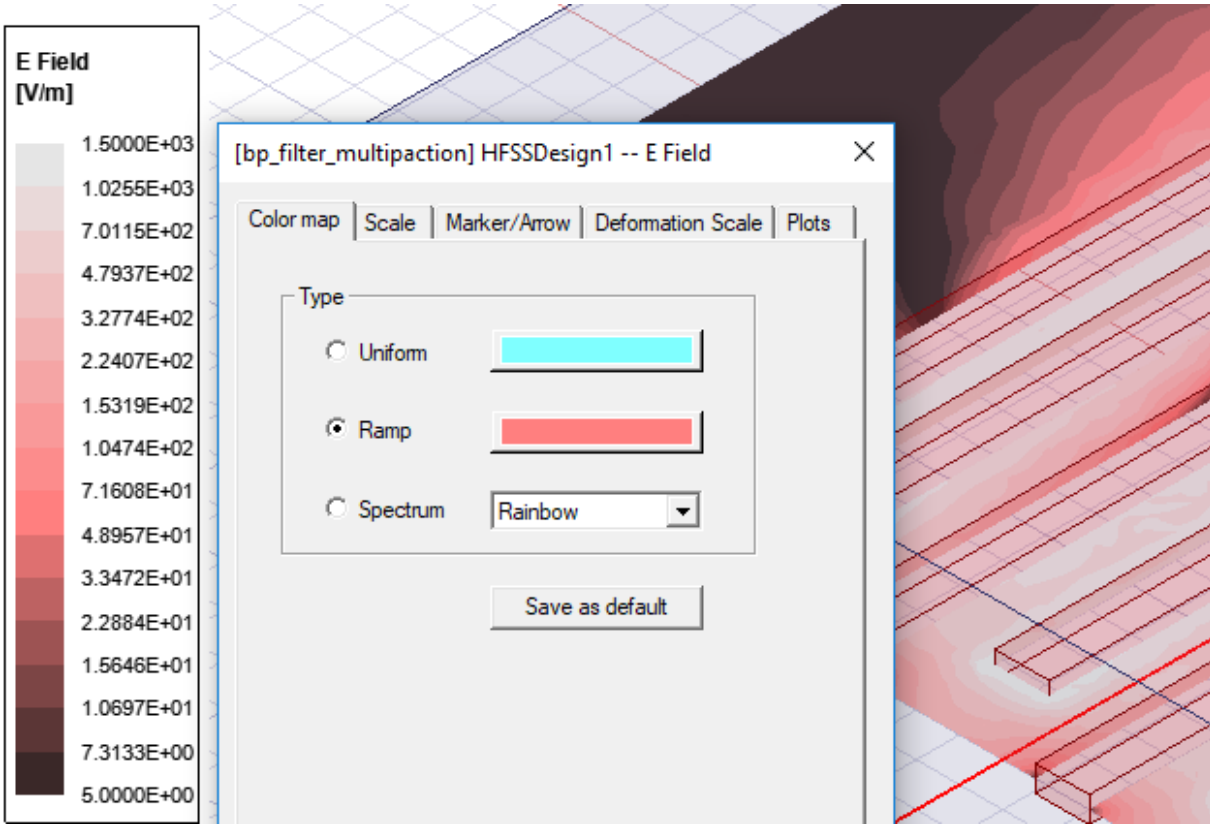
Uniform Field quantities are plotted in a single color. Click the button to choose the plot color from the **Color** button palette.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



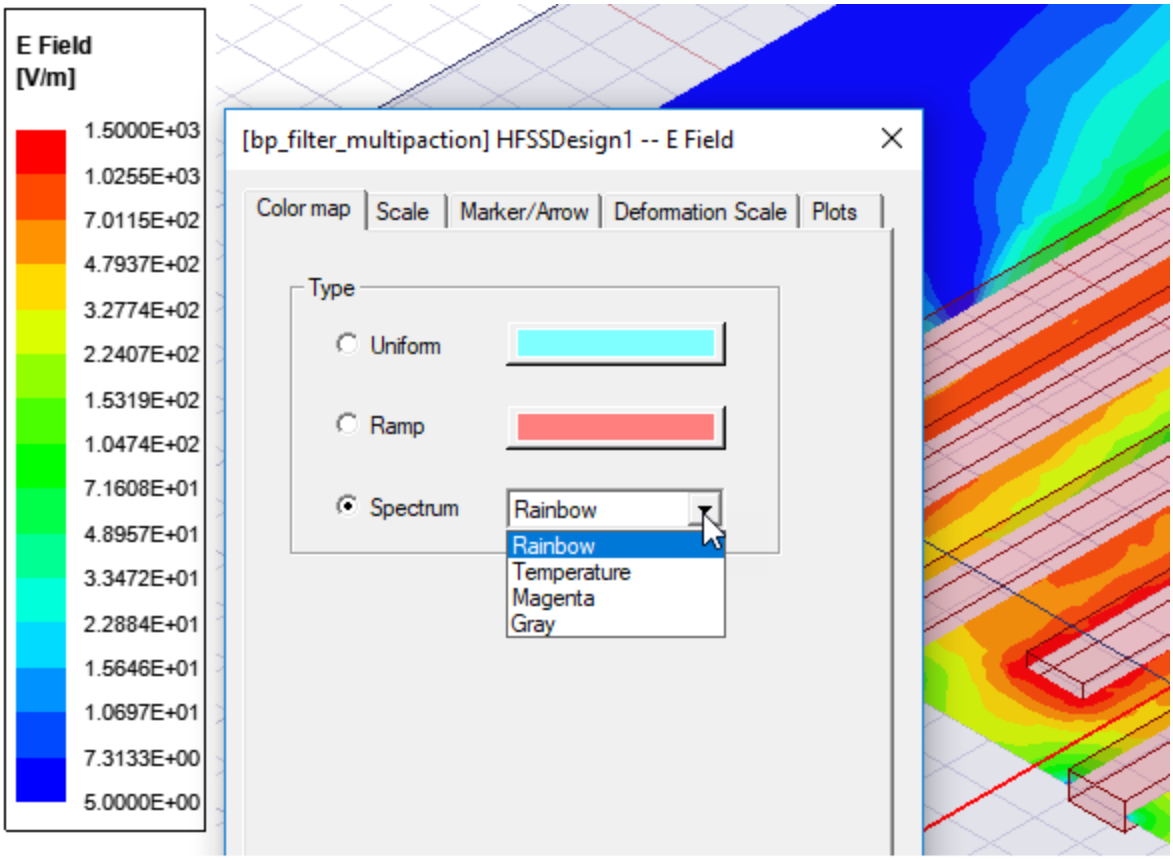
Ramp Field quantities are plotted in shades of a single color. Choose the plot color from the **Color** palette. The shade of the color corresponds to its field value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spectrum Field quantities are plotted in multiple colors. Choose a color spectrum from the drop-down menu. The values are Rainbow, Temperature, Magenta, and Grey. Each field value is assigned a color from the selected spectrum.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can choose **Save as Default**, if you want to use the current settings.

Select **Real time mode** if you want these, or subsequent changes to take effect immediately in the view window.

If this option is cleared, click **Apply** when you want to see the changes.

5. Click the **Scale Tab**.
6. In the **Num. Divisions** field, enter the number of colors to use in the plot.

   You can choose **Save as Default**, if you want to use the current settings.

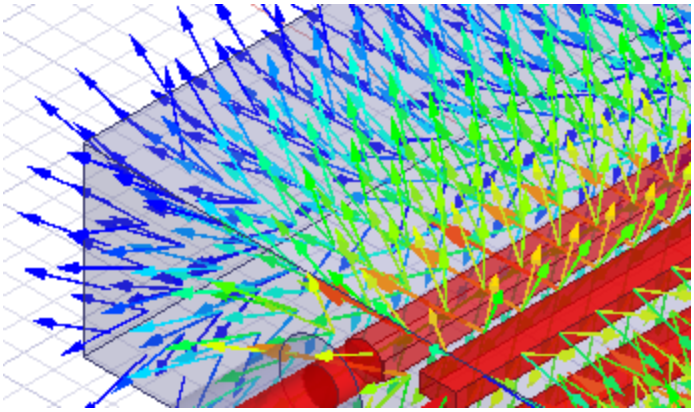
   Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Vector Field Plot Arrows

To change the appearance of a vector field plot's arrows:



1. Click **Mechanical**> **Fields**> **Modify Plot Attributes** or, in the Project Manager, right-click on **Field Overlays** and select **Modify Plot Attributes**. Alternatively, under **Field Overlays** in the Project Manager, right-click a subfolder (overlay type) heading and select **Modify Attributes**.

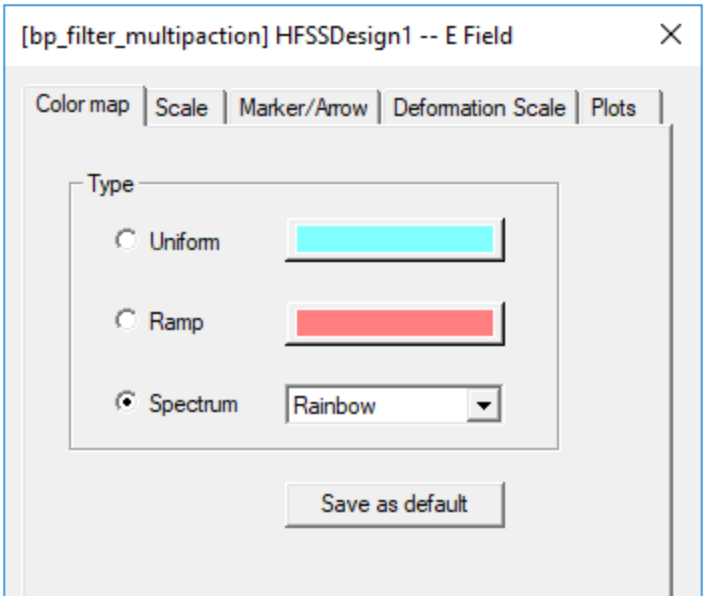
   If you select Modify Plot Attributes, the **Select Plot Folder** dialog box appears. If you select Modify Attributes, the appropriate attributes dialog box appears, bypassing the **Select Plot Folder** dialog box. In this case, skip to step 3.

2. Select the plot folder you want to modify, and then click **OK**.

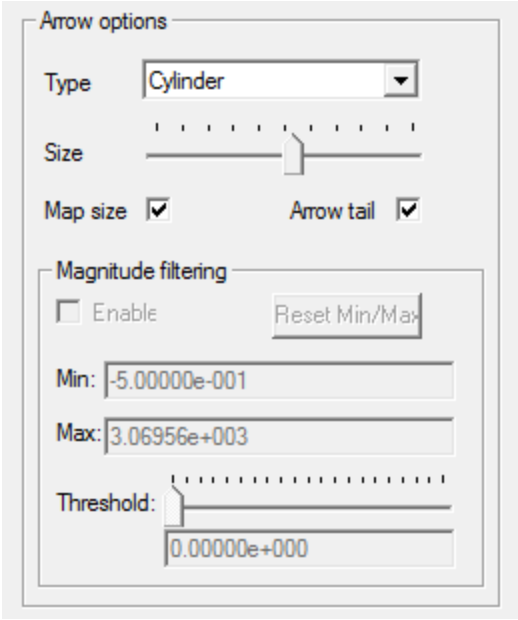
   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the **Marker/Arrow** tab.



4. Under **Arrow Options**, select one of the following arrow types:

Line	The arrows are displayed as 2D/flat.
Cylinder	The arrow tails are displayed as cylinders. The arrowheads are displayed as

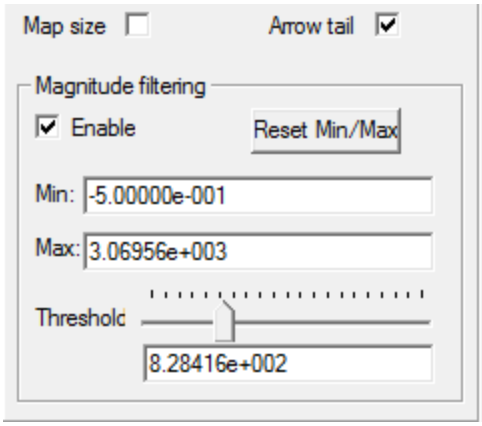
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D/round.

Umbrella The arrow tails are displayed as 1D lines. The arrowheads are displayed as 3D/round.

5. Use the **Size** slider to increase (move to the right) or decrease (move to the left) the length and dimensions of the arrows. The arrows are resized relative to the size of the model geometry.
6. Select **Arrow tail** to include tails on all arrows.
7. Check **Map size** to scale the size of the arrows to the magnitude of the field quantity being plotted.

   If you uncheck **Map size**, the Magnitude filtering is enabled. You can specify the Min and Max magnitude filtering, or use a Threshold slider to set a threshold for plotting vectors. Vectors under the threshold or below the Min are not plotted.



   The **Reset Min/Max** button lets you reset min/max to the current intrinsic values.

8. Click the **Plots** tab.
9. Mechanical plots arrows on a grid that is superimposed on the surface or object you selected for the plot. Under **Vector plot**, use the **Spacing** slider to increase (move to the right) or decrease (move to the left) the distance between arrows (grid points.)
   - Select **Uniform** if you want the arrows to be spaced equally.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.

11. Click **Close** to dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Scalar Field Plot Isovalues

1. Click **Mechanical**> **Fields**> **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

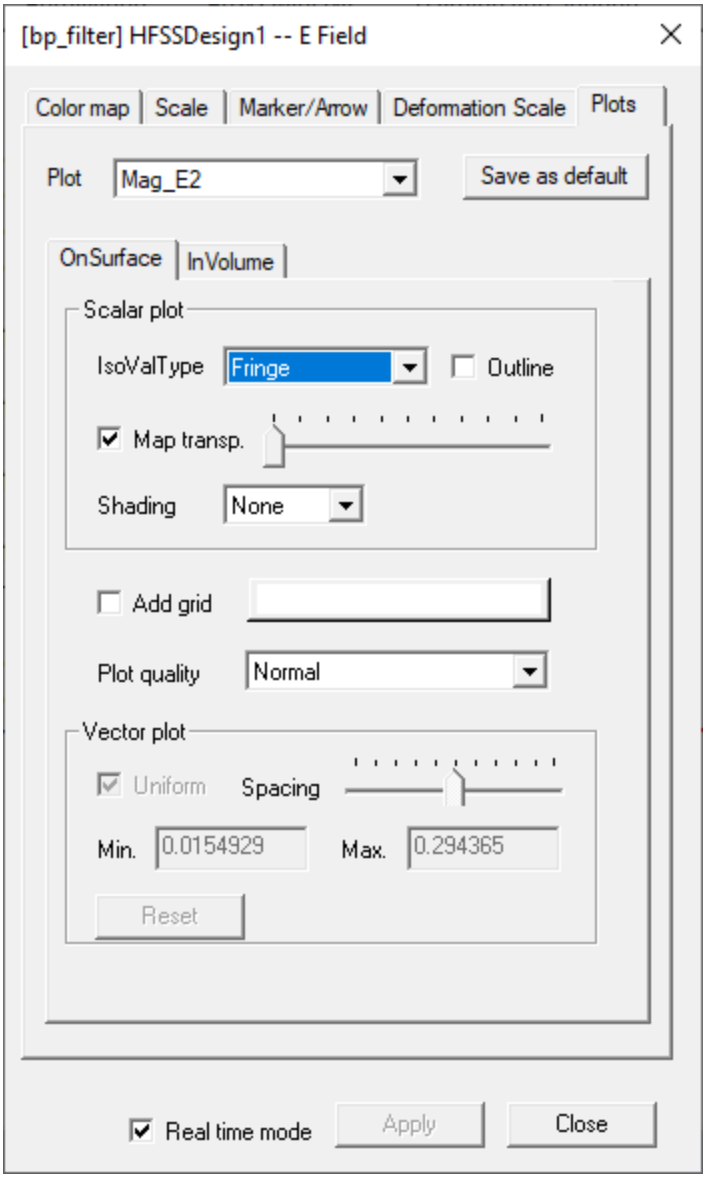
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab. If you have more than one plot, a drop down lets you select. The OnSurface tab appears for surface plots and InVolume for volume plots.

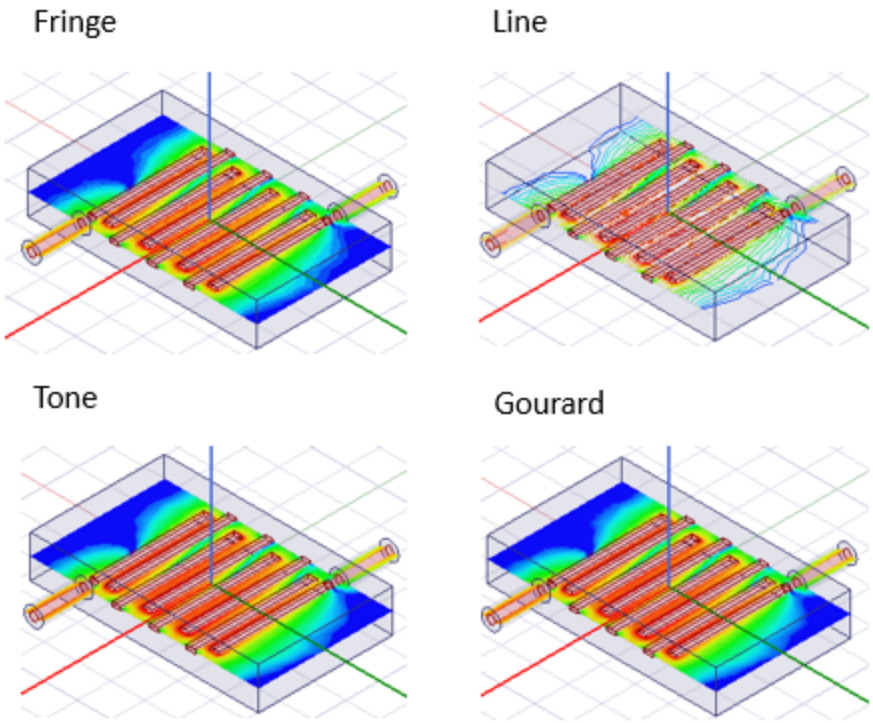
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. If the plot is a scalar surface plot, do the following:
   a. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Line	Lines are drawn along the isovalues.
Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Optionally, if you selected **Fringe** or **Tone**, select **Outline** to add a border line between isovalues.

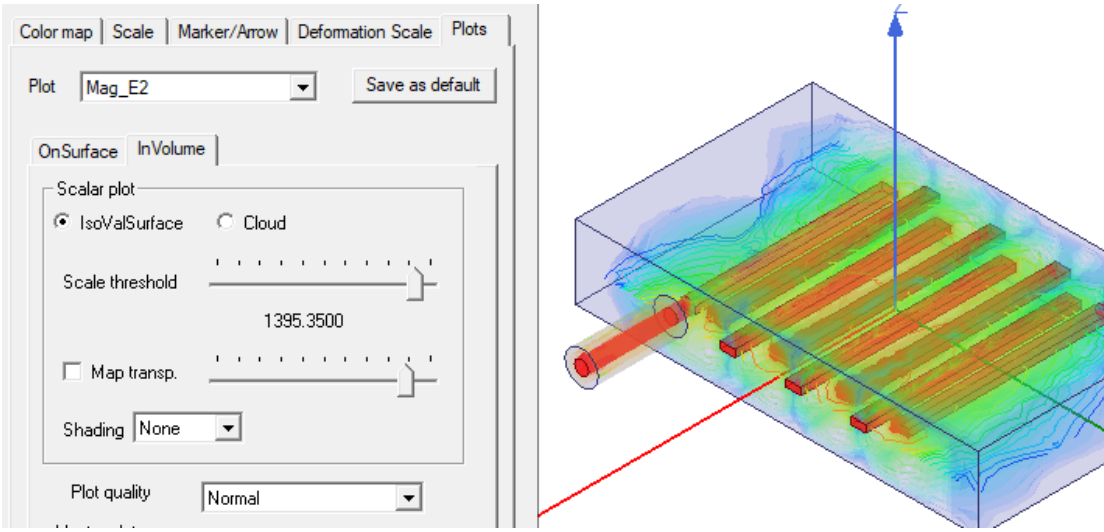
c. For shading on OnSurface plots, if you are using Lighting, you can adjust the Shading as None, Flat, or Smooth.

5. If the plot is a scalar volume plot, do the following:

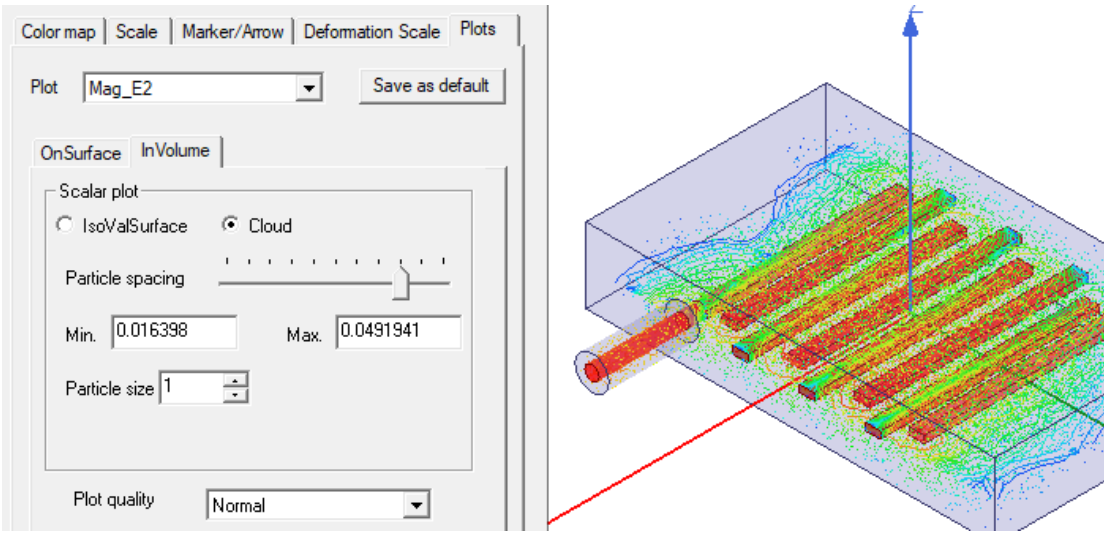
a. Select one of the following display types:

IsoValSurface	Color is drawn on the isovalues. This choice includes “Scale threshold” slider barthat allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it. When "Map transp." is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cloud Field values are represented by points that illustrate the spatial distribution of the solution. The higher the solution value, the greater the cloud density. You can adjust particle spacing, Min. and Max, values, and Particle size.



6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Close** to dismiss the window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Mesh Visibility on Field Plots

To display or hide the mesh on field plots, or change the mesh's color:

1. Click **Mechanical**> **Fields**> **Modify Plot Attributes** .

   The **Select Plot Folder** dialog box appears.

2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Select **Add Grid** to display the mesh.
5. Optionally, select a color for the mesh from the **Color** palette.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Close** to dismiss the window.

Setting Field Overlay Plot Defaults

Each new field plot uses the default plot settings specified in the **Set Plot Defaults** dialog box.

To modify the default plot settings:

1. If a plot folder has not been created, click **Field Overlays** in the Project Manager.
2. Click **Mechanical** > **Fields** > **Set Plot Defaults**.

   The **Set Plot Defaults** dialog box appears.

3. Select the solution to plot from the **Solution** drop-down menu.
4. Select the plot folder in which new plots will be stored from the **Quantity type** drop-down menu. Choose one of the following options:

New Folder	Each new plot will be stored in a separate folder in the project tree.
Automatic	Each new plot will be stored in a folder determined by Mechanical as the most appropriate based on the plotted field quantity. For example, all surface magnitude E plots will be stored in the same folder.
An existing	Select the existing folder in which you want to store new plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

folder

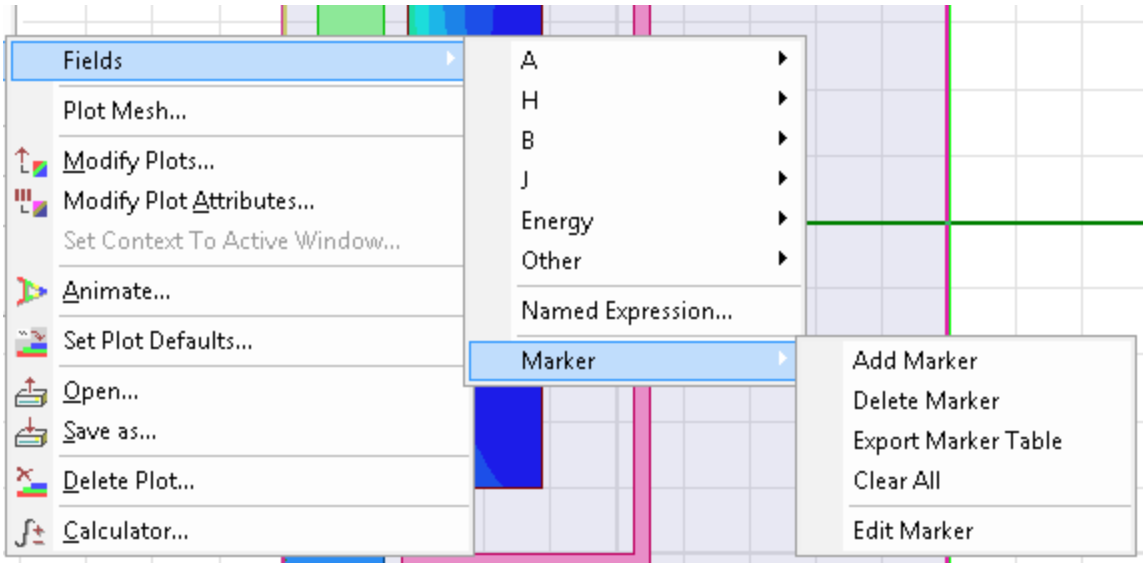
> **Note:**
>
> Plots stored in the same folder will use the same color key. The **Auto** scale setting will be based on the maximum field solution value present in a plot.

5. Under **Intrinsic Variables**, specify the frequency and phase angle at which the field quantity is evaluated.
6. Click **OK**.

Working with Scalar Field Plot Markers

The field overlay plot marker feature enables you to create a marker at selected points in the scalar field overlay plot geometry, and to obtain the field value at that point. The fields Marker sub-menu enables you to:

- Add Marker
- Delete Marker
- Export Marker Table
- Clear All markers
- Edit Marker



Adding a Field Plot Marker

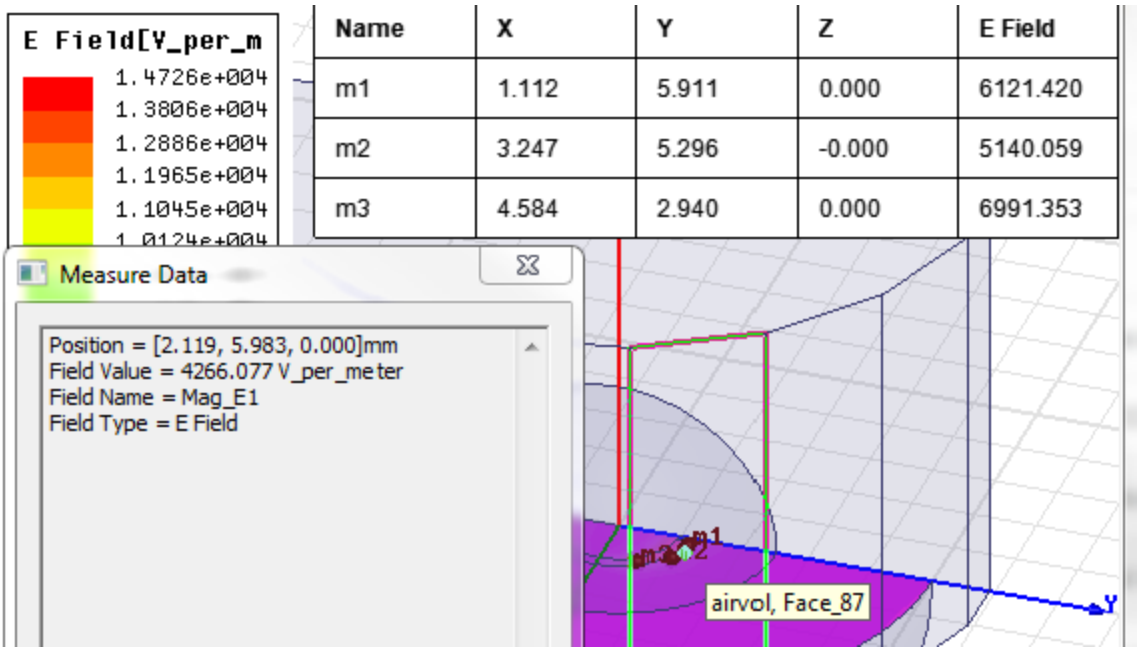
To add one or more field plot markers to a scalar field:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the main menu click **Mechanical** and then select **Fields** > **Plot Fields** > **Marker** > **Add Marker**. Alternatively, you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Fields** > **Marker** > **Add Marker**.

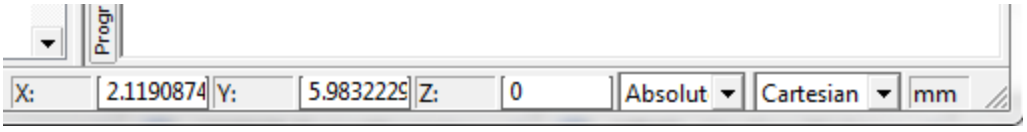
   The *Measure Data* dialog box opens, and a round dot appears at the tip of the cursor.

2. Drag the dot over the spot on the field overlay plot where you want to add a marker. The Measure Data dialog box shows detailed information for the spot currently under the dot.



Name	X	Y	Z	E Field
m1	1.112	5.911	0.000	6121.420
m2	3.247	5.296	-0.000	5140.059
m3	4.584	2.940	0.000	6991.353

3. Click the desired point in the field overlay plot to add the marker at that location. Alternatively you can select the position of the marker by entering the values manually in the edit fields at the bottom of the window as shown below.



   A table showing the marker coordinates and associated field value is also created and added to the modeler window.

4. Repeat as desired to add additional markers.
5. Press the **Esc** key when finished adding markers.

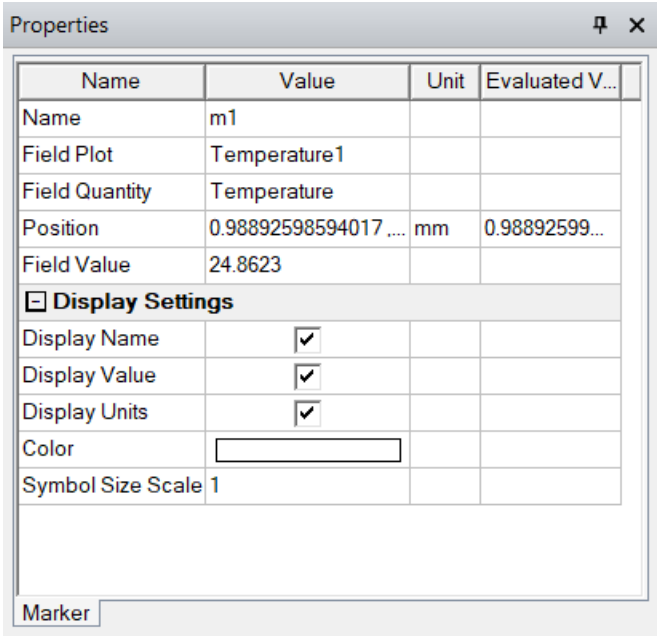
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Field Plot Markers

To edit field plot markers:

1. In the marker table, click on the marker you wish to edit. The marker in the 3D Modeler window and its row in the table are highlighted. The properties of the marker are displayed in the **Properties** window.

> **Note:**
>
> Press **Ctrl** and select multiple marker rows in the table to edit marker properties simultaneously.



2. You can edit the following marker properties:
   - **Name**: Specify the name that appears next to the marker in the 3D Modeler window.
   - **Display Name**: Toggle to show or hide the marker's name.
   - **Display Value**: Toggle to show or hide the marker's value.
   - **Display Units**: Toggle to show or hide the marker's units.
   - **Color**: Click the color to change the marker's color in the **Color** dialog box.
   - **Symbol Size Scale**: Enter a different number to adjust the size of the marker in the 3D Modeler window.
3. Optionally, when a marker is selected, press the **Delete** key to delete the marker and its corresponding entry in the marker table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deleting a Field Plot Marker

To delete a field plot marker:

1. Click on the marker you want to delete to select it. The row corresponding to the selected marker will be highlighted in the marker table.

   Press and hold the **Ctrl** key and click to select multiple markers.

2. On the main menu click **Mechanical** and then select **Fields> Plot Fields> Marker> Delete Marker** to delete the selected marker(s).
   - Alternatively, you can right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Delete Marker**.
   - You can also simply press the **Delete** key to delete the selected marker(s).

Exporting a Field Plot Marker Table

You can export a field plot marker table to either a comma- or tab-delimited file as follows:

1. On the main menu click **Mechanical** and then select **Fields> Plot Fields> Marker> Export Marker Table**.

   Alternatively you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Export Marker Table**.

2. In the *Export As* dialog box, choose the export format, either **.csv** or **.tab**, and save the file in the desired location.

   The exported file can then be imported into another application such as a spreadsheet.

	A	B	C	D	E
1	Name	X	Y	Z	H[A_per_m]
2	m1	2	22.935	1.517	1234
3	m2	1.517	17.373	1.517	123
4	m3	2.488	23.516	1.517	234
5	m4	2.751	23.828	1.517	1324

Clearing All Field Plot Markers

To clear all field plot markers in the active modeler window do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the main menu, click **Mechanical** and then select **Fields> Plot Fields> Marker> Clear All**.
- Alternatively, right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Clear All**.

Deleting all the markers also removes the marker table.

Mapping Scalar Field Plot Transparency to Field Values

1. Click **Mechanical** > **Fields**> **Modify Plot Attributes**.

   The *Select Plot Folder* dialog box appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Use the **Map transp.** slider to increase (move to the right) or decrease (move to the left) the transparency of the plot.
   - If you select **Map transp.**, the transparency of field values increases as the solution values decrease.
5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the dialog box.

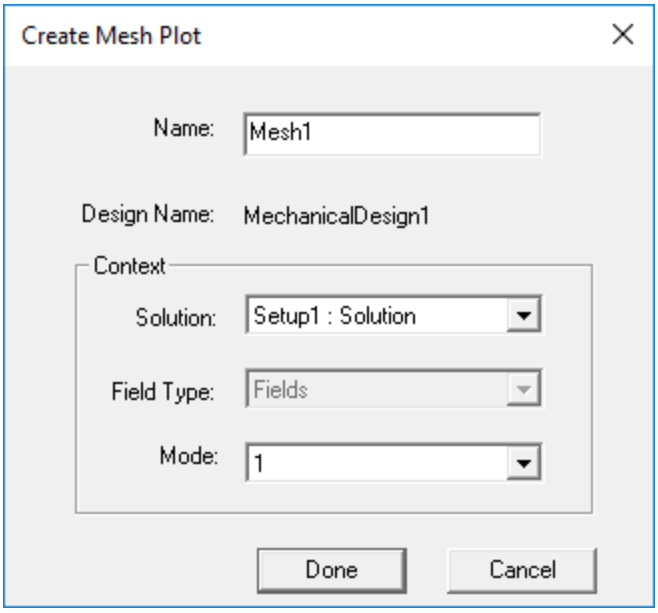
Plotting the Mesh

Before or after the solution is complete, you can plot the finite element mesh on surfaces or within 3D objects. For an HFSS or Maxwell design that includes Layout Components, you can also select nets and layers from the layout components to plot.

1. Select one or more surfaces or objects to create the mesh plot on or within. If the design is HFSS and includes Layout Components, you can select the surfaces either before or after selecting **Plot Mesh**.
2. Select **Plot Mesh** in one of two ways:
   - Click **Mechanical** > **Fields**> **Plot Mesh**.
   - In the Project Manager, right-click **Field Overlays** and select **Plot Mesh...**

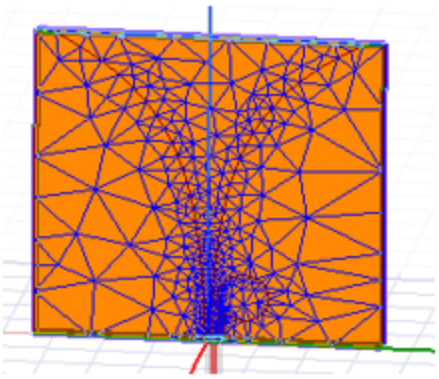
     The *Create Mesh Plot* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

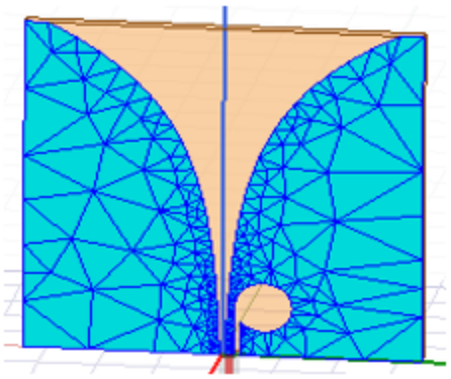


3. Select the **Solution** to use from the drop-down menu.
4. If **Field Type** is enabled, you can select from the drop-down menu for a mesh plot.
5. Click **Done**.

   The mesh appears on the surface or object you selected. For example:



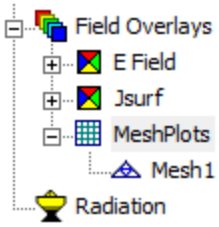
Mesh Plot of All Objects



Mesh Plot of Selected Surfaces

An icon for the mesh also appears in the Project Manager under **Field Overlays** > **Mesh Plots**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If a solution is ongoing, you can select the **Mesh Plots** icon in the in the Project Tree, right-click to display the shortcut menu, and select **Update Plots**. This updates the mesh plot to include the latest data available. After the last adaptive pass, the mesh plot is automatically updated.

If a mesh includes seeding, these effects appear at the start of the adaptive passes. Any mesh adaptation at the start of a subsequent pass is not plotted until after that pass is completed. This delay ensures that mesh plots and actual solutions remain consistent with each other.

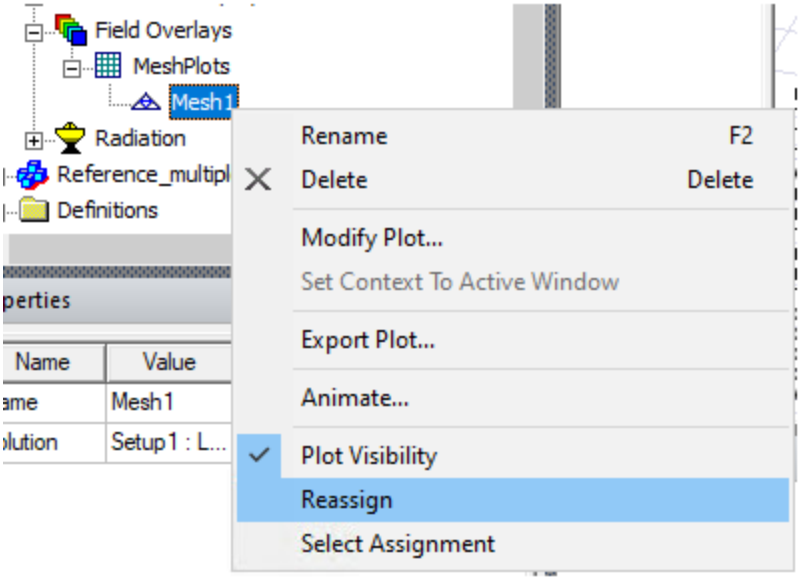
You can modify an existing plot by selecting the plot and changing the properties.

If you modify the geometry of an object, the solution is invalidated and the mesh plot is removed.

If you have plotted the mesh of a Solve-Inside object, you can view the internal mesh by using either the clip-plane feature or the Model Analysis dialog box.

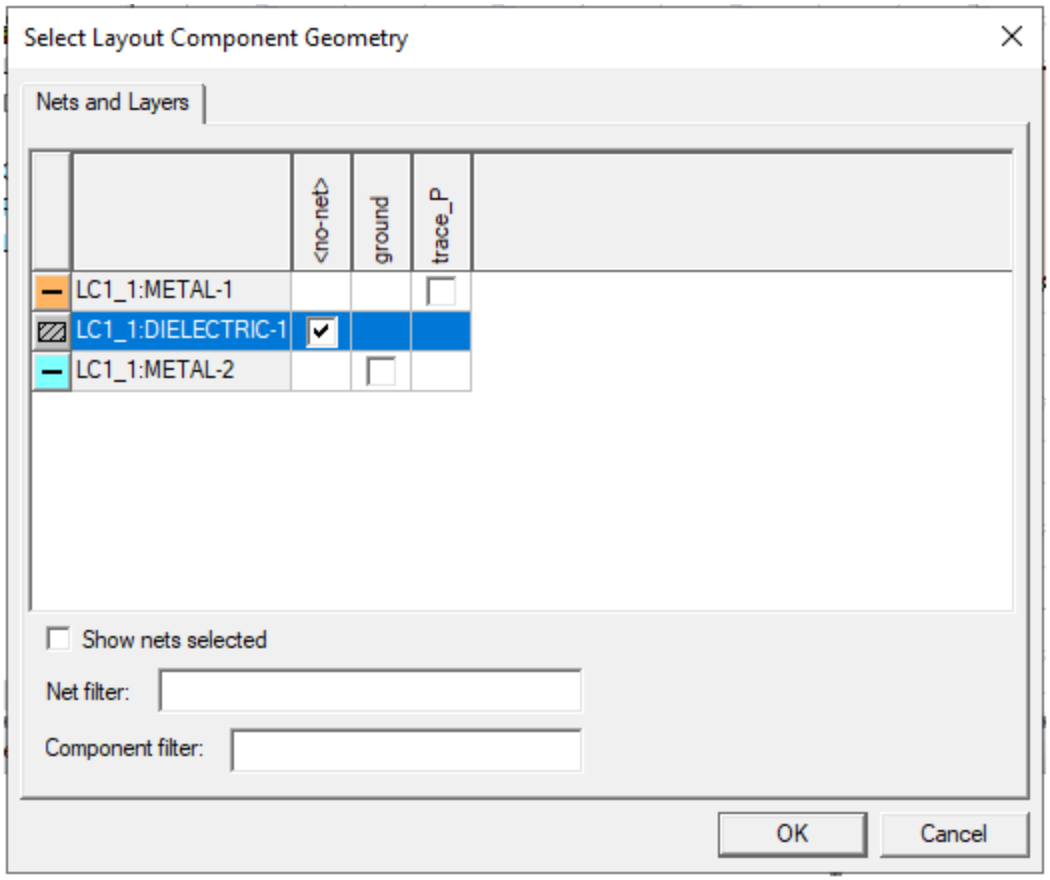
After making selections click **Done**. The mesh plot will be generated on the objects belonging to the selected layer/net combinations.

If you want to reassign a mesh or field plot, either select 3D geometries from layout components in the modeler window before selecting **Reassign** in the right-click menu, or select **Reassign**.



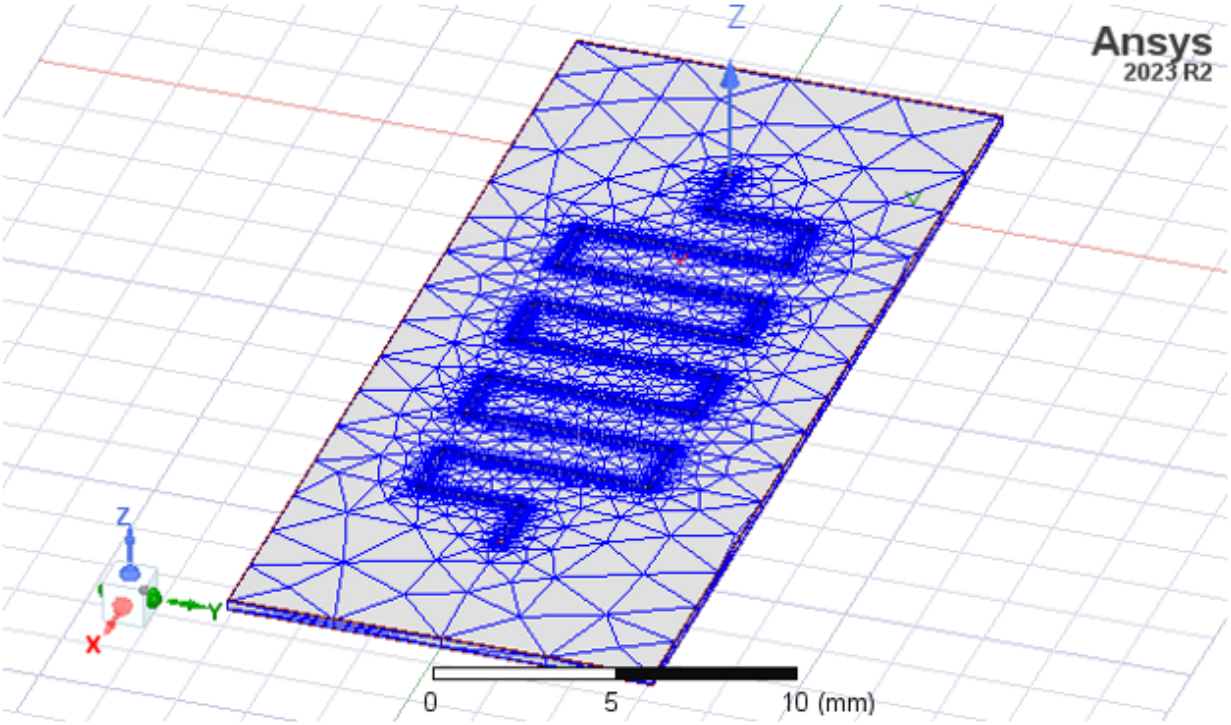
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For HFSS designs with layout components, the **Select Layout Component Geometry** dialog appears. This has the same functionality and menus as the this **Nets and Layers** section of the **Create Mesh Plot** dialog.



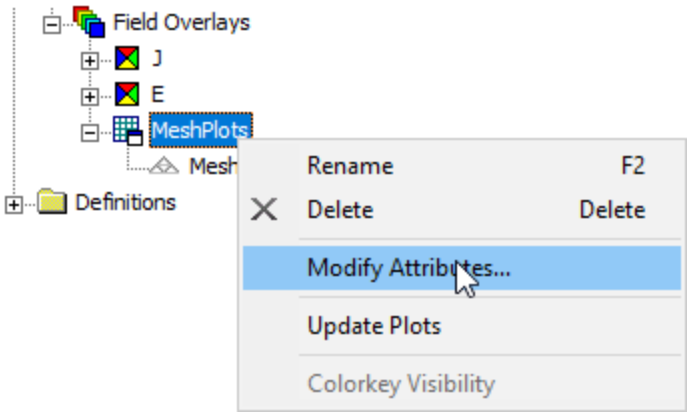
Change the selection in the dialog and click **OK**. The mesh plot will be re-assigned to the newly selected objects, in addition to any 3D geometries previously selected from the modeler window. If you click cancel, the plot is only re-assigned on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message. Change the selection in the dialog and click **OK** to display the reassigned mesh plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



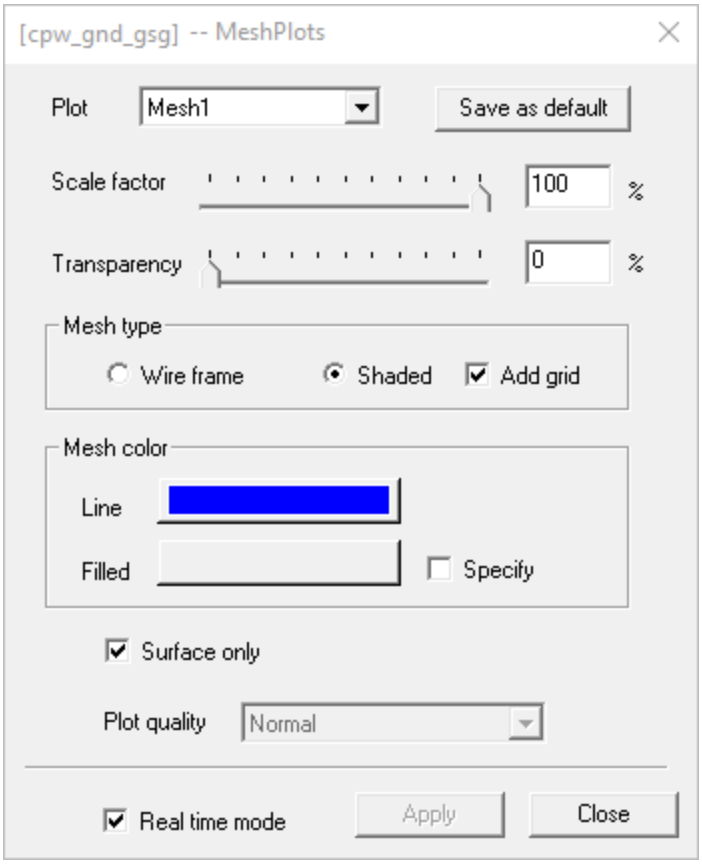
Setting Mesh Plot Attributes

1. In the Project Tree, expand **Field Overlays**.
2. Right-click **Mesh Plots** and select **Modify Attributes**.



The *MeshPlots* window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can also open the MeshPlots window from **Mechanical** > **Fields** > **Modify Plot Attributes**. You will be prompted to select the folder containing the mesh you want to modify.

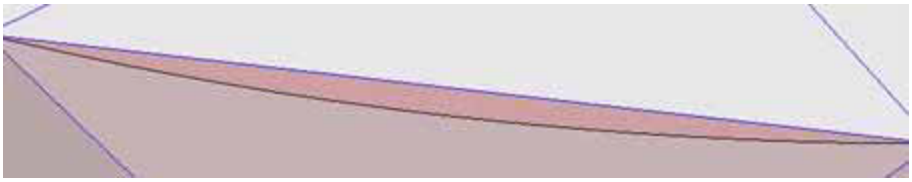
For Mesh plots, the following attributes can be modified:

Plot	A drop-down menu of available plots.
Scale Factor	The size at which the tets are displayed. Scaling may let you analyze particular situations better. For example, a scale factor of 80% draws the tetrahedra at 80% of their original size. Use the **Scale factor** slider to increase or decrease the size of the tetrahedra.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparency	The degree of transparency for the tets. This is useful for viewing objects or plots behind the current plot. Use the **Transparency** slider to increase or decrease plot transparency.
Mesh Type	Whether to display the tets as wire frame or shaded, and whether to Add Grid.
Mesh Color for Line and Fill	The color for the tet edge lines and fill. Clicking the button for each displays a color selection dialog box. You must select **Specify** to enable fill color.
Surface Only	Whether to plot the surface only, or all tets inside selected objects.
Plot Quality	The resolution quality of the plot. Normal by default. Use the drop-down menu to select Coarse, Normal, Fine, or Very Fine. The higher the resolution, the more memory used.
Real Time	Whether to show changes to a mesh immediately in the view window. If this option is cleared, click **Apply** when you want to see the changes.

For **Plot Quality**, higher quality selections permit visualization of curvilinear mesh elements. With the **Coarse** setting, facets do not snap to the midpoint edge:



The **Normal** setting displays snapping to the midpoint edge:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Fine** setting displays more of the curvilinear tets:

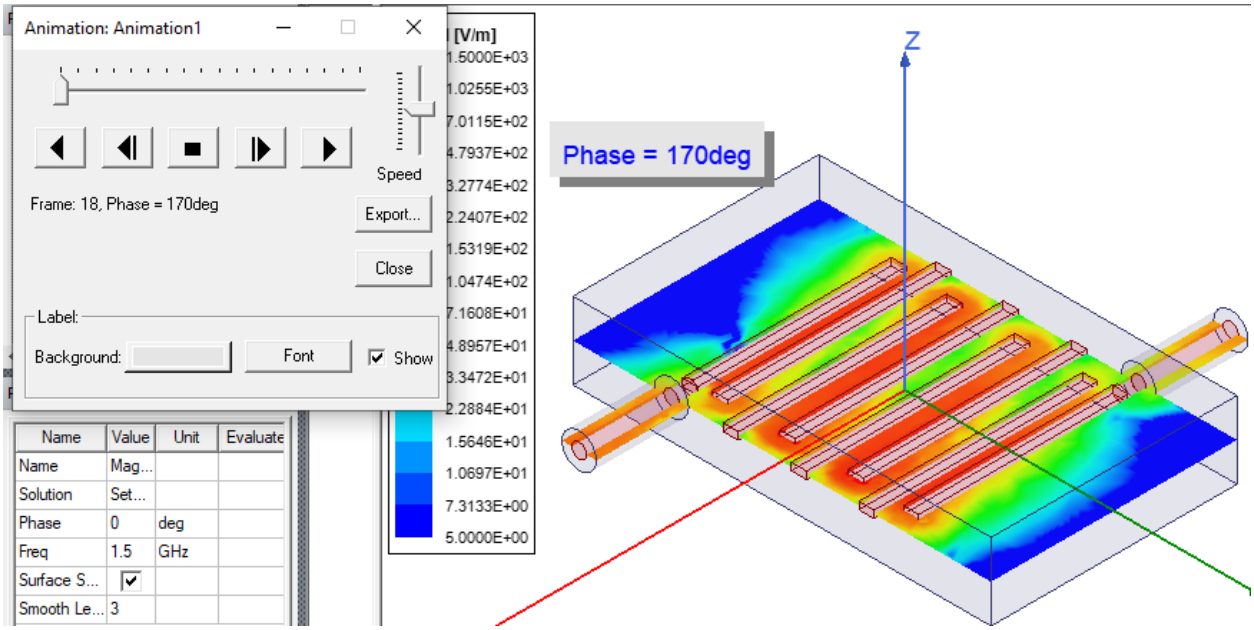
The **Very fine** setting gives the most accurate display, but uses the most memory:

3. Click **Save as default** button if you want the tab's settings to apply to mesh plots created after this point.
4. Click **Close**.

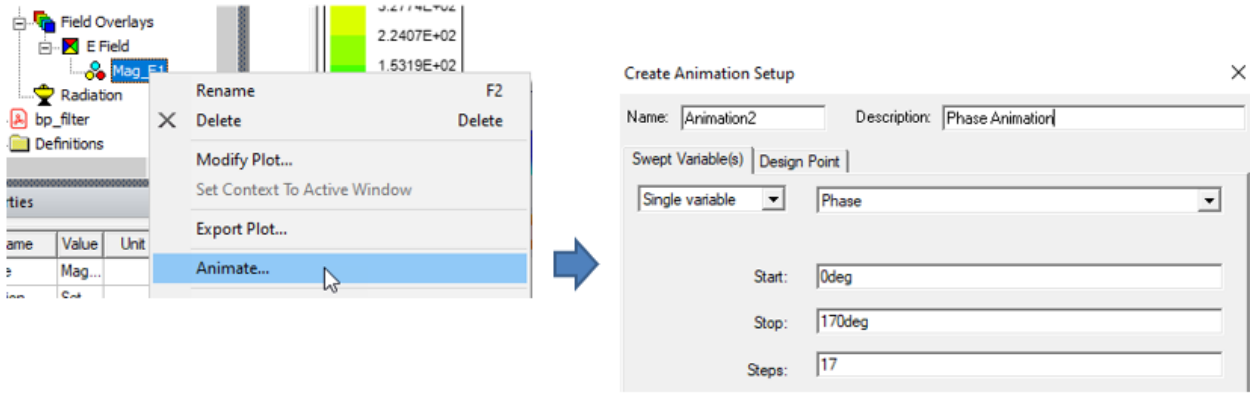
Creating Animations

An animated plot is a series of frames that displays a field, frequency, parametric value, mesh, virtual ray trace or geometry at varying values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To create an animated plot, you specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation. You specify how many frames to include in the animation.



The following sections describe how to create different kinds of animations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Each animation frame requires memory for storage which depends upon the mesh size and type of plot. Memory usage may become very large during plot animations. To reduce memory usage, specify the minimum number of frames possible. See General Options for more information.

Note:

Graphics System requirements for Optimal Performance

In order to obtain optimal performance improvements on fields overlay plot, a workstation-class 3D-capable graphics card with at least 512 MB of memory that supports OpenGL version 2.0 or higher is needed.

On Windows, the default OpenGL version support is v1.1, so you might need to update graphics driver to the latest version;

If you access the application through Windows Remote Desktop which only supports Generic GDI (functionally equivalent to OpenGL v1.0), the performance improvement will also not be available;

To view OpenGL version/extensions supported by your card, the OpenGL Extension Viewer tool is accessible via softpedia.com.

If animation is slow, especially for complex models, for some older graphics cards, you can improve performance by accessing **NVIDIA Control Panel**> **3D Settings**> **Manage 3D Settings – Global Settings** tab and choosing the **Workstation App – Dynamic Streaming** option from the **Global Presets** drop-down menu.

You can export the animation to animated Graphics Interchange Format (GIF), to Audio Video Interleave (AVI) format, or to WebM (.webm) format.

Creating Geometry Animations

You can create geometry animations to evaluate the effect of varying geometry variables on the model. You can also create more complex animations by creating time variables associated with object locations. This section describes the general procedure for geometry animation, and shows a geometry animation with a field overlay, and a second animation for an HFSS Antenna Example project that includes multiple objects moving with respect to a time variable.

Prerequisites

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You must define at least one variable associated with the geometry before you create a geometry animation.

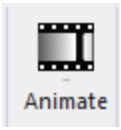
If you want to overlay a Field Solution on a geometry animation you must have solved fields for an Optimetrics setup using the same variable.

Procedure:

Following is the general procedure for creating an animation that varies a part of the model geometry.

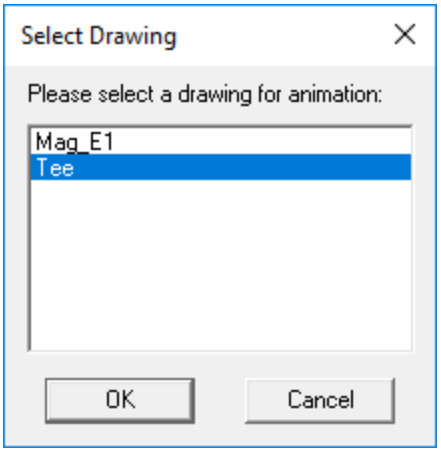
1. Right-click in the *Modeler* window, and click to **View** > **Animate...** or

   Select the **View** tab of the ribbon, and click the **Animate** icon.



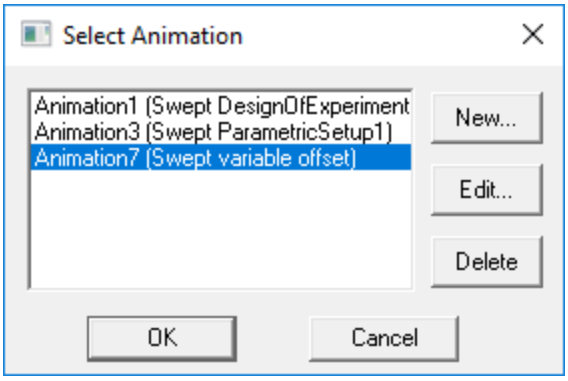
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

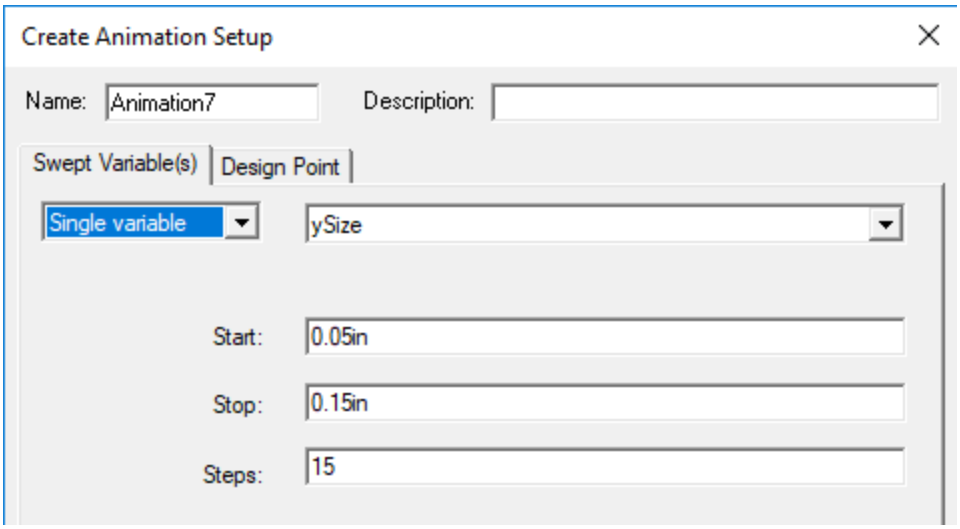


   If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

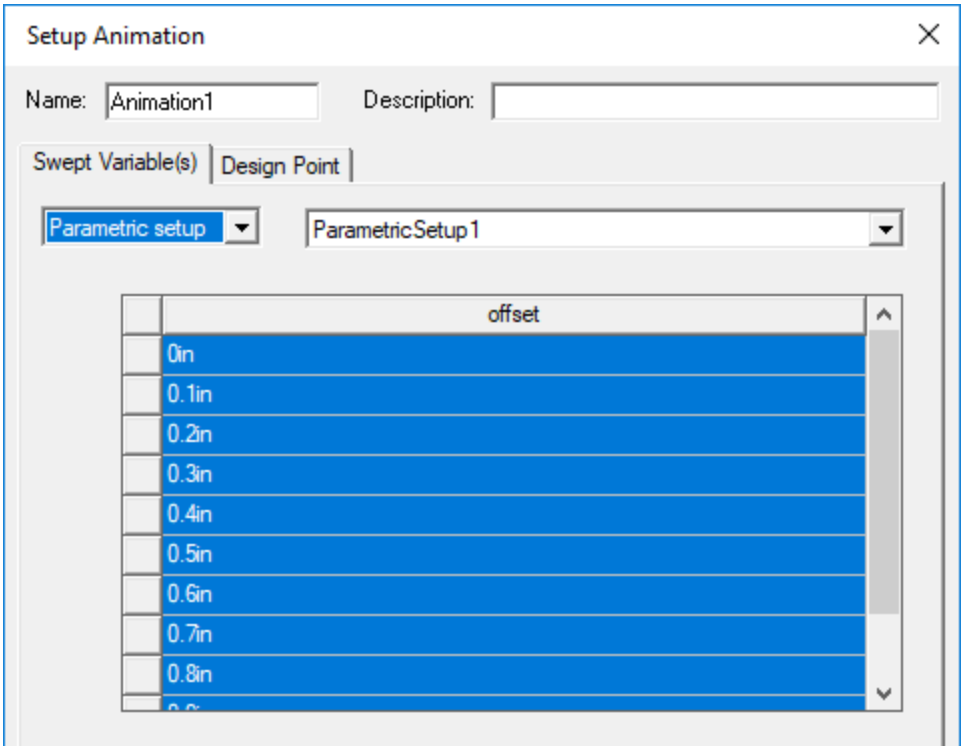


The *Create Animation Setup* dialog box appears:



3. For single variable:
    a. Select the geometry variable that you want to animate from the **Swept Variable** list.
    b. Specify the Start, Stop, and Steps values to include in the animation.
4. If your have created one or more Parametric Sweeps for one or more geometric variable, you can select, **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. If you have created a **Design of Experiments** setup, you can select to define an animation based on that setup and values.
6. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.
7. Click **OK**.

   The animation begins in the view window. It will display one frame for each variable value.

Creating Parametric Animations

Parametric animations can be based on parametric sweeps. Following is the general procedure for creating an animation of a plot based a parametric sweep.

Prerequisites

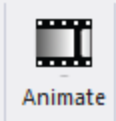
- For animation of a field plot/overlay plot over a geometric variable, you must define and solve at least one optimetric sweep with same variable values and Save Fields selected before you can create a parametric animation of a field solution. See Creating Geometry Animations.
- For animation of geometry and VRT plots, solving parametric setup is not required.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Animations can be based on time variables and objects whose location is linked to the time variable. See the animated example below.

Procedure:

1. Right-click in the **Modeler** window, and click **View**> **Animate...**, or select the **View** tab of the ribbon, and click the **Animate** icon.

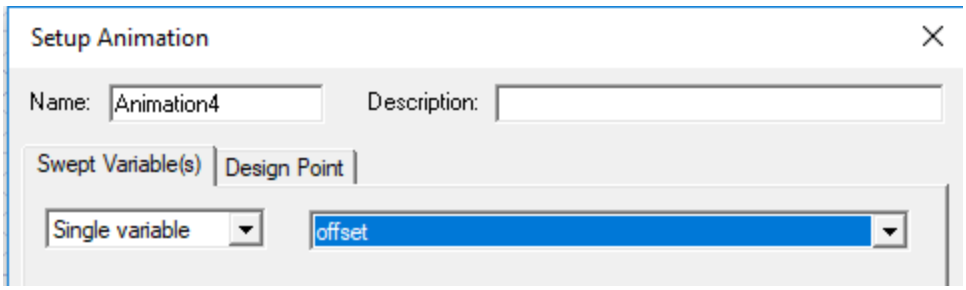


   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box:
   a. Select the geometry variable to vary in the animation.
   b. Select the object you want to animate.

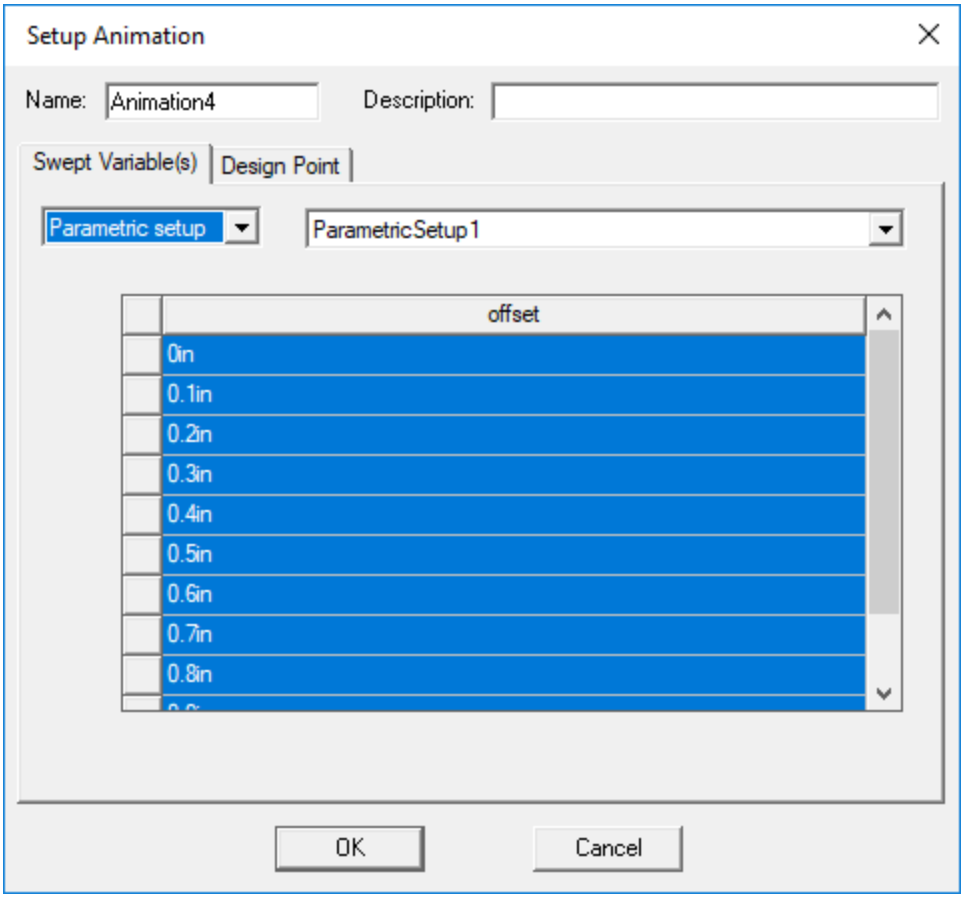
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

   The *Setup Animation* dialog box appears.



3. If your have created one or more Parametric Sweeps for one or more geometric variables, you can select **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

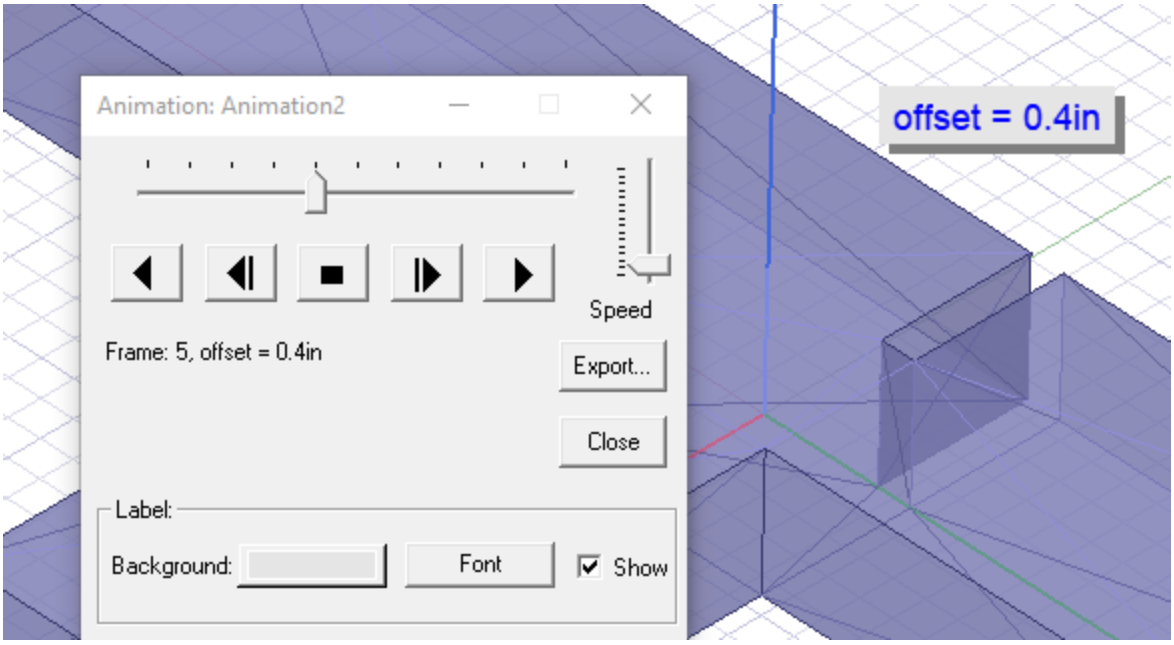


4. Click **OK**.

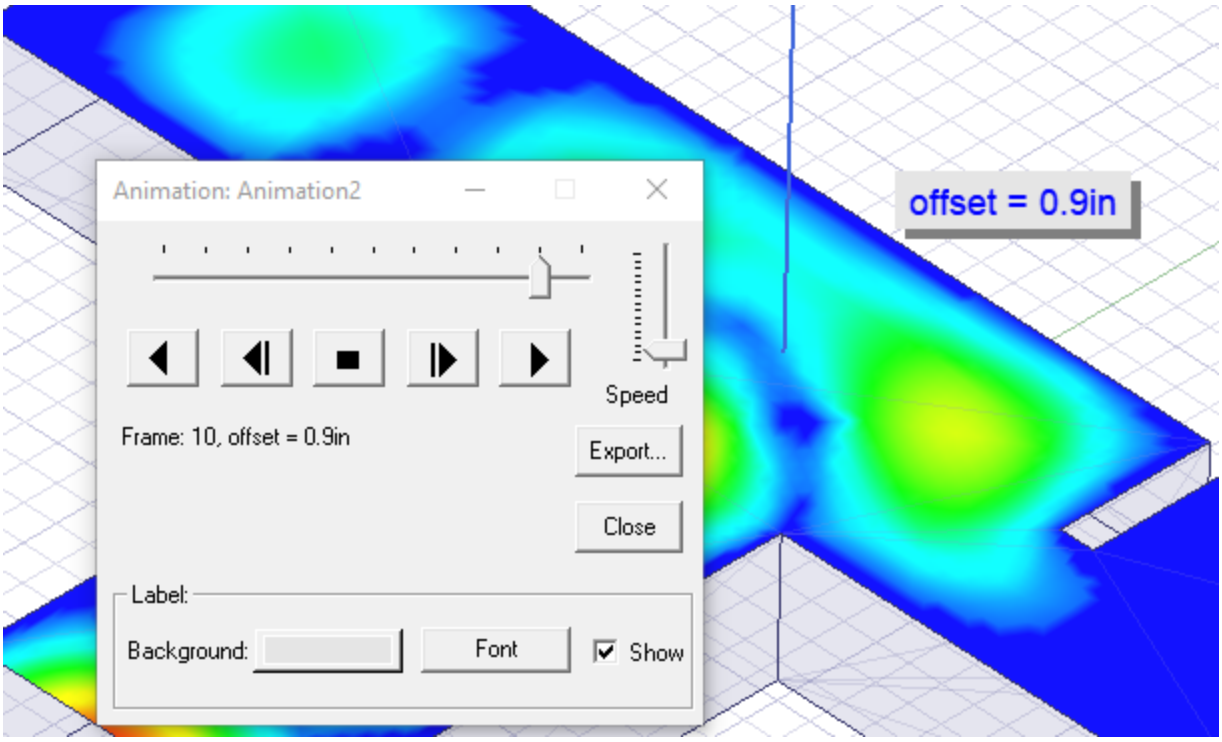
   The animation begins in the view window. It will display one frame for each variable value.

   The **Animation** control panel lets you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While a parametric animation is running, you can also turn on other field plots.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Mesh Animations

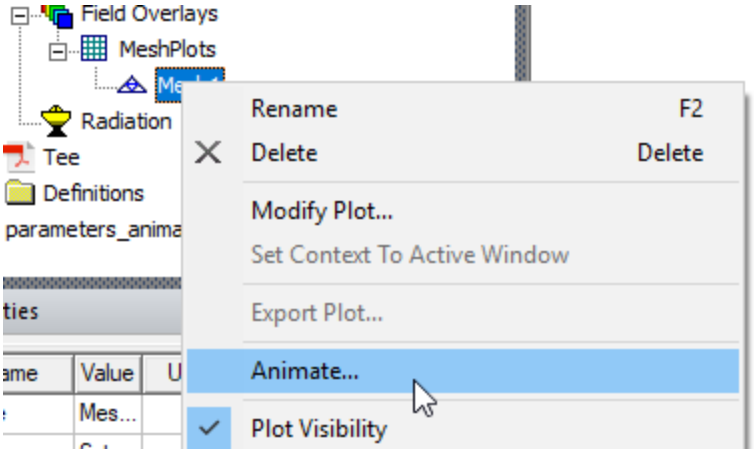
You can create mesh animations to evaluate the effect of single variables or parametric sweep variables on the model. Following is the general procedure for creating a mesh animation that varies a part of the model geometry.

Prerequisites:

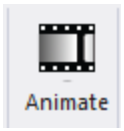
- You must define at least one variable associated with the geometry before you create a mesh animation.
- You must create a Mesh plot.

Procedure:

1. Select the Mesh plot in the Project Manager, right-click in the *Modeler* window, and click **View**> **Animate...**, or right-click the Mesh plot, and click **Animate...**



   Select the **View** tab of the ribbon, and click the **Animate** icon.



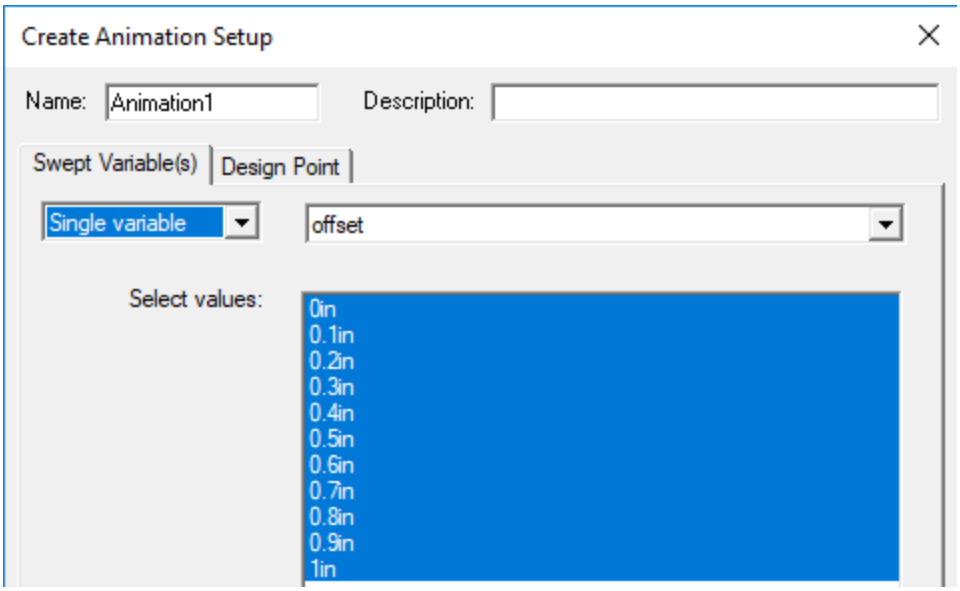
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

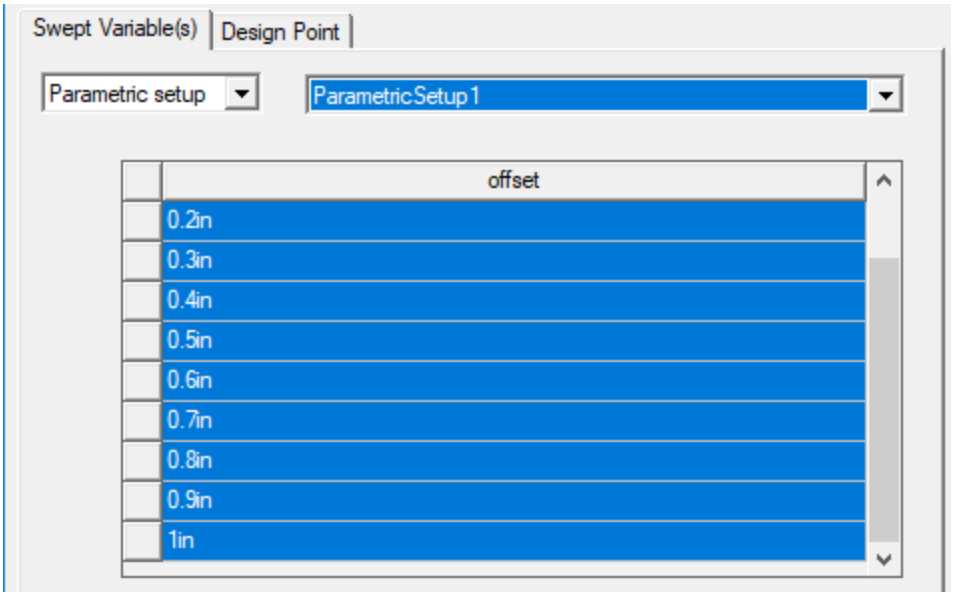
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. If no existing animation setup is acceptable, select **New** and continue below.

The *Create Animation Setup* dialog box appears.

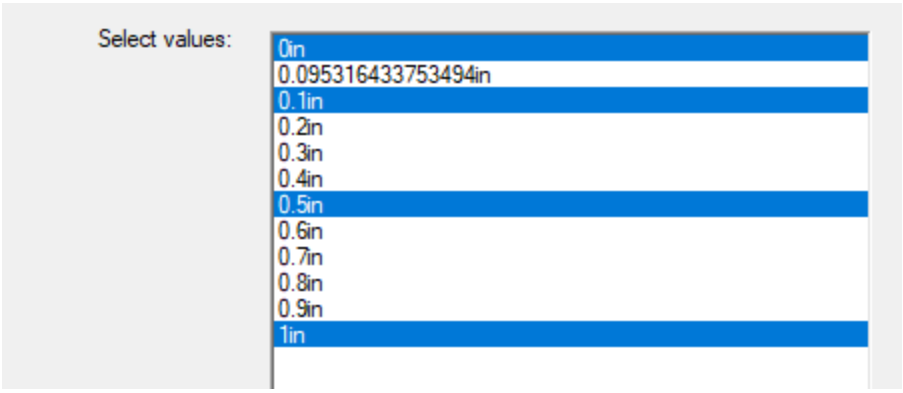


3. In the *Create Animation Setup* dialog box:
   a. Select the Single variable, or Parametric setup, or DOE setup if available. If multiple single variables are available, you can select from a drop-down menu. If one or more Parametric setups exist, you can choose from the drop-down menu. The values for the selected parametric setup are displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. If desired, specify the values to include in the animation. Selected values are highlighted.

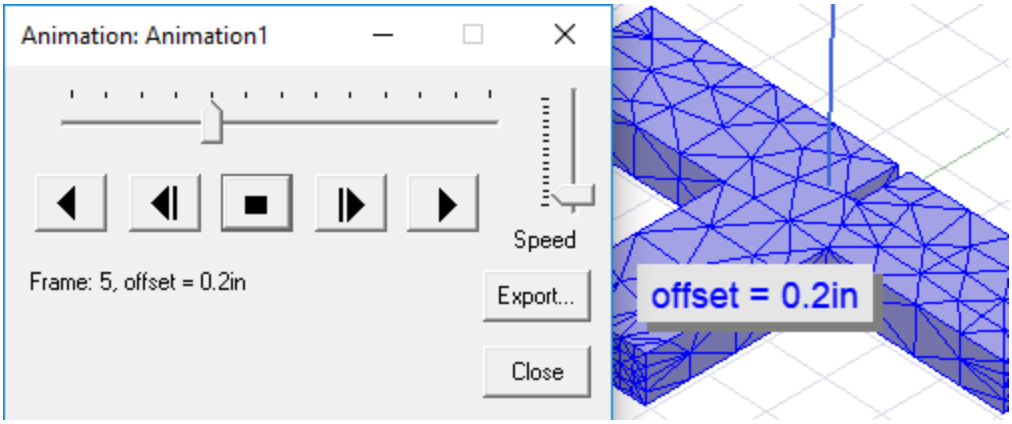


c. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.

4. Click **OK**.

The animation begins in the view window. It will display one frame for each variable value.

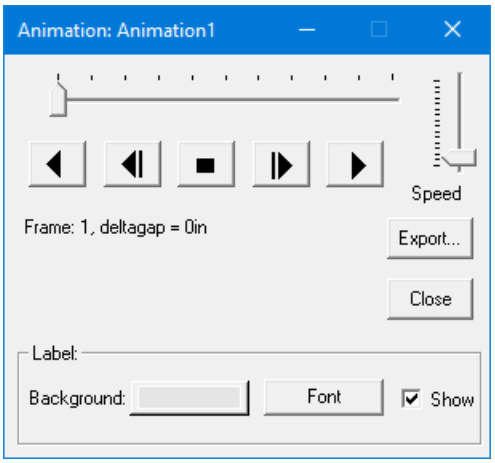
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Animation* control panel lets you to stop, restart, and control the speed and sequence of the frames.

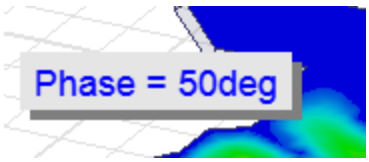
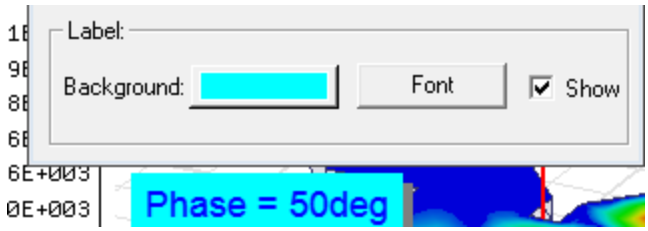
Controlling the Animation's Display

When an animation is displayed in the view window, the **Animation** window (also called the *play panel*) appears in the upper-left corner of the desktop. It has buttons that enable you to control the speed and sequence of the frames, start and stop the animation, and export the animation. Click an area of the window image below to learn its function.



Animation slider	Each dot on the slider represents a frame in the animation. Drag the slider to the right to display the next frame in the animated plot. Drag the slider to the left to display the previous frame in the animation.
	Plays the plot's animation sequence backwards.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

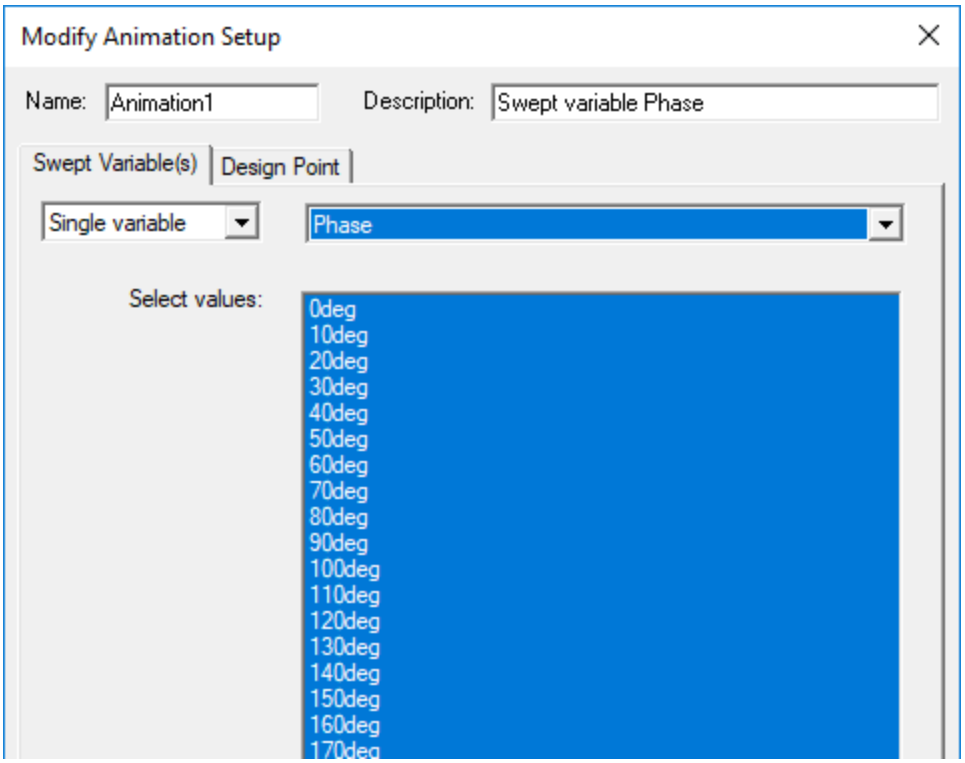
	Steps backward through the animated plot one frame at a time.
	Stops the animation.
	Steps forward through the animated plot one frame at a time.
	Plays the plot's animation sequence forwards.
	Drag the **Speed** slider to the top to increase the speed of the animation. Drag the **Speed** slider to the bottom to decrease its speed.
Frame information	The current frame and phase at which the plot is being displayed is listed below the control buttons.
	Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.
	Closes the animation window.
Show label check box	If you select the Show check box, a label showing the swept variable value appears in the animation. You can select the label with the mouse and drag it to a location. 
Background button.	Click the background button to open a color pallet dialog that lets you set the background for the swept variable label. 
	The font button opens a font selection dialog that you can use to set the Font, Font Style, and Size for the label. The Default is Arial Narrow 14pt.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying an Animation Setup

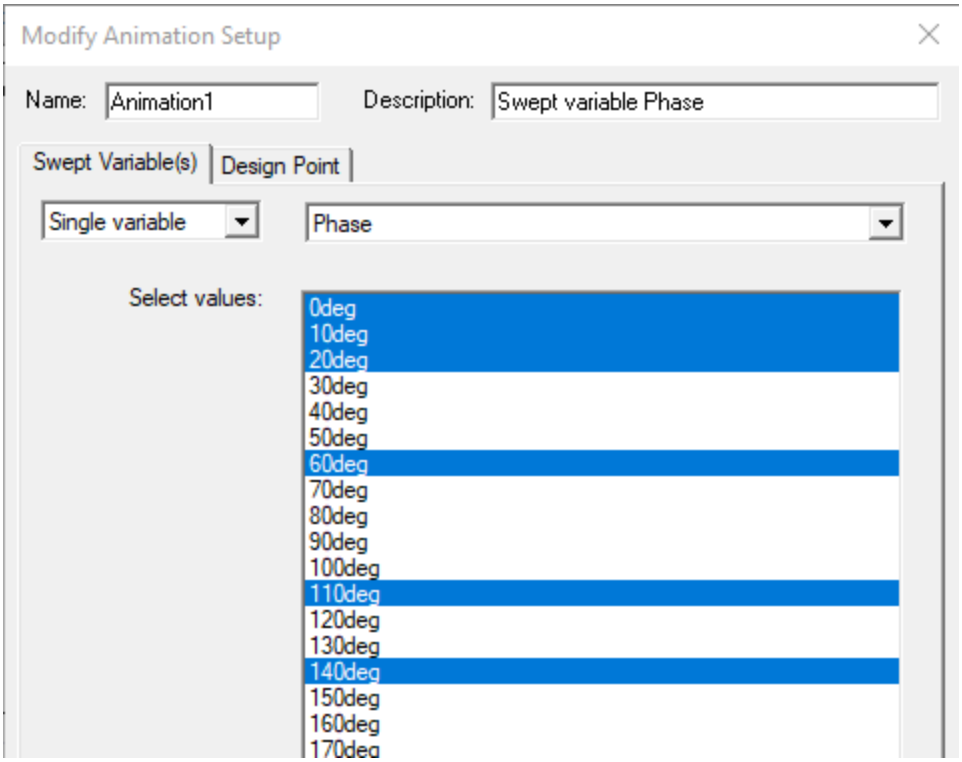
The **Edit** button on the *Select Animation* dialog box opens the *Modify Animation Setup* dialog box.

It opens showing the selections for variables and the selected values.

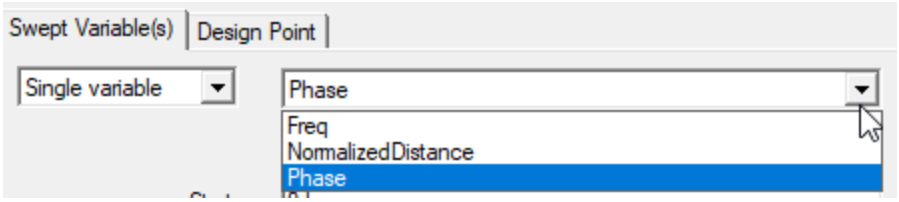


You can de-select or select any values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

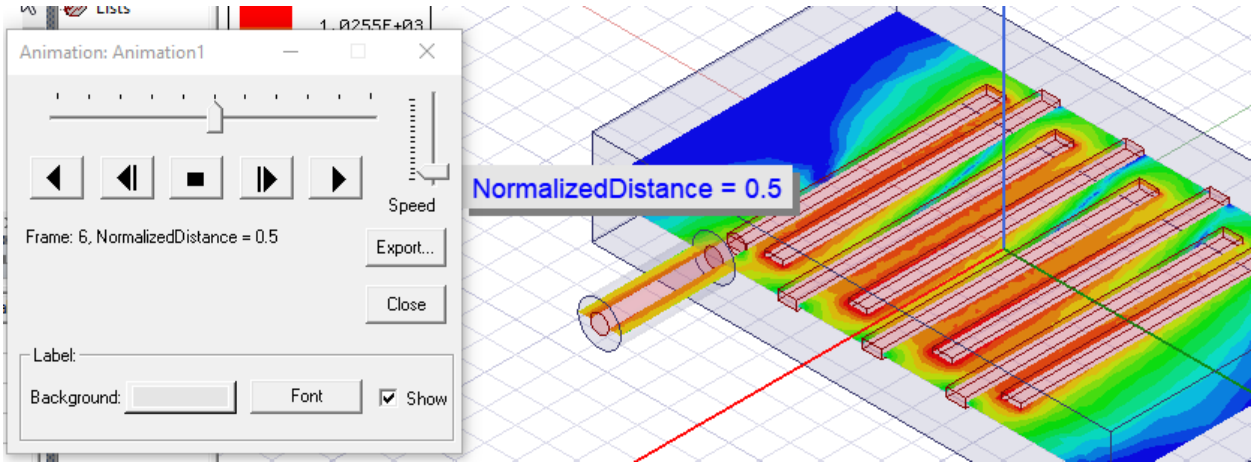


If other variables have been defined, you can select from the drop down list.

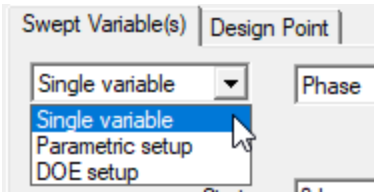


If the other variables have or can be set to appropriate values, you can modify the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If Parametric Setups or DOE Setups are available, you can select these.



Creating Animations

Controlling the Animation's Display

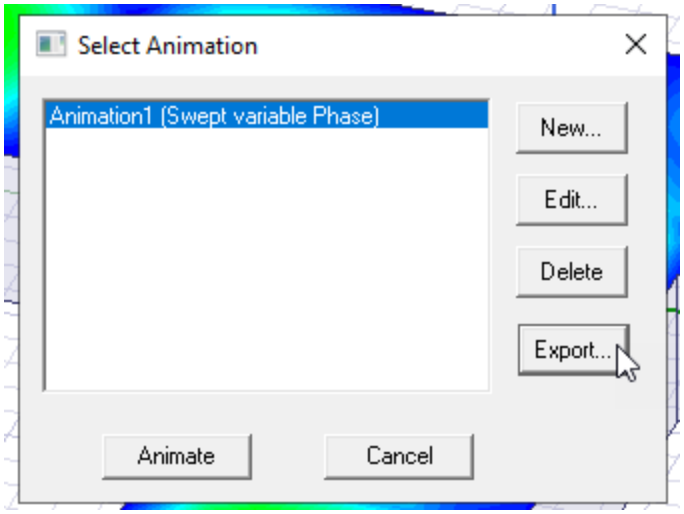
Exporting Animations

You can export animations from the **Animation** control dialog, or, for Ensight format, from the **Select Animation** dialog.

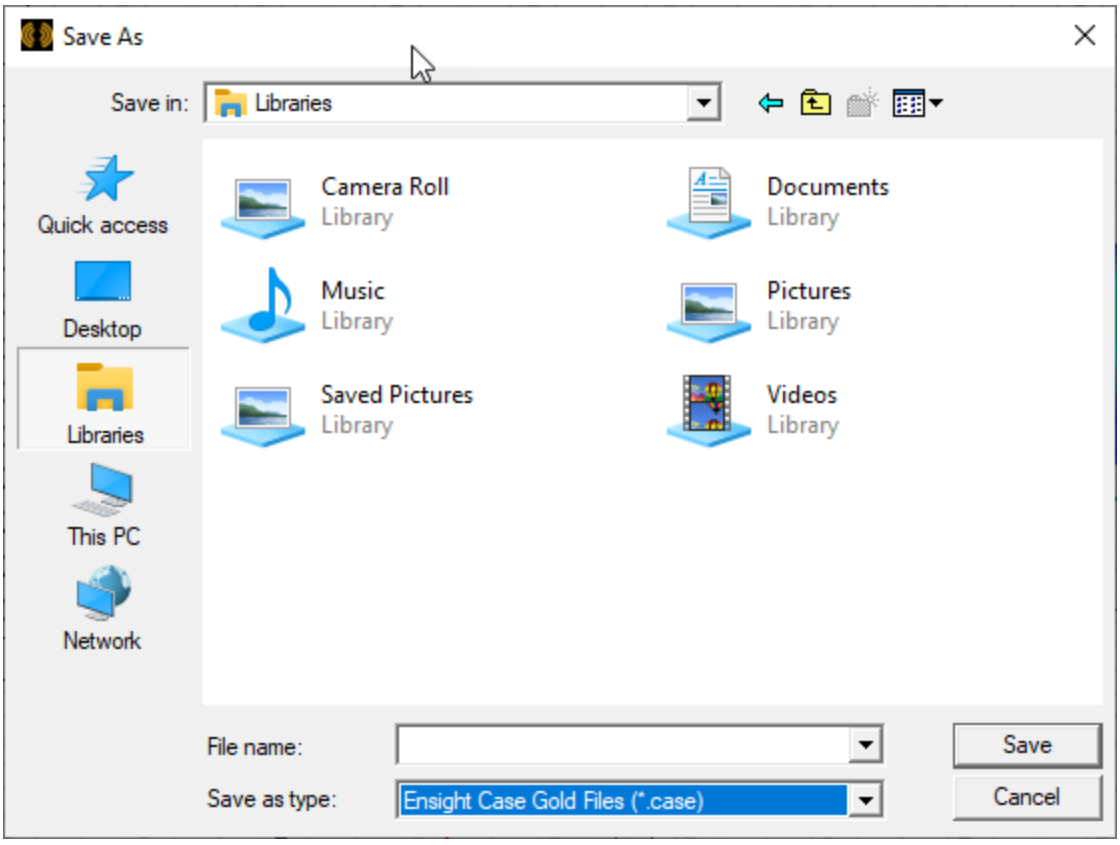
Exporting Ensight from the Select Animation Dialog

1. Create the animation you want to export.
2. In the **Select Animation** dialog, select **Export...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



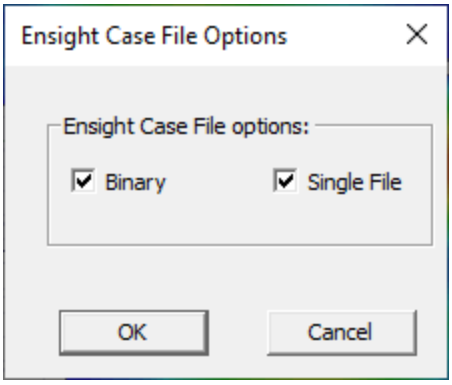
3. This opens the Browser window.



Navigate to the desired location and provide a file name. The format will be Ensight .case files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

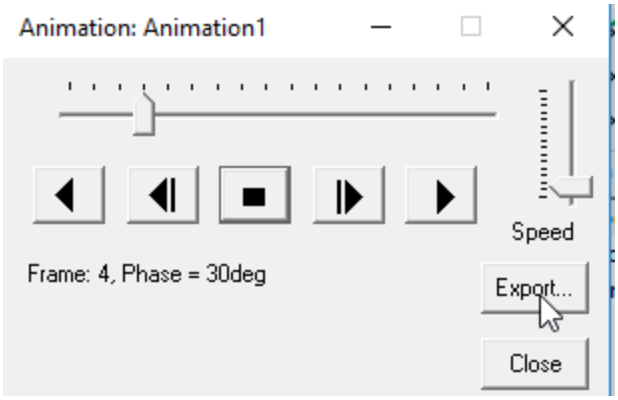
4. When you select **Save**, the **Ensight Case File Options** dialog opens.



The binary option permits a more compact and efficient animation creation process in Ensight. The Single File options leverages the Ensight transient dataset capability to export all frames of field/geometry data to a single case file. On **OK**, every visible and animatable field plot should generate a case file with name in the format of "UserTypedName_fieldPlotName.case"

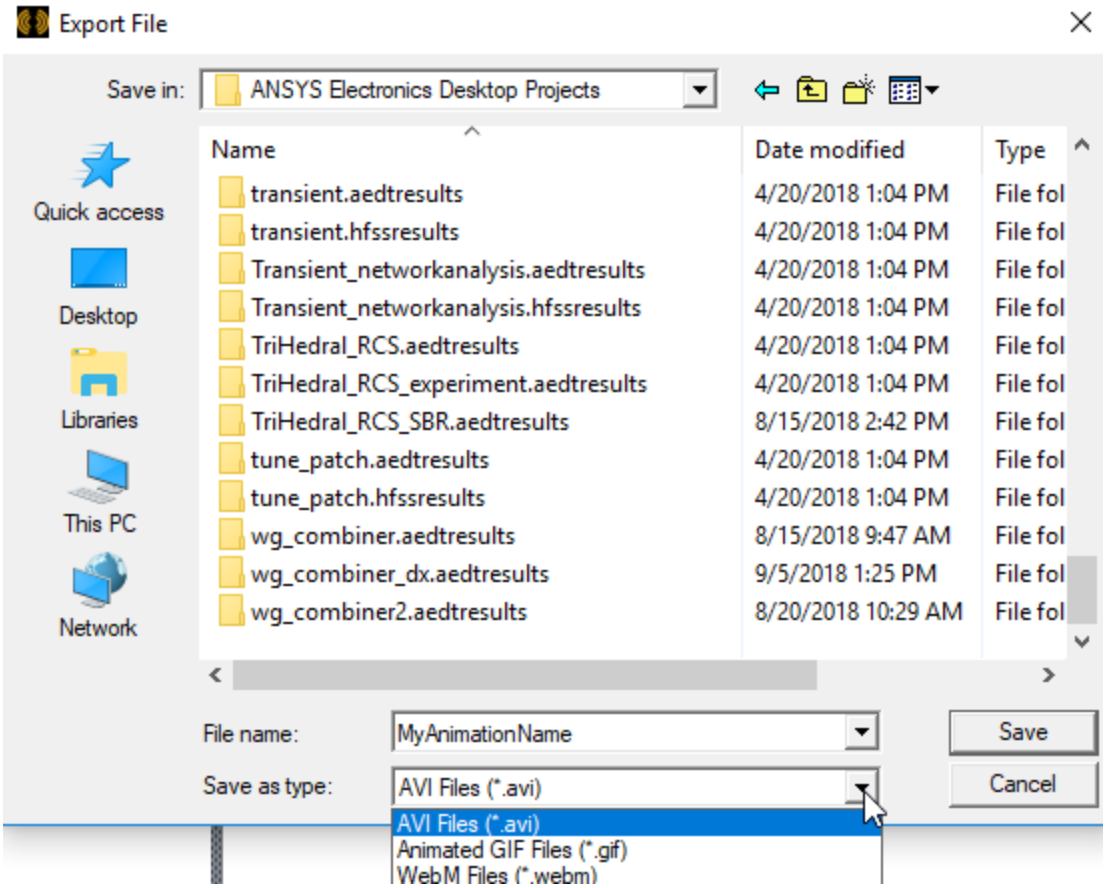
Exporting from the Animation Control Dialog

1. Create the animation you want to export.
2. In the play panel, click **Export...**.



The *Export File* dialog box appears.

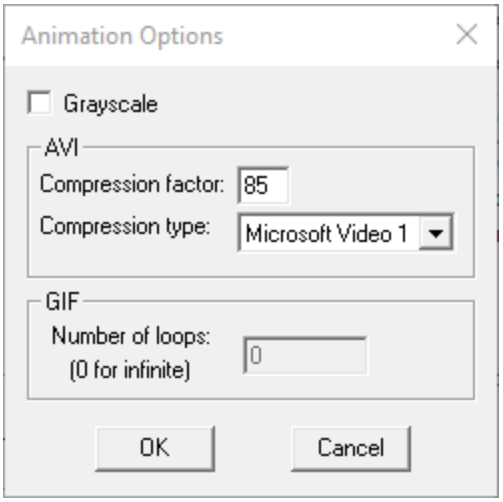
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify the directory to **Save in**, the **File name**, and use the **Save as type** drop-down menu to select **Animated GIF File (.gif), AVI File (.avi),** or **WebM File (.webm)**.

   The *Animation Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. To replace colors in the file with 256 shades of gray, select **Grayscale**.

   Grayscale animations tend to use less memory than full color animations.

5. For AVI format export, specify the **Compression factor** (the default is 85) and one of the following **Compression types**:
   - INTEL Indeo
   - Cinepak
   - Microsoft Video 1
   - None
6. For GIF format export, specify the number of loops. The default "0" denotes infinite loops.
7. Click **OK** to close the *Animation Options* dialog box.

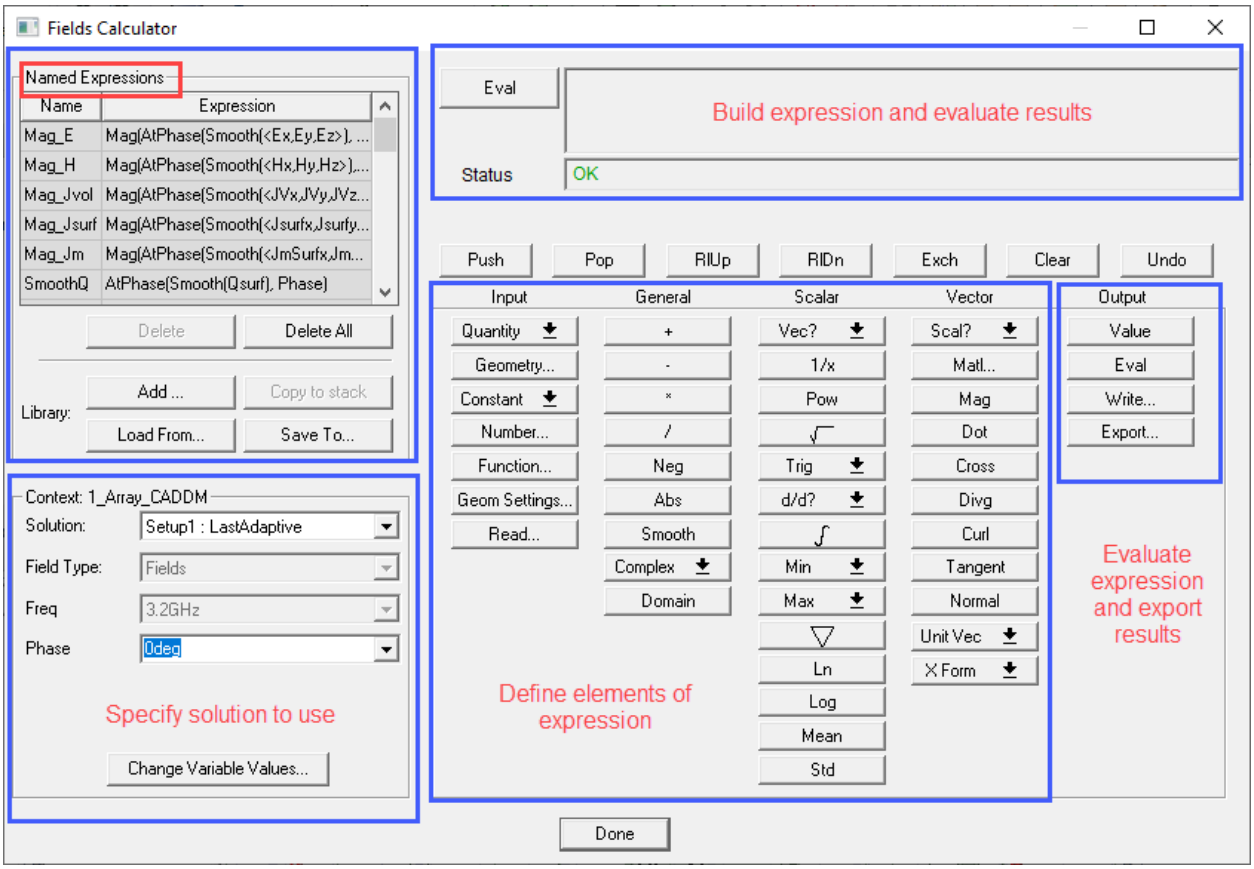
   The animation is exported to the file format you specified.

Using the Fields Calculator

The Fields Calculator is a very powerful tool for post processing. It is available for HFSS, Maxwell, Q3D, Icepak, and Mechanical solutions. While standard post -processed results (such as S-parameters, Y or Z matrix, animated field plots, and near and far field patterns) serve most simulation requirements, the Fields Calculator enables you to perform further computations using basic field quantities. The calculator computes derived quantities from the general electric, thermal, or displacement field solution; writes field quantities to files; locates maximum and minimum field values; and performs other operations on the field solution. Using this calculator, you can perform mathematical operations on all saved field data in the modeled geometry at a single frequency. The resulting quantities can be plotted, tabulated, or exported.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Fields Calculator includes predefined expressions appropriate for each solver and lets you create and save additional named expressions.



At the top-left corner of the calculator is a list of **Named Expressions**, which are standard or user-defined field quantities that are accessible from outside of the calculator. They can be added, copied to stack, saved to, or loaded from a library file using the buttons right beneath the list.

At the lower-left corner of the calculator is the **Solution Context** section, in which you can select the desired solutions, field types, frequency, and phase for the current session.

The top right of the calculator contains the **Data Stack**, in which calculator entries are held in stack registers. The data type in the Data Stack is denoted by its prefix abbreviation.

Immediately beneath the stack is the row of **Stack Command** buttons that define some basic operations for the data in the Data Stack.

The bottom half of the calculator holds the columns containing the actual calculator buttons, organized into columns, classifying them by the type of operation and the type of data upon which the operation can be performed. These columns are headed Input, General, Scalar,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vector, and Output. At the very bottom of the calculator is the button to exit, **Done**. The Fields Calculator interface is discussed in detail in subsequent sections.

The calculator does not perform the computations until a value is needed or is forced for a result. This makes it more efficient, saving computing resources and time; you can do all the calculations without regard to data storage of all the calculated points of the field. It is generally easier to do all the calculations first, then plot the results.

An algebraic expression is either a `namedExpression`, a `constant`, a `name`, an `operator`, two `expressions` separated by `binaryOperator`, or an `expression` enclosed in parenthesis.

An `operator` is an *opName* followed by `arguments`.

An `argument` is either empty parenthesis, an `expression` enclosed by parenthesis, or two `expressions` separated by a comma and enclosed by parenthesis.

A `binaryOperator` is one of the following literals: '+', '-', '*', '/'

A `name` is either a project, design, or intrinsic variable (scalar or vector), or a geometry.

A *constant* can be one of the following:

- scalar literal; for example, "1.5e-10"
- complex number; for example, "3.5-1.0i" or "<(3.5, -1.0)>"
- 3D vector; for example, "<1.0, 2.0, 3.0>"
- complex 3D vector; for example, "<1.0-2.0i, 2.0-3.0i, 3.0-4.0i>" or "<(1.0, -2.0), (2.0, -3.0) ,(3.0, -4.0)>"

The grammar for an algebraic expression is summarized below:

```
expression : namedExpression
   | constant
   | name
   | operator
   | expression binaryOperator expression
   | '(' expression ')'
operator : opName arguments
binaryOperator : + | - | * | /
arguments : '(' <empty> ')'
   | '(' expression ')'
   | '(' expression ',' expression ')'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting Support

All calculator operations are fully scriptable. You can save the commands used in a Fields Calculator session by first clicking the **Tools** > **Record Script to File** menu and replay the same commands by clicking the **Tools** > **Run Script** menu in a later session.

Preliminary Considerations for Using the Fields Calculator

The following sections provides some consideration notes regarding use of the Post-Processor Field Calculator. Most of the statements below are fairly generalized, and may not apply to all 3D solver projects. When in doubt about the applicability of a particular consideration for a particular project, please feel free to contact your local HF Applications Engineer for further assistance.

In order to obtain high accuracy results from calculations on field data, we recommend that you take extra precautions to assure that the model's field data is dependable. This might include:

- Seeding or manually refining the mesh in the areas to be used for calculations.
- Running parametric variations to isolate sensitivity to modeling parameters (such as adaptation frequency or circular cross-section facetization).

As long as the accuracy of specific field data points to be used has been assured, the results of the Field Calculator operations should provide valuable information for your electromagnetic design tasks.

Opening the Fields Calculator

To open the Fields Calculator, from the Desktop Menu bar do one of the following:

- Click **Mechanical**> **Fields**> **Calculator**

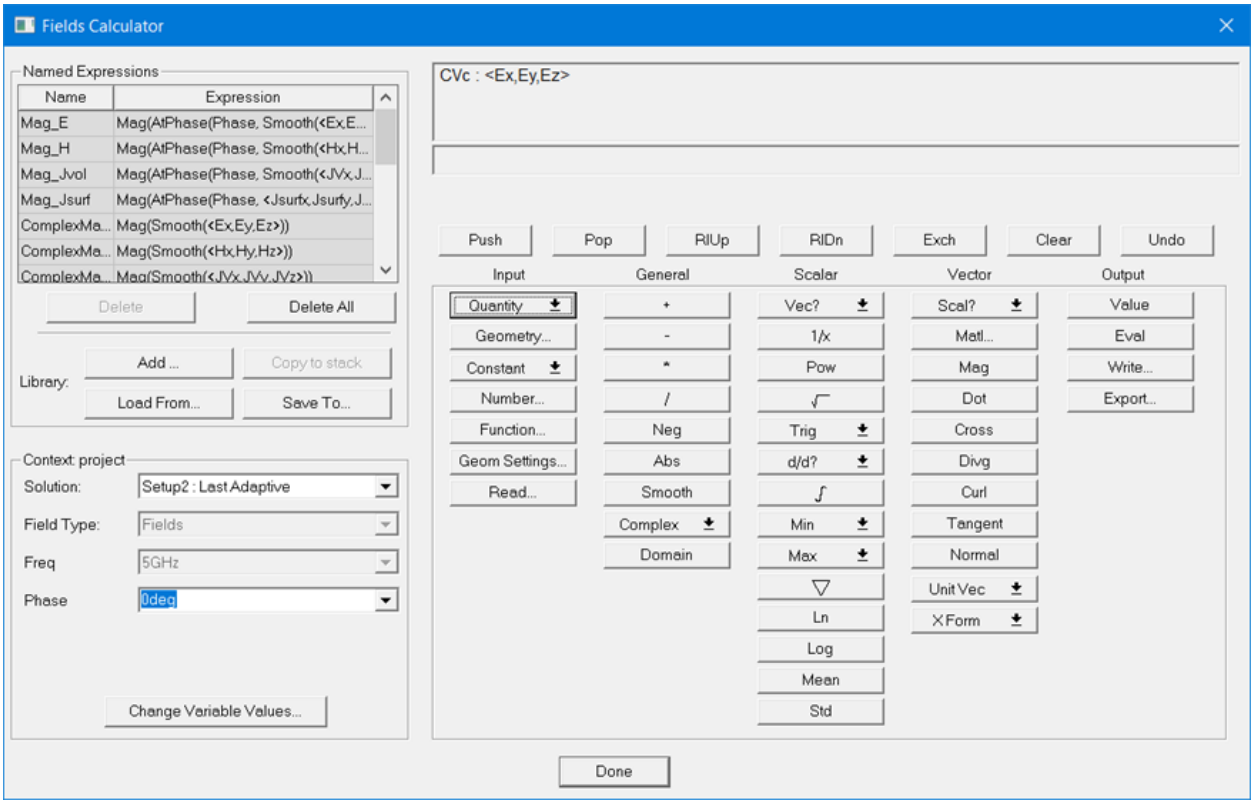
  or

- Right-click **Field Overlays** in the Project Manager and choose **Calculator** from the shortcut menu.

The *Fields Calculator* window appears.

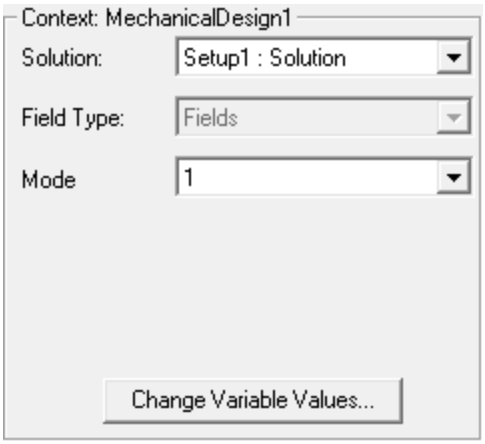
To view information on a command or screen area, click over the button or screen area on the illustration below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Context Area

The panel at the upper right of the window identifies the context to be used for the calculations. The top line identifies the design. Choose the solution setup on which to base the calculation from the **Solutions** drop-down menu. For Modal solutions, choose the vibration mode from the **Mode** drop-down menu.



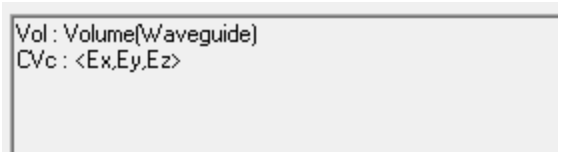
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Field Type here is not related to the edit sources. This is a general term among Ansys EM products (HFSS, Maxwell, Q3D, Icepak, and Mechanical). Some products have more than one field type for different solution types. If only one Field Type is available the drop-down menu is unavailable.

The **Change Variable Values** button opens a *Set Variable Values* dialog box. By default, it has *Use Nominal Design* checked. Unchecking the box lets you select another variable value. **OK** the dialog box to accept the selection.

The Calculator Stack

The calculator is made up of a stack of registers. Registers are displayed in the register display area at the center of the calculator window.

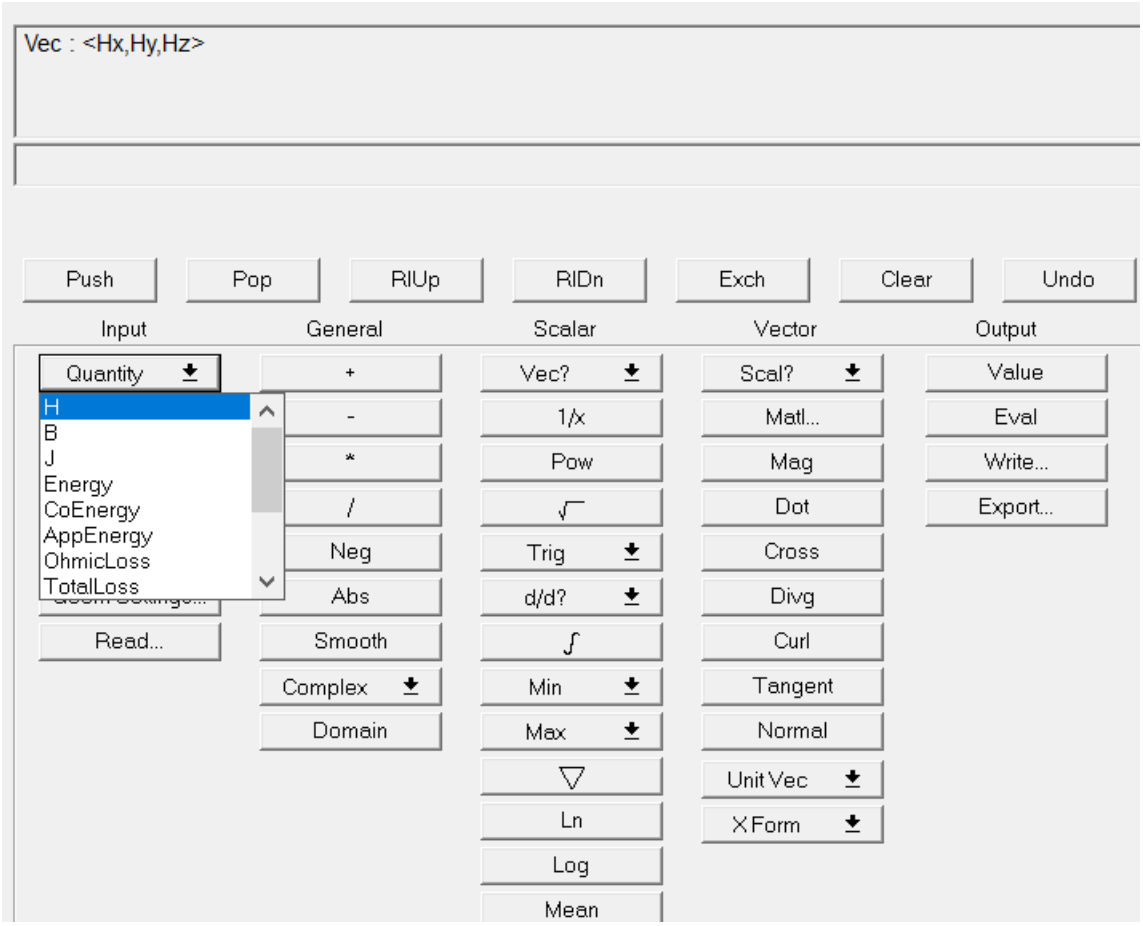


Each register can hold

- field quantities such as the H-field or E-field.
- functional or constant scalars and vectors.
- geometries — points, lines, surfaces, or volumes — on which a field quantity is to be evaluated.

To perform a computation on the field solution, you must first load a basic field quantity into a register on the stack:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once a quantity is loaded into a register, it can be

- manipulated using mathematical operations such as curls, gradients, cross products, divergences, and dot products.
- integrated over lines, surfaces, or subvolumes of the solution region — either predefined surfaces, volumes, and lists, or lines, surfaces, and volumes that were defined using the **Draw** commands.
- exported to a file, allowing you to superimpose saved solutions.

Stack Commands

Stack Commands influence the entries in the calculator stack and their position. These commands are for manipulating stack contents only, such as copy/paste, delete, or rearrange. The names for these commands match standard stack manipulation conventions. Many are self explanatory, and some can be used in pairs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Push

Reloads the quantity in the top register onto the top of the stack, creating a new register. The contents of the top two registers are identical.

Pop

Deletes the top register from the stack.

RlUp

Rolls the top register to the bottom of the stack, moving the other registers up the stack.

RlDn

Rolls the bottom register to the top of the stack, moving the other registers down the stack.

Exch

Exchanges the top two registers in the stack.

Clear

Clears the contents of the stack.

Undo

Use this command to undo the effect of the last operation you performed on the contents of the top register. Successive **Undo** commands act on any previous operations.

> **Note:**
>
> You cannot undo a simple operation such as loading a field quantity, constant, function, or geometry into the calculator. Instead, use the **Pop** or **Clear** commands to delete these items from the calculator stack.

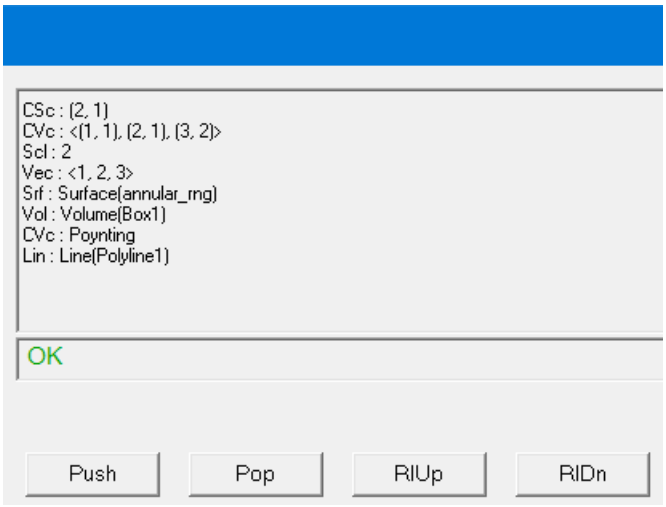
Registers

Calculator registers hold field quantities, numbers, vectors, and geometries. No registers are created until you load something into the calculator; therefore, this part of the window is initially blank. As items are loaded into the calculator, it creates new registers to hold them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For Maxwell and HFSS users, make sure the Calculator is in **Show Stack** mode.

The calculator is capable of performing operations on a number of different data types. In many instances, a calculation requires certain type(s) of data to be present in the correct order in the stack register. Many operations result in a different data type than the inputs. In order to show you the type of data contained in each stack entry, calculator denotes its data type by a prefix indicator as shown in Stack Contents showing Data Type Indicators (at left), below.



The following table describes and defines each indicator, and provides guidance regarding operations which can convert data from one type to another.

Vec	Vector (non-complex) quantities, which have both direction and magnitude at each point in space. The x-, y-, and z-components of these quantities are stored in the register. Vectors are always evaluated in the coordinate system of the model. To convert a vector quantity to a scalar, use the Scal? drop-down menu from the Vector column. Suboptions ScalarX, ScalarY, and ScalarZ will take the appropriate scalar component of the vector data. Optionally, you can also Dot the vector with another vector to obtain the appropriate scalar result, or use the Tangent (return the tangential scalar component of) or Normal (return the normal scalar component of) operations to relate the vector quantity to a geometric data (Lin, Srf) stack entry. Convert to a Complex quantity using the CmplxR and CmplxI operations described in Scl, above.
Scl	Scalar quantities, which have a magnitude only. This is a simple numerical value. To convert a scalar to a vector quantity, use the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Vec? drop-down menu in the Scalar column. The choices VecX, VecY, and VecZ convert the scalar data to vector data aligned with the X, Y, or Z unit vectors, respectively. You can also multiply the scalar quantity by a desired vector direction entered manually (Num drop-down in the Input column) or obtained using the Unit Vec button from the Vector column. To convert a scalar to a complex quantity, use either CmplxR (assign the scalar value as the real component of a complex quantity) or CmplxI (assign the scalar value as the imaginary component of a complex quantity), both found under the Cmplx drop-down in the General Column.
Cvc	Complex vector quantities. This is a numerical value with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx dropdown in the General column.
Csc	Complex scalar quantities. This is a numerical value with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx drop-down in the General column.
Pnt	Points.
Lin	Lines. Lines may be straight, curved, or “polylines” in three dimensional space. Lines may also be open (have two endpoints) or closed (ending vertex same as starting vertex).
Srf	Surfaces. Surfaces need not be planar, and may actually comprise a list of object faces (faces list) as well as planar slices through the entire model space (cutplanes).
Vol	Volumes. Volumes may include sets of discontinuous object volumes created as an Object.
SclLin	Scalar value on a line.
VecLine	Vector value on a line.
SclSrf	Scalar value on a surface.
VecSrf	Vector value on a surface.
List	These indicators may exist alone, representing geometric data only, or in combination with one of the categories above, indicating a type of data applied to the geometric entity in question. For example, the notation SclSrf identifies a stack entry containing Scalar data on a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Surface geometry set. To select only the portion of a given data entry which exists along, on, or within a given geometry quantity, use the Value button in the Output column of the calculator. Other operations (e.g., integration, the Normal button) operate when a data quantity is in the second stack register and a geometric quantity is in the top stack register. Full descriptions of the register requirements for each individual command is available in the online help.

When examining calculator registers, keep the following in mind:

- To move or delete calculator registers, use the stack commands.
- To save a register to a disk file, use the **Write** command.

Users must be cautious concerning what type of data the Fields Calculator is manipulating and whether or not it is compatible with the desired operation. For example, the integral operation is often misused. Note that the integral sign is in the *Scalar* column, implying that, to integrate complex number/quantities, the user will have to integrate the real and imaginary components separately. In other words, performing integration on complex number/quantities must be achieved by parts.

Enlarging the Register Display Area

If there are too many registers to fit into the display area, do one of the following:

- Use the scroll bars to view the hidden registers.
- Enlarge the calculator window by dragging window's borders.

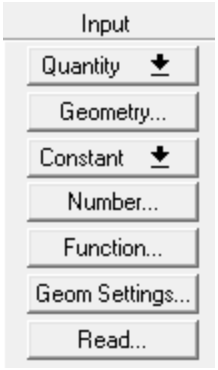
Units of Measure

Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

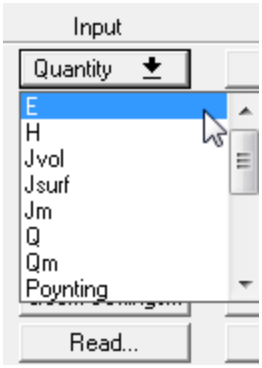
Input Commands

The *Input* Column contains all the calculator functions which place new values into the stack, such as field data, constant, geometry data, coordinate system selection, or user-entered vector or complex numbers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Field data (e.g., E-field, H-field, Poynting vector) for the current project solution is input from the **Quantity** drop-down menu selection.



The quantities specifically available from the calculator are dependent on the solution type and include the E-field, H-field, J-vol (volume current), Poynting vector, Displacement, Temperature, Heat Flux, and Surface and Volume Loss Densities. All quantities are Peak Phasors, and not RMS quantities, with the phase information captured in the real and imaginary components and the field orientation captured in the vector components. Although the Poynting vector is automatically calculated by the interface as 0.5, it will appear in the calculator stack as a Complex Vector quantity. The imaginary portion should however be zero or ignored.

Use the following commands to load data onto the top of the calculator stack:

Quantity	Basic field quantities, such as E and H, and simple derived quantities such as volume current.
Geometry	Geometries such as planes, points, polylines, 3D lines, face lists, and volumes
Constant	Predefined constants such as π, ε_0, and conversion factors between various units of measurement.
Number	Vector and scalar constants, including complex

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	numbers.
Function	User-defined or intrinsic variables
Geom Settings	Number of equally spaced points used to integrate fields and other quantities on a line.
Read	previously saved calculator registers containing field quantities.

These quantities can be manipulated using the **Stack** commands, **General** commands, **Scalar** commands, and **Vector** commands. The results of these calculations can then be examined using the **Output** commands.

Quantity Command

The **Quantity** command loads a specified field quantity into the top register of the calculator.

The available quantities are:

Temp	The resultant scalar temperature result, available from Thermal solutions.
Heat_Flux	A derived scalar quantity indicating the rate of heat flow on a per unit area basis. It is computed from the temperature field for Thermal solutions.
Surface_Loss_Density	This quantity concerns thermal excitations imported from the surfaces in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Volume_Loss_Density	This quantity concerns thermal excitations imported from the volumes in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Displacement	The displacement vector results from modal solutions. The displacement direction and magnitude at one point

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	on the model relative to another point demonstrates the shape of the vibration mode. However, the absolute magnitudes of the displacement are not meaningful because they are not scaled to any specific excitation. Rather, the displacement magnitudes are arbitrary for natural frequency (modal) solutions.

Geometry Command

The Fields Calculator **Geometry** command opens a dialog box for selecting a geometry to load into the top register of the calculator.

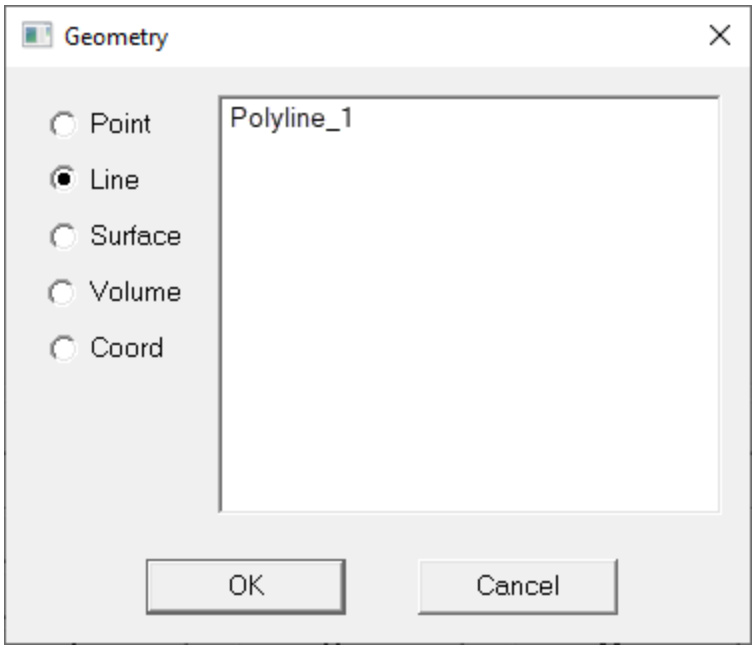
Do this to

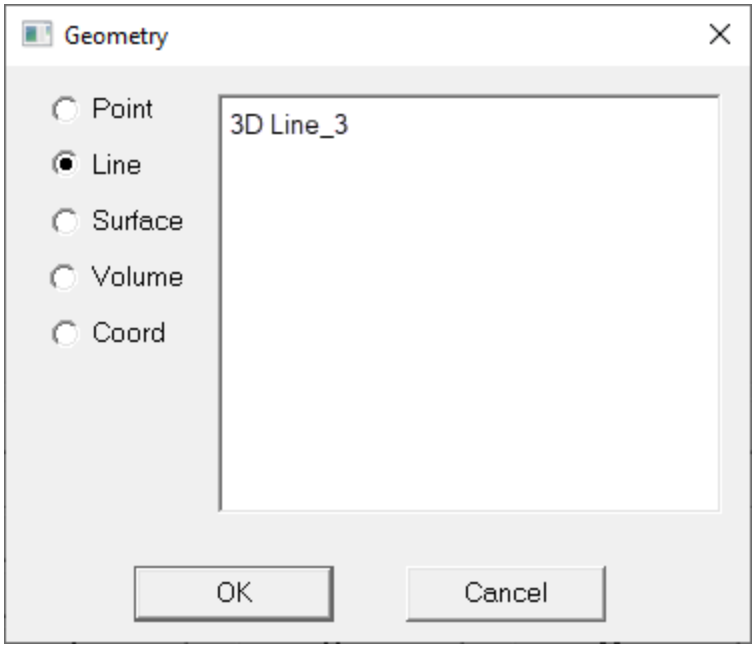
- find the value of derived field quantities on any point, line, surface, or volume.
- plot quantities directly from the calculator.
- display a previously defined isosurface, maximum or minimum field point using the **Draw** command.

The following types of geometries are available:

Point	See Drawing a Point Object. Drawn points are listed in the History Tree, and in the calculator's *Geometry* dialog box when **Point** is selected.
Line	See Drawing a Line Segment or Drawing a 3D Line Segment. Drawn lines are listed in the **History Tree**, and in the calculator's *Geometry* dialog box when **Line** is selected. To set the number of points on a line, see Geom Settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



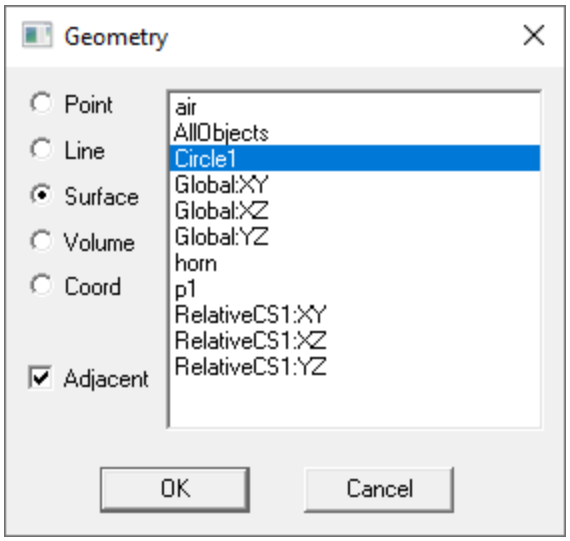


Surface

Creatable **Sheet** objects and Face Lists are listed in the History Tree and in the calculator's **Geometry** dialog box when **Surface** is selected.

Due to the ambiguity of the normal vector of a sheet, the result may require a multiplication by (1) or (-1). If a Sheet object is selected from the Face list, the dialog also lists an **Adjacent** command that controls the side of the sheet on which the fields are calculated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This check box is only available when users select a sheet object for surface geometry

- Allow operation on the adjacent side surface geometry just like regular surface geometry.
- All calculation on this geometry will be done using data from the adjacent side of the surface.
- Support scripting, saving and loading expression with adjacent side surface geometry.

```
Srf : AdjacentSurface(Circle1)
CVc : <Ex,Ey,Ez>
```

Volume 3D objects, Regions, and Object Lists (of 3D objects including AllObjects) are available in the calculator's *Geometry* dialog box when **Volume** is selected.

Coord Coordinate systems are available in the calculator's *Geometry* dialog box when **Coord** is selected.

To load a geometry into the calculator:

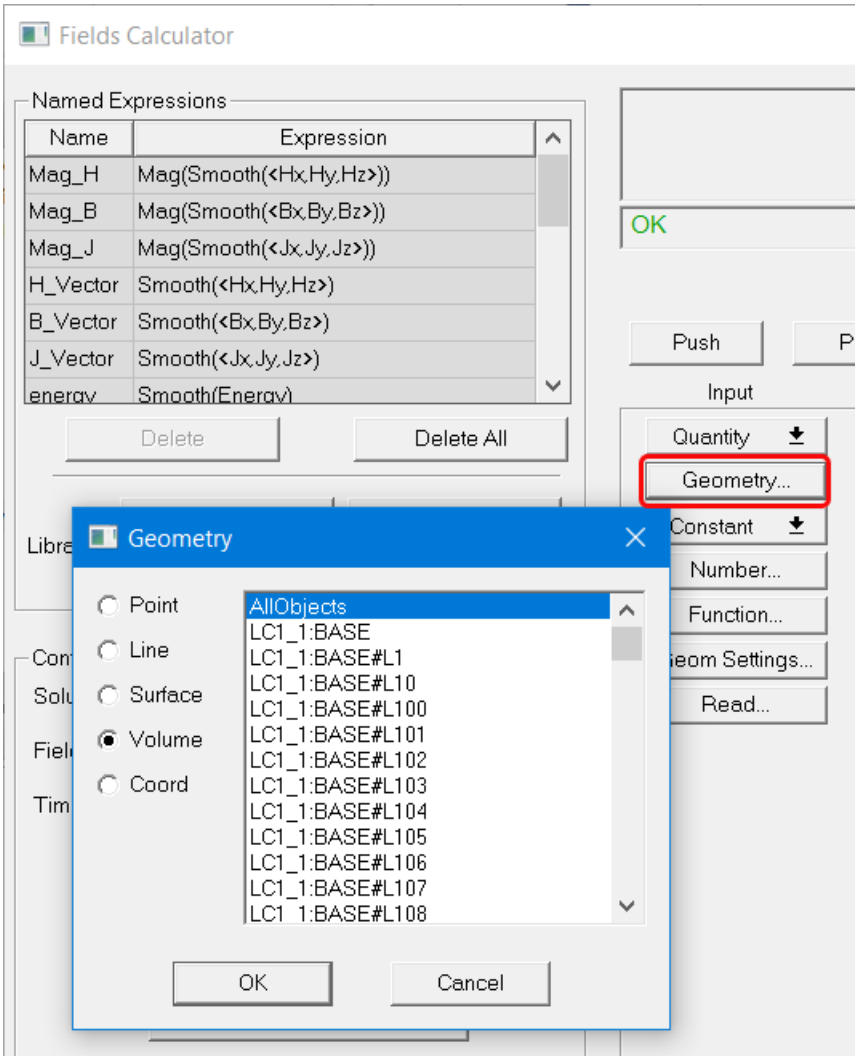
1. In the Fields Calculator, click **Geometry**.

   The **Geometry** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select a geometry type.

   A list of all applicable geometries appears. If a design includes Layout Components, the Geometry list shows the available component geometries as well.



In the modeler window, these Layout Component objects are drawn as visualizations without actual geometry. However, they can be used to calculate field quantities as shown below.

These geometry operators can be used to perform computations just like one constructed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

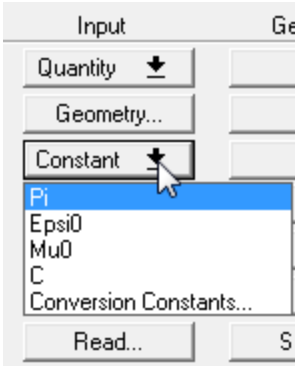
from regular 3D geometry. These additional objects are generated only after reading a mesh, so they won't show up if there is no solution.

3. Click the geometry.
4. Click **OK** to load the geometry.

Note: Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1. Box2 is actually implicitly subtracted from Box1 as is done in our solvers. So Box1 is used as if Box2 were already subtracted from Box1. Volume(Box1) is Box1 excluding Box2, and Surface (Box1) contains faces from both Box1 and Box2.

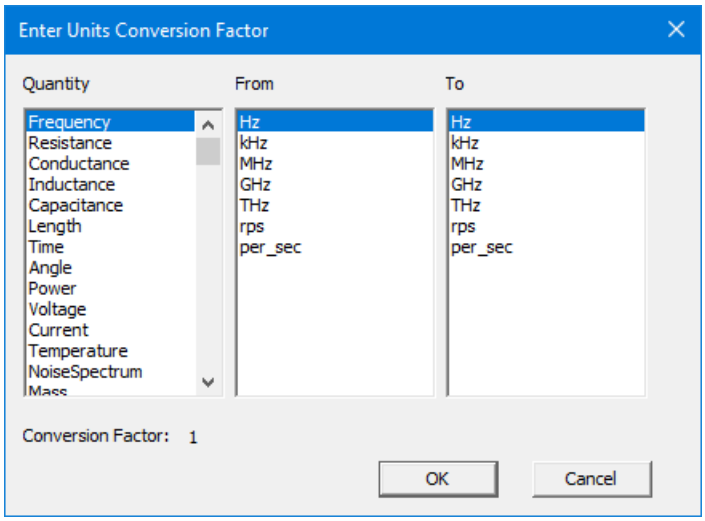
Constant Command

The **Constant** command loads one of the following four predefined constants or a selected conversion constant into the top register of the calculator:



Pi	π
Epsi0	The permittivity of free space, $\varepsilon_0 = 8.85418782 \times 10^{-12}$ C^2/Nm^2
Mu0	The permeability of free space, $\mu_0 = 4\pi \times 10^{-7}$ Wb/Am
c	The speed of light in vacuum, $c = 2.99792458 \times 10^8$ m/s
conversion constant	Displays the *Enter Units Conversion Factor* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lists a range of Quantities (such as frequency, resistance, and others) along with a list of Units (Hz to Thz, and rps) to convert From and To. The ratio of the Units From to the Units to is displayed for the selected values as **Conversion Factor**.

Number Command

The **Number** command enters one of the following into the top register of the calculator:

Scalar	A scalar constant. To enter a constant scalar number: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Scalar**. 3. Type the scalar value in the **Value** text box. 4. Click **OK** to load the number into the top register.
Vector	A vector constant. To enter a constant vector: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Vector**. 3. Enter the x-, y-, and z-components of the vector. 4. Click **OK** to load the vector into the top register.
Complex	A complex constant, entered in the form $C=A+jB$, where A represents the real part of the constant and B represents the imaginary part.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Number**.

   The *Input Number* dialog box appears.

2. Select **Scalar** or **Vector**.
3. Select **Complex**.
4. Enter the real and imaginary components of the number.
5. Click **OK** to load the number into the top register.

Function Command

The **Function** command enters a predefined scalar or vector function into the top register of the calculator:

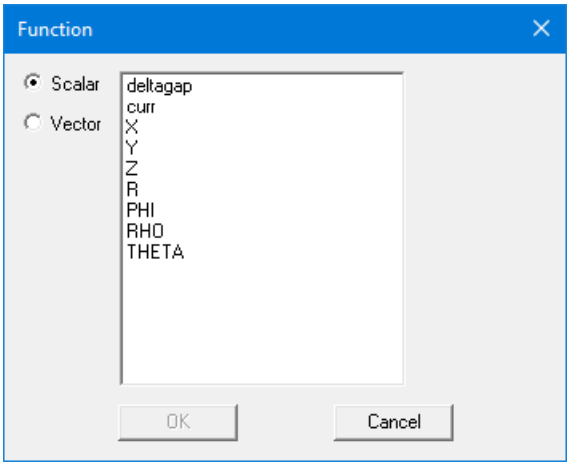
Scalar

A scalar function.

To enter a function:

1. Click **Function**.

   The *Function* dialog box appears.



2. Select **Scalar**.
3. Select the function from the list.
4. Click **OK** to load the functional scalar into the top register.

Vector

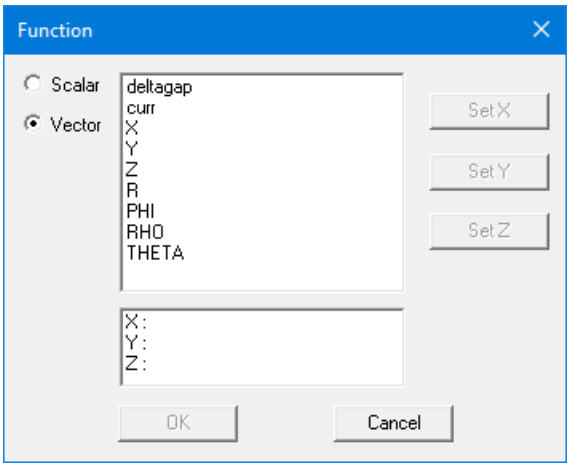
A vector function, in which the values of the vector's x-, y-, and z-components are given by functions.

To enter a functional vector:

1. Click **Function**.

   The *Function* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **Vector**.
3. Select the function from the list.
4. For each component of the vector, click **SetX, SetY,** and **SetZ**.
5. Click **OK** to load the functional vector into the top register.

Note:

The predefined variables **X, Y,** and **Z**; **RHO, THETA, R,** and **PHI**; and any functions that you created can be used to define functional scalar and vector quantities.

Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

Geom Settings Command

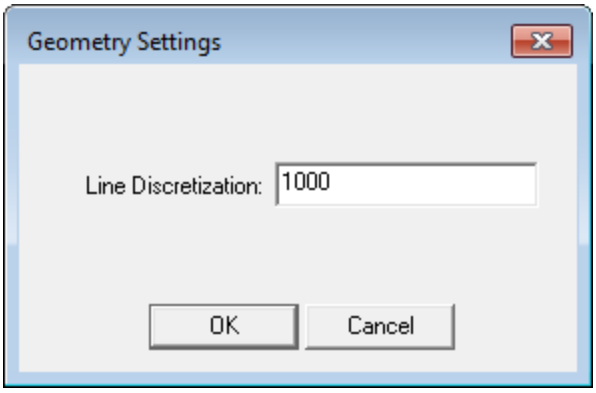
Clicking the **Geom Settings** button opens the *Geometry Settings* dialog box. This dialog box allows you to specify the line discretization, the number of equally spaced points used to integrate fields and other quantities on a line. The default is 1000 points.

To set the line spacing for geometry settings:

1. In the Fields Calculator, click **Geom Settings**.

   The *Geometry Settings* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter a value in the **Line Discretization** text box and click **OK**.

Read Command

This **Read** command copies the contents of a disk file into the top register. The register must be one that has been saved using the Write output command.

To read in a register:

1. Click **Read**.
2. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assumed for register files.
3. Click **OK**.

   The contents of the file are copied to the top register in the stack.

General Commands

The General Column contains calculator operations which can be performed on many different data types (e.g., vector, scalar, complex). With the exception of the Complex menu, all are distinct functions. Most are self-explanatory, with the exception of Smooth which performs some data “smoothing” or statistical averaging on the top stack entry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these Fields Calculator commands to perform operations on both vector and scalar quantities.

+ (Add)	/ (Divide)	Smooth
-- (Subtract)	Neg	Complex
* (Multiply)	Abs	Domain

+ (Add)

Adds the quantities in the top two registers of the calculator.

- (Subtract)

Subtracts the quantity in the top register from the quantity in the second register. The two registers must hold the same type of quantity (both scalar or both vector). You cannot subtract a scalar from a vector (or vice versa).

> **Note:** Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1.
>
> If you explicitly subtract Box2 from Box1, any calculation on the surface (faces) of Box1 will use the 6 exterior faces and the 6 interior faces. Any calculation on the volume of Box1 will use the difference in volume between Box1 and Box2.
>
> If you do not explicitly subtract Box2 from Box1, the inner box is only implicitly subtracted. Any calculation on the surface of Box1 in this case will use only the 6 exterior faces of Box1. Any calculation on the volume of Box1 will use the entire volume without subtracting the volume of Box2.

*** (Multiply)**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiplies the quantity in the top register by the quantity in the second register. One of the two registers must contain a scalar value; the other register can be either a scalar or a vector.

/ (Divide)

Divides the quantity in the second register by the quantity in the top register. The second register must contain a scalar value; the top register can be either a scalar or a vector.

Neg

Changes the sign of the quantity in the top register.

Abs

Takes the absolute value of the quantity in the top register.

Smooth

Smooths the quantity in the top register. Because of the numerical solution technique used, field values are not always continuous across the boundaries of the individual elements that make up the finite-element mesh. Smoothing makes the values continuous by taking a weighted average from all of a node's neighboring elements. The weights are based on angles, so elements with larger angles provide larger contributions. In general, use smoothing before plotting a quantity.

Complex

These commands perform operations on a complex quantity in the top register. Complex quantities are indicated by a **C** at the beginning of the register label. They can be represented in terms of real and imaginary components, or in terms of magnitude and phase:

$$C = A + jB = Me^{j\phi}$$

where:

- *A* is the real part of the complex number.
- *B* is the imaginary part of the complex number.
- *M* is its magnitude, which is equal to $\sqrt{A^2 + B^2}$.
- ϕ is its phase, which is equal to **atan**(B/A).

The **Complex** commands includes a drop-down menu that let you do the following:

Real	Takes the real part of the complex quantity (A).
Imag	Takes the imaginary part of the complex quantity (B).
CmplxMag	Takes the magnitude of the complex quantity (M). Due to interpolation issues, the sequence of calculations may cause a loss of accuracy. It is best to define the points , separately obtain the value of the real part, then the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	imaginary part, and use those values to calculate the magnitude and and phase. For the sequence for using the Fields Calculator to obtain the real and imaginary parts, see the procedure here.
CmplxPhase	Takes the phase of the complex quantity(ϕ).
Conj	Takes the complex conjugate of the quantity in the top register. If a complex number is given by $C = A + jB$, its complex conjugate is given by $C^* = A - jB$.
AtPhase	Lets you specify the phase angle, θ, at which an field quantity is evaluated. These quantities can be represented in the form $A(x,y,z,t) = A(x,y,z)\cos(\omega t + \theta(x,y,z))$. where • ω is the angular frequency at which the quantities are oscillating, specified during the solution. • θ(x,y,z) is the phase angle (the offset from a cosine wave that peaks at t=0). Entering the phase angle lets you compute the real part of the field's magnitude at different points in its cycle.
CmplxReal	Converts the real scalar of the top register to the real part of a complex number.
CmplxImag	Converts the real scalar of the top register to the imaginary part of a complex number.
CmplxPeak	Calculates the peak value of a given complex vector. Intuitively, this calculates the maximum magnitude of the equivalent real vector in a waveform.

Domain

This limits a calculation to the volume you specify. The domain filter works for scalars, vectors, complex scalars and complex vectors. This operation requires the top two entries of the stack to be a volume geometry and a numeric field quantity. To do this:

1. Load the field quantity into the top register, and perform any necessary operations on it.
2. Load the volume using the **Geometry** command.
3. Click **Domain**.

The **Domain** command is often used to limit a calculation or plot to the intersection of a surface and an object or group of objects. If you export a domain filtered numeric, points that are filtered out by the domain will not be written out.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Commands

The Scalar Column contains calculator operations which can only be performed on scalar stack entries.

Drop-down menus in this column include Vec? (convert scalar to vector), Trig (trigonometric, containing sin, cos, etc. functions), d/d? (derivative with respect to...), Max and Min (self-explanatory), Gradient, ln (natural logarithm base e), log (logarithm base 10), mean, and Std (standard deviation). Note that the calculator's Integrate function is located in the Scalar column. The implication is that integration can only be performed on scalar quantities. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.

Use these commands to perform operations on scalar quantities.

Vec?	Makes the scalar quantity in the top register a vector component.
1/x	Takes the inverse of the scalar quantity in the top register.
Pow	Raises a scalar quantity to the power you specify.
√? **(Square Root)**	Takes the square root of the quantity in the top register.
Trig	Performs a selected trigonometric operation on the value in the top register of the calculator stack (Sin, Cos, Tan, ASin, ACos, ATan). Angles are in radians.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Additionally, the ATan2 operation is available, which requires two arguments. This operator calculates the arc tangent of the following ratio: The second register in the stack divided by the top register. **Example:** If the top register value is 1.5, and the second register value is 2, the ATan2 function returns ATan(2/1.5) = 0.5880 radians. All other Trig functions operate on a single argument (top register only).
d/d?	Takes the partial derivative of the quantity in the top register.
∫? (Integral)	Takes the integral of a scalar quantity over a volume, surface, or line. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.
Min	Computes the minimum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
Max	Computes the maximum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
∇ (Gradient)	Takes the gradient of the scalar quantity in the top register.
ln	Takes the natural logarithm (base e) of the scalar quantity in the top register.
log	Takes the logarithm (base 10) of the scalar quantity in the top register.
Mean	Takes the mean of the quantity in the top register.
Std	Takes the standard deviation of the quantity in the top register.

Vec? Command

Makes the scalar quantity in the top register a vector component. Choose from the following:

VecX	The x-component of a vector.
VecY	The y-component of a vector.
VecZ	The z-component of a vector.

1/x (Inverse) Command

Takes the inverse of the scalar quantity in the top register.

Pow Command

Raises a scalar quantity to the power you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To raise a scalar quantity to a power:

1. Enter the quantity into the calculator.
2. Enter the exponent to which it is to be raised into the calculator.
3. Click **Pow**.

   The results are displayed in the top register.

(Square Root) Command

Takes the square root of the quantity in the top register.

d/d? (Partial Derivative) Command

Takes the partial derivative of the quantity in the top register:

d/dx	Takes the partial derivative of the quantity with respect to x.
d/dy	Takes the partial derivative of the quantity with respect to y.
d/dz	Takes the partial derivative of the quantity with respect to z.

(Integral) Command

Takes the integral of a scalar quantity over a volume, surface, or line. The top register must contain a geometry and the second register must contain the scalar quantity to be integrated.

To perform an integration:

1. Load a quantity into the top register of the calculator, and perform any required operations on it.
2. Use one of the **Geometry** commands to load the line, surface, or volume over which the quantity is to be integrated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you computed the tangent or normal of the quantity to be integrated, you do not have to load any geometry selection onto the calculator stack. Mechanical integrates the tangential or normal component of the quantity over the line on which you computed its tangent, or the surface on which you computed its normal.

3. Choose the ∫ command to integrate the scalar quantity over the geometry.

To find the numerical results of an integration, use the **Eval** command.

Min Command

Computes the minimum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the minimum value of the field.
Position	Finds the point where the minimum field value occurs. You can then: • Plot the minimum field value at the point. • Plot basic field quantities at the point. • Load the point into the calculator. • Change the point's location.

These commands operate in the same way as the **Max** commands. Use the **Eval** command to display the actual minimum field value or the coordinates of the point where it occurs.

Max Command

Computes the maximum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the maximum value of the field.
Position	Finds the point where the maximum field value occurs. You can then: • Plot the maximum field at the point. • Plot field quantities at the point. • Load the point into the calculator.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Change the point's location.

To compute the maximum field value:

1. Load a field quantity into the calculator, and perform any necessary operations on it. Keep the following in mind:
   - You cannot find the maximum value of a vector quantity. Therefore, make sure that the result is a scalar.
   - Before computing the maximum value of a complex quantity, you must find the real part of the quantity using the **CmplxReal** or **CmplxAtPhase** commands.
2. Load a point, line, or volume into the calculator using one of the **Geometry** commands.
3. Do one of the following:
   - Choose **Max/Value** to compute the maximum field value on the geometry.
   - Choose **Max/Position** to identify the point at which this value occurs.

Use the **Eval** command to display the actual maximum field value or the coordinates of the point where it occurs.

∇ (Gradient) Command

Takes the gradient of the scalar quantity in the top register.

Ln Command

Takes the natural logarithm (base e) of the scalar quantity in the top register.

Log Command

Takes the logarithm (base 10) of the scalar quantity in the top register.

Takes the mean of the quantity in the top register.

Std Command

Takes the standard deviation of the quantity in the top register.

Vector Commands

The Vector Column contains calculator operations which can only be performed on vector stack entries. Drop-down menus in this column are Scal? (convert vector to scalar) and Unit Vec (create unit vector). Standard vector algebra operations (Dot, Cross, etc.) are also present.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these commands to perform operations on vector quantities.

Scal?	Replaces the vector in the top register with a scalar quantity whose value is a component of the vector, your choice of ScalarX, Scalar Y, or ScalarZ.
Matl	Multiplies or divides the vector field quantity in the top register by a material property, or if you select MassDensity as the material property, produces a scalar that operates like a named variable.
Mag	Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.
Dot	Takes the dot product of the vector quantities in the top two registers.
Cross	Takes the cross product of the vector quantities in the top two registers.
Divg	Takes the divergence of the vector quantity in the top register.
Curl	Takes the curl of the vector quantity in the top register.
Tangent	Computes the tangential component of a vector quantity along a line
Normal	Computes the normal component of a vector quantity on a surface such as a cutplane or object surface.
Unit Vec	Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation. The drop-down lets you select from Tangent, Normal, or CoordSys[X, or Y, or Z]
X Form	Transforms to your choice of Cylindrical or Spherical.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scal? Command

Replaces the vector in the top register with a scalar quantity whose value is a component of the vector. Choose from the following:

ScalarX	Returns the x-component of the vector.
ScalarY	Returns the y-component of the vector.
ScalarZ	Returns the z-component of the vector.

Matl Command

Operates on the vector field quantity in the top register of the Fields Calculator based on a material property. At each tetrahedron, the field quantity is operated on by the value of the selected material property — taking the different material attributes of each object into account.

To operate on a vector quantity by a material property:

1. Click **Matl**.

   The *Material Operation* window appears.

2. Select a material property. Available properties are

Permittivity (epsi)	The permittivity, ε.
Permeability (mu)	The permeability, μ.
Conductivity	The conductivity, σ.
Omega (w)	The angular frequency, ω. The angular frequency is equal to $2\pi f$, where f is the frequency at which the solution was generated.
MassDensity	This is based on the value of the Mass Density material property. MassDensity is treated like a named expression. Selecting MassDensity disables the Operation radio buttons for Multiply or Divide in the *Material Operations* dialog box.

3. For Permittivity, Permeability, Conductivity or Omega, select an operation — **Multiply** or **Divide**. Selecting MassDensity disables these operations. The MassDensity scalar can be used like any other named expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Choose **OK** to operate on the field quantity by a material property or **Cancel** to stop the operation. If you selected MassDensity and click **OK**, a scalar named expression MassDensity is pushed onto the stack.

Mag Command

Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.

With a complex vector on the calculator stack, the **Mag** button returns a non-negative scalar. In previous software versions, this command returned a complex scalar.

Dot Command

Takes the dot product of the vector quantities in the top two registers.

Cross Command

Takes the cross product of the vector quantities in the top two registers.

Divg Command

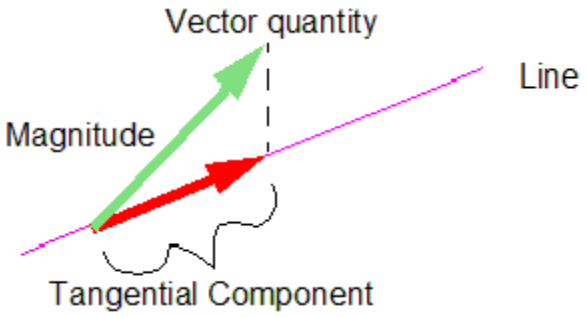
Takes the divergence of the vector quantity in the top register.

Curl Command

Takes the curl of the vector quantity in the top register.

Tangent Command

Computes the tangential component of a vector quantity along a line.



To take the tangent of a vector:

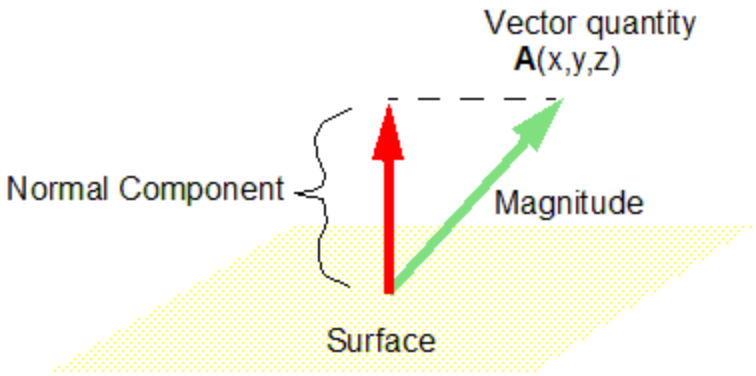
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Load a vector quantity into the top register.
2. Load a line into the top register using the **Geometry/Line** command.
3. Click **Tangent**.

Normal Command

Computes the normal component of a vector quantity on a surface such as a cutplane or object surface. This is the equivalent of taking the dot product of the quantity with the surface's unit normal vector:

$$Normal = A\left(x, y, z\right) \cdot \hat{n}$$



To take the normal of a vector:

1. Load a vector quantity into the top register.
2. Load a surface into the top register using the **Geometry/Surface** command.
3. Click **Normal**.

> **Note:**
>
> Because surface normals of sheets are not well defined the fields calculator can produce incorrect results if an expression is evaluated on a sheet. To enforce the correct direction of the surface normal of a sheet, a faceted 3D object (such as a box) can be defined such that one of its planar faces is coincident with the sheet. Because surface normals of a valid object are always defined in an outward direction in Mechanical, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit Vec Command

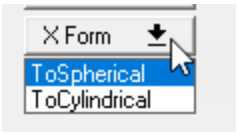
Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation.

Select from the following:

Tangent	Computes the unit vector tangent to the line specified at the time of plotting, integrating, or report generation based on the context.
Normal	Computes the unit vector normal to the surface specified at the time of plotting, integrating, or report generation based on the context.
CoordSys (X)	Computes the unit vector in the X-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Y)	Computes the unit vector in the Y-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Z)	Computes the unit vector in the Z-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.

X Form

Fields are defined in the Cartesian coordinate system. In many applications it is much more convenient to use a cylindrical or spherical coordinate system – for example, when the object is of cylindrical or spherical shape. The **XForm** command allows the user to transform vectors into either the cylindrical or spherical coordinate system. Selecting **XForm** displays a selection menu for the coordinate system (**ToCylindrical** or **ToSpherical**):

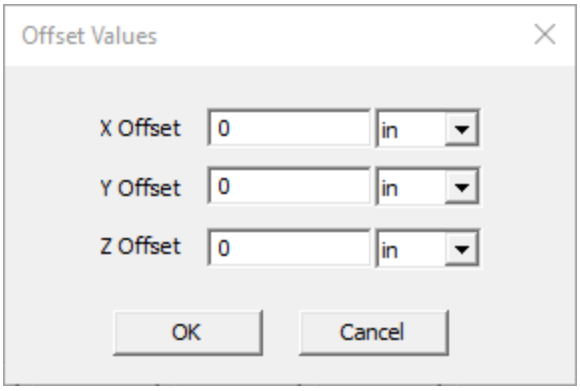


ToSpherical – Transforms the vector in the top register of the calculator stack into the spherical coordinate system.

ToCylindrical – Transforms the vector in the top register of the calculator stack into the cylindrical coordinate system.

After you select the coordinate system you want, the **Offset Values** dialog displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you enter X, Y, and Z offsets to be used as the origin of the new coordinate system.

Click **OK** to compute the offset coordinate transformation to apply to the (complex) vector quantity in the top register on the calculator stack based on your selection of Cylindrical or Spherical coordinates and your input of X, Y, and Z offsets. The new expression will be pushed onto the stack. This provides the origin of the new coordinate system. It will be something like

CVc : ToCylindrical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

or

CVc : ToSpherical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

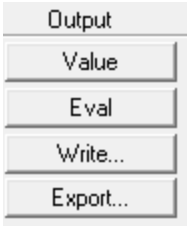
After you transform the quantities, you can then access the vector components in the Fields Calculator.

You will then use the ScalarX/ScalarY/ScalarZ commands under Scal?. For a vector converted to cylindrical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component, and ScalarZ gives the Z component. For a vector in spherical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component and ScalarZ gives the theta (azimuthal) component.

Output Commands

The Output Column contains those calculator operations that result in final data outputs from calculations. The Eval button obtains final numerical results from the last stack placeholder (such as integrations).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

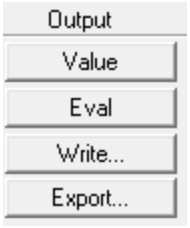


Use these commands to compute or evaluate expressions and to output the data in the calculator.

Value command	Computes the value of a field quantity on a geometry.
Eval command	Numerically evaluates and displays the results of calculator operations from the last stack placeholder, such as integrations.
Write command	Saves the contents of the top register to a disk file for future re-use.
Export command	Saves field quantities in a format that can be read by other modeling or post-processing software packages.

To plot/tabulate the extracted data using create report or create field overlays, you should use a named expression.

Value Command



This computes the value of a field quantity at a point. Use it to find:

- The magnitude of a scalar field quantity at that point.
- The x-, y-, and z-components of a vector field quantity at that point.

The **Value** command applies the value of the next-to-top stack entry (a field quantity) on the geometry entry at the top stack entry. The field quantity entry may be scalars, real or complex vectors. The geometry entry may be of any type (point, surface, line, or volume).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if the top stack entry in the Field Calculator is a surface and the second stack register is for the real part of the E Vector ("Real(<Ex, Ey, Ez>)"), the resulting quantity will be an expression for the real part of the E vector on the surface (a VecSrf quantity). Again, the result of this command is still a placeholder expression, not the final numerical values.

Strictly speaking, the Value command provides access to the field quantity, not data outputting. It simply provides a handle to the (numerical) values of a field quantity on a given geometry, and thus the outcome of the command is not numerical values themselves. If the numerical value of a field quantity is desired, you will need to follow it up with one of the three other output commands, or define a named expression for the quantity and plot it with the Reporter.

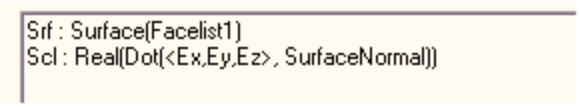
To find the value of a field quantity at a point:

1. Load the field quantity into the top register, and perform any needed operations on it.
2. Load the appropriate point into the calculator using the **Geometry/Point** command.
3. Click **Value**.

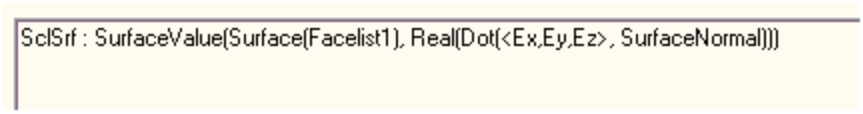
To view the numerical results of this operation, use the **Eval** command.

You can also use the **Value** command to access the intermediate SurfaceValue function.

For example, after inputting an expression for a quantity, such as an E field, and then selecting a surface geometry, the calculator stack displays something like this:



Clicking the **Value** command changes the display to the following, showing the intermediate SurfaceValue function.



In this case, SurfaceValue provides the x, y, z, coordinates of the FEM mesh and Lagrangian points so you can use **Write** to generate an .fld file containing an evaluated scalar quantity at those points.

In general for **Value**:

1. Enter any quantity onto the stack.
2. Enter a volume / surface / line / point onto the stack.
3. Press the **Value** button and you will get an appropriate geometry value on the stack.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

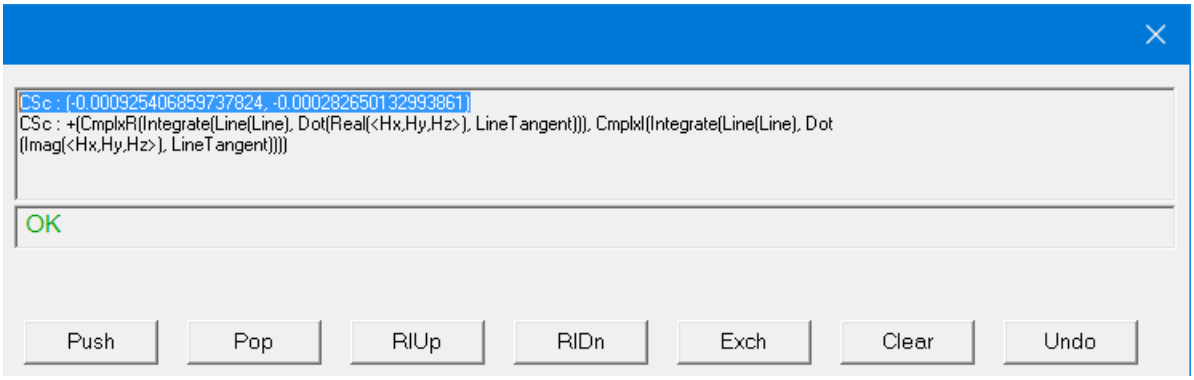
Now you can perform suitable operations such as Write, Integrate, etc. For PointValue you can also do **Eval**.

Eval Command

The **Eval** command finalizes computations and converts the text-string indicating the computation being performed to an actual numerical value. This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values at points, and so forth. For this command to work properly, the stack entry must be reduced to a single-value entry which can be a scalar, complex, or vector (real or complex). The quantity to be evaluated must be in the top register. The **Eval** command computes the numerical results of the operation, which replace the contents of the register. Units of the numerical value are not provided.

For a calculator expression that evaluates to a single value, the **Eval** command offers a quicker way to obtaining the final numerical result than defining a named expression and then trying to plot/tabulate it in the reporter. But for many expressions that don't evaluate to single values, such as the wave impedance along a line mentioned earlier, **Eval** command does not apply. As such, use of named expression and the Report Editor will be the only option.

Evaluate the expression for voltage along a line shows the definition of an expression for the voltage between two conductors by integrating the electric field along a line using the recipe given for Calculating the Current along a Wire or Trace. Then, by clicking the Evaluate command in the Output column, we obtain the numerical value for the voltage as shown in the top of the stack. It can then be either written down or copied/pasted to other programs.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

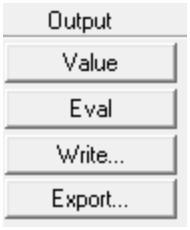
$$I = \oint H \cdot dl$$

Because **H** and **I** are complex quantities, you will need to evaluate the real part of **H** to obtain the real part of **I**, then evaluate the imaginary part of **H** to obtain the imaginary part of **I**. To do this:

1. Load **H** into the calculator using the **Qty** command.
2. Take the real part of **H** using the **Cmplx/Real** command.
3. Load the rectangular loop using the **Geom/Line** command. Create the loop, a closed polyline, to integrate over.
4. Click **Tangent** to get the component of **H** along the line.
5. Take the integral around the loop using the ∫ command.
6. Click **Eval** to evaluate the integral. The real part of **I** appears in the top register.
7. Repeat this process using the imaginary part of **H** (found with the **Cmplx/Imag** command) to obtain the imaginary part of **I**.

Write Command

If the calculator operations performed have obtained a stack entry that is intended for use in still other calculator operations, the stack entry can be saved for this purpose by using the **Write** button in the calculator Output column.



Note that in this case no name will be requested for the expression; only a save filename location will be required. This function will not work for field values derived upon a specific geometric quantity (those containing either Lin, Srf, or Vol in the stack data type indicator) as the calculator cannot know that these geometric quantities exist in identical forms in other postprocessing sessions.

When you try to read a file written with the **Write** command by clicking the **Read** button in the Input column, you must ensure that the mesh used for generating the current solution is the same as the one with which the saved field data is generated. Since the two results must be generated on the same mesh, not only the geometries of the designs on which the **Write/Read** commands were applied must be the same, they must also have gone through the same adaptive meshing process. In practice, this limits the use of Write/Read command to solutions for different frequency sweeps of the same analysis (adaptive) setup of the same design.

This command saves the contents of the top register to a disk file. Use this command to:

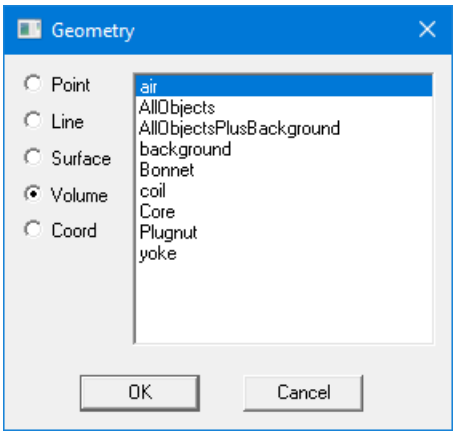
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Save registers for use during a later postprocessing session.
- Save a field quantity for use when postprocessing a different model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To save a register:

1. Click **Write**.
2. If the register includes numeric data with a constrained quantity (such as jsurf), you see a dialog box that gives a choice of constraining geometries. For example:



3. Select the geometry of interest, and select **OK**.

   This displays a file browser.

4. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assigned to register files and a .fld extension is assigned to field files. You can choose to save both .reg and .fld files, or either one.
5. Click **OK**.

   The contents of the register are saved to the file you specified.

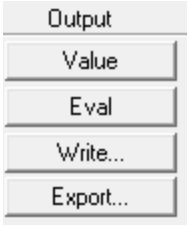
Export Command

To output a field quantity or calculation result for use by some third-party post-processors, use the **Export** button in the calculator Output column. You can map the field quantity to either a customized grid of points specified by a points file or a three-dimensional Cartesian, Cylindrical, or Spherical grid specified interactively through a dialog box. In the latter case, you must specify

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the dimensions and spacing of the grid in the coordinate system with units. For Cylindrical and Spherical coordinate systems, you can also specify an offset from the origin.

The **Export** command opens the **Export Solution** dialog box, from which you can export the field quantity in the top register to a file, mapping it to a grid of points.



Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available for defining the grid points on which to export:

Input grid points from file	Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points and units.
Calculate grid points	Maps the field quantity to a three-dimensional Cartesian grid. You specify the dimensions and spacing of the grid in the Cartesian, cylindrical, or spherical coordinates, with units that you specify. The initial units are taken from the model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To export a field quantity to a customized grid:

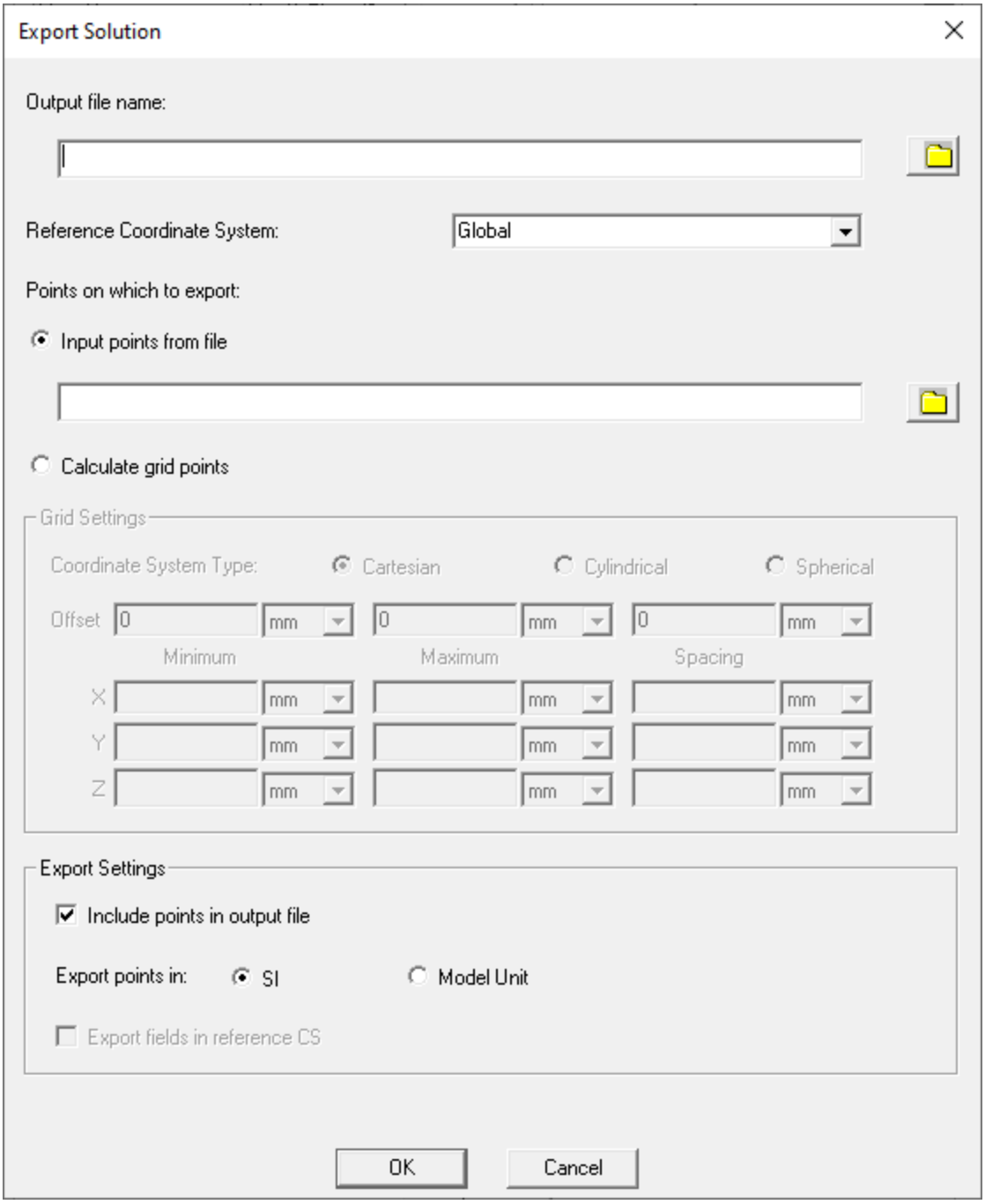
1. Load the quantity into the top register for the fields calculator, and perform any operations on it.
2. If desired, load a volume using the **Geometry** command.

   You can use the Domain command to limit the calculation to the volume you specify. If you export a Domain filtered numeric, points that are filtered out by the domain will not be written out.

3. Click the **Export** button in the Fields Calculator.

   This opens the **Export Solution** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Type or select the name of the file in which the field quantity is to be saved in the **Output File Name** text box. You can use the file icon to open the file browser to specify the file name and directory path. The **.fld** extension is automatically assigned to this file.
5. If the design contains multiple coordinate systems, a drop-down menu lets you select the coordinate system to use. This can be helpful in evaluating the fields output when relative

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate systems used and unitized in the design.

6. Click either the **Input grid points from file** radio button if you have a created a .pts file containing the grid points, or click the **Calculate grid points** radio button.
   - If you select **Input grid points from file**, either type the name and directory of the file containing the points on which the field is to be mapped, or, click on the file icon and use the file browser to locate the point file (**.pts** extension).

     Note:

     The **.pts** file should contain the units to use for the export as shown in this file stub:

     Unit=mm

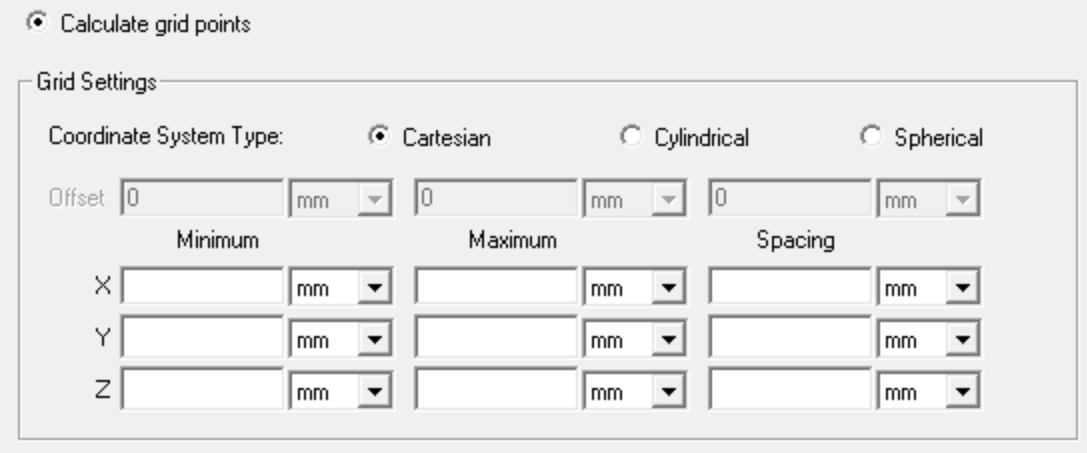
     -5.5 -5.5 -5.21475

     -5.5 -5.5 -5.14425

     -5.5 -5.5 -5.07375

     -5.5 -5.5 -5.021

   - If you select **Calculate grid points** button, you can specify the coordinate system as Cartesian, Cylindrical, or Spherical.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian: for each grid dimension on **X**, **Y**, and **Z**, enter the Minimum, Maximum, and grid point spacing.

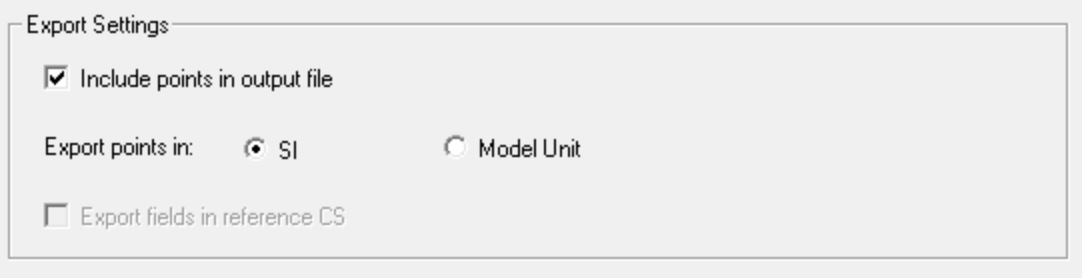
Cylindrical: for each dimension Rho, Phi, and Z, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

Spherical: for each dimension R, Theta, and Phi, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

> **Note:** When you export fields on a 1D or 2D line/surface from the field calculator, the start and stop values must be the same for one or two of the coordinate system start/stop ranges. If you specify a zero spacing for a dimension, the export uses only the minimum value.

The default coordinate system will be Cartesian. The default offset will be all zeroes. The length units will default to model unit and default angle unit will be degree. At the start the minimum/maximum/Spacing entries are blank. The user entered values are not remembered when the dialog is closed.

7. For larger files, for the Export Settings, you may want to uncheck the **Include points in output file** box. If you uncheck the box, the file header will include minimum, maximum and spacing information from which you can recalculate the grid points.
8. You can also specify SI or Model Units. Model Units can be helpful to evaluate the fields output when relative coordinate systems are unitized in the design.



9. If field to be exported has type vector or complex vector, the 'Export Field in reference CS' checkbox is enabled and you can specify that field to be exported in reference CS. It can be helpful to evaluate the fields output when rotated coordinate systems are unitized in the design
10. Click **OK** to export the file.

    The field quantity is mapped to the grid and saved to the file you specified (**.fld** extension).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 2D Reports from Named Expressions

You can create a 2D report from that evaluate as a single scalar value or that evaluate along polylines.

To create a report from a named expression evaluating as a single scalar value:

1. Click **Mechanical**, and then click **Results** > **Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.
7. Add one or more traces to include in the report.
8. Click **Done**.

   The report appears in the view window and is listed in the project tree. Once you have created a report, additional options become available on the **Results** submenu.

To create a report from a named expression evaluating along a polyline:

1. Click **Mechanical** and then click **Results> Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.
7. Select the geometry you want to plot from the **Geometry** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Add one or more traces to include in the report.

> **Note:**
>
> If you select a polyline for the **Geometry** and a **Category** that is a named expression based on the line, then the **Quantity** list only lists corresponding expressions. The list will not contain scalar values when a line is selected as the geometry.

9. Click **Done**.

   The report appears in the view window and is listed in the project tree. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes. Once you have created a report, additional options become available on the **Results** submenu.

Calculating Derived Field Quantities Using Named Expressions

The **Named Expressions** panel displays expressions that can be included in register definitions by name. You can add additional expressions to the Named expression list by creating the expression in the register display area and then clicking the Add button. This lets you add to the Named expression library.

Predefined named expressions for Mechanical designs are shown below for each solution type:

Named Expressions

Name	Expression
Mag_Displacement	Mag(Smooth(<Ux,Uy,Uz>))
Displacement_Vector	Smooth(<Ux,Uy,Uz>)

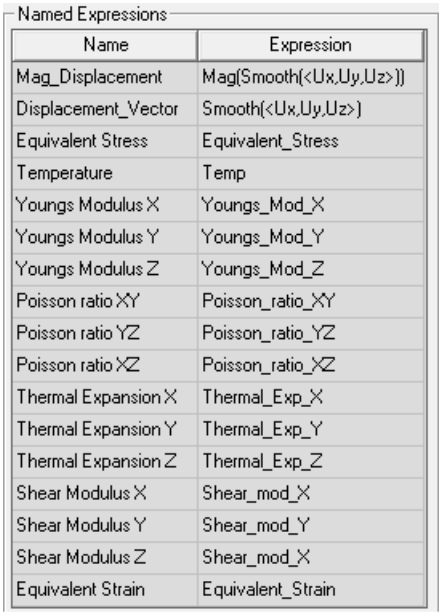
Modal Solutions

Named Expressions

Name	Expression
Temperature	Temp
HeatFlux	<qx,qy,qz>
Mag_HeatFlux	Mag(<qx,qy,qz>)
Surface Loss Density	Surface_Loss_Density
Volume Loss Density	Smooth(Volume_Loss_Density)
Linked Heat Transfer Coefficent	Linked_Heat_Transfer_Coefficient
Heat Flow Rate	Dot(<qx,qy,qz>, SurfaceNormal)
Thermal Conductivity X	Kx
Thermal Conductivity Y	Ky
Thermal Conductivity Z	Kz

Thermal Solutions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.


Named Expressions

Name	Expression
Mag_Displacement	Mag(Smooth(<Ux,Uy,Uz>))
Displacement_Vector	Smooth(<Ux,Uy,Uz>)
Equivalent Stress	Equivalent_Stress
Temperature	Temp
Youngs Modulus X	Youngs_Mod_X
Youngs Modulus Y	Youngs_Mod_Y
Youngs Modulus Z	Youngs_Mod_Z
Poisson ratio XY	Poisson_ratio_XY
Poisson ratio YZ	Poisson_ratio_YZ
Poisson ratio XZ	Poisson_ratio_XZ
Thermal Expansion X	Thermal_Exp_X
Thermal Expansion Y	Thermal_Exp_Y
Thermal Expansion Z	Thermal_Exp_Z
Shear Modulus X	Shear_mod_X
Shear Modulus Y	Shear_mod_Y
Shear Modulus Z	Shear_mod_X
Equivalent Strain	Equivalent_Strain

Structural Solutions

Click on a named expression to select it. When a named expression has been selected, the **Copy to Stack** button is activated. Click **Copy to Stack** to push the expression on the top of the stack.

When a Mechanical design is open and a Solution Setup has been performed, the following predefined named expressions are available.

Expression Name	Expression Definition
Mag_Displacement (Modal and Structural Solutions)	**Mag(Smooth(<Ux,Uy,Uz>)):** The resultant magnitude of the displacement vectors, which represent the deformed shape of the selected vibration mode for Modal solutions and the actual structural deformation for Structural solutions. **Note:** For Modal solutions, the displacement results are not scaled to any specific structural excitation but are arbitrary. Only the relative displacements throughout the model are meaningful in that they demonstrate the vibration mode shape. The absolute displacement magnitudes are not meaningful for Modal solutions and may in fact be well out of proportion with the physical size of the structure.
Displacement_Vector (Modal and Structural	**Smooth(<Ux,Uy,Uz>):** A vector representation of the preceding displacement expression. For Modal solutions, see the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solutions)	preceding note for *Mag_Displacement*, which also applies to this vector expression.

> **Note:**
>
> If you create and run the Modal Toolkit Example, four additional expressions will be added to the *Named Expressions* panel: the normalized displacement (≤ 1 length unit) and the X, Y, and Z components of the displacement vector.

Temperature (Thermal and Structural Solutions)	**Temp:** The temperature at each calculation point throughout the model. This result is useful for verifying that temperatures have been properly mapped to the model when importing thermal conditions from a separate source solution.
HeatFlux (Thermal Solutions)	**<qx,qy,qz>:** The heat flow rate vector defined on a power per unit area basis.
Mag_HeatFlux (Thermal Solutions)	**Mag(<qx,qy,qz>):** The magnitude of the HeatFlux vector.
Surface Loss Density (Thermal Solutions)	**Surface_Loss_Density:** This quantity concerns thermal excitations (EM Losses) imported from the surfaces in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Volume Loss Density (Thermal Solutions)	**Smooth(Volume_Loss_Density):** This quantity concerns thermal excitations (EM Losses) imported from the volumes in another design. The electromagnetic losses are due to inherent inefficiencies in the materials, and these become a source of heat in the thermal solution.
Linked Heat Transfer Coefficient (Thermal Solutions)	**Linked_Heat_Transfer_Coefficient:** The imported convection *Film Coefficients* when you have choosen the **Non-Uniform** option and have set up a link to an Icepak source solution.
Heat Flow Rate (Thermal Solutions)	**Dot(<qx,qy,qz>, SurfaceNormal):** The surface-normal component of heat flux used to approximate total heat flow rates at specified faces by integration. A face list is required for the integration faces to be selectable within the Fields Calculator. Note that there are effects at the edges of faces that skew the total integrated heat flow values when this method is employed. As such heat flow may appear to be out of equilibrium. More accurate Heat Flow Rate results can be obtained by creating a Fields Summary. Additionally, you can directly select boundaries, such as Contact and Convection assignments, in the *Create Fields Summary* dialog box. That is, you need not create face lists to determine the total heat flow rates at boundaries when creating a Fields Summary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Conductivity X, Y, and Z (Thermal Solutions)	**Kx**, **Ky**, and **Kz**: The thermal conductivity in the X, Y, and Z directions, respectively, for layout component materials. (See **Note [1]** at the bottom of this table.)

> **Note:**
>
> If you create and run the Thermal Toolkit Example, three additional expressions will be added to the *Named Expressions* panel: specifically, the global maximum, mean, and minimum temperature results (*Global_Max_Temp, Global_Mean_Temp,* and *Global_Min_Temp*). Each of these named expressions reports a single value from the set of all object volumes.

Equivalent Stress (Structural Solutions)	**Equivalent_Stress:** The von Mises equivalent stress at each calculation point throughout the model. The equivalent stress result combines the effects of the six individual stress tensors or three principal stresses. It is typically compared to the material's yield strength for ductile metals. For brittle materials, this stress may be compared to the ultimate tensile strength.
Young's Modulus X, Y, and Z (Structural Solutions)	**Youngs_Mod_X**, **Youngs_Mod_Y**, and **Youngs_Mod_Z**: The modulus of elasticity in the X, Y, and Z directions, respectively, for layout component materials. (See **Note [1]** at the bottom of this table.)
Poisson's Ratio XY, YZ, and XZ (Structural Solutions)	**Poisson_ratio_XY**, **Poisson_ratio_YZ**, and **Poisson_ratio_XZ**: These ratios quantify how the cross-section of a material sample changes as it is strained in the perpendicular direction. Three values are available, each being the negative of the ratio of transverse strain and axial strain resulting from an applied axial load. (See **Note [1]** at the bottom of this table.)
Thermal Expansion X, Y, and Z (Structural Solutions)	**Thermal_Exp_X**, **Thermal_Exp_Y**, and **Thermal_Exp_Z**: The coefficient of thermal expansion in the X, Y, and Z directions, respectively, for layout component materials. (See **Note [1]** at the bottom of this table.)
Shear Modulus X, Y, and Z (Structural Solutions)	**Shear_mod_X**, **Shear_mod_Y**, and **Shear_mod_Z**: Shear Modulus is also know as the modulus of rigidity, as it is a measure of the rigidity of a material. Specifically, it is the ratio of the shear stress and shear strain. (See **Note [1]** at the bottom of this table.)
Equivalent Strain (Structural Solutions)	**Equivalent_Strain:** The von Mises equivalent strain at each calculation point throughout the model. The equivalent strain result is calculated from the six strain tensor values and the effective Poisson's ratio (v'), which equals the specified Poisson's ratio for isotropic materials undergoing elastic strain. Material strain is a dimensionless quantity though typically

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

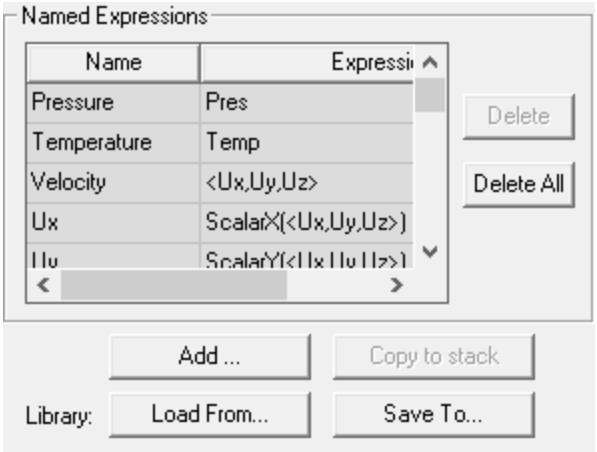
expressed in units of length/length.

Note:

[1] – Directional results are applicable to designs that include layout components (that is, indirect representation of metals by locally altering dielectric material properties of the layout components). Properities are mapped to individual elements comprising the mesh. The resultant values can vary by direction even though the constituent material properties are isotropic.

Named Expression Library

The named expression library in the Fields Calculator provides a way to conveniently calculate frequently used quantities. Some standard field quantities have been predefined and are always accessible. These field quantities have a gray background color and cannot be deleted. You can assign a name to the top entry of Data Stack and add it in the Named Expression list. The library comes with several predefined expressions. Once these new expressions are added/loaded, they are available through Mechanical's standard post-process capabilities. For example, they can be copied back to the Data Stack in calculator by **Copy to Stack** for further operations, or be plotted using Report Editor and Field Overlay.



You can combine calculator Input commands in any legal fashion, including complex quantities, to produce new named expressions.

- Adding named expressions to the Fields Calculator expression library
- Copying named expressions to the Calculator Stack
- Saving named expressions to a Personal Library
- Loading named expressions
- Deleting named expressions that you added

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To add a named expression of your own to the Fields Calculator list:

1. In the register display area, create the expression by using the calculator Input commands.

   You can combine input commands in any legal fashion, including the use of complex quantities. If you select an input command that is not legal for a current operation, you receive an error message.

2. When you finish creating the expression, click **Add** in the *Named Expressions* panel.

   The *Named Expression* dialog box appears.

3. Type a name for the expression in the **Name** text box.

   The new expression is added to the list of named expressions.

To copy a named expressions to the Calculator Stack:

- You can scroll through the list, select any desired named expression, and click **Copy to Stack** to move it to the calculator stack, where you can use it to generate calculated outputs.

To delete named expressions that you added:

When the *Named Expression* list contains one or more user-defined expressions, the **Delete** and **Clear All** buttons are active. (You cannot delete or clear the predefined named expressions.)

- To delete the selected user-defined named expression, click **Delete**. To delete all user-defined named expressions, click **Clear All**.

To save one or more named expressions for the Fields Calculator to a personal Library:

1. Click **Save To** on the Fields Calculator.

   The *Select Expressions for Saving* dialog box displays.

2. If any new named expressions exist, you can select one or more to save to a file.
3. Specify a file name and click **OK** to save the file.

To load named expressions for the Fields Calculator from a personal library:

1. From the Fields Calculator, click **Load From**.

   This displays a file browser that you can use to search for existing .clc files.

2. Select the library to load and click **OK**.

   This loads the expression file you have selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exiting the Fields Calculator

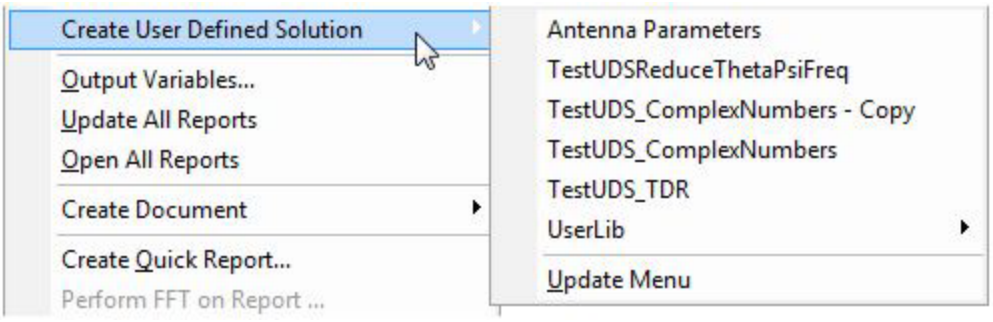
Click **Done** to exit the Fields Calculator.

User Defined Outputs (UDOs)

User defined outputs (UDOs) allow users to define calculations through IronPython scripts or any .NET language (and used by the IronPython script). The UDO scripts need to be in the **UserDefinedOutputs** directory under either **syslib**, **userlib**, or **Personallib** with any directory structure needed for organization. (The **Lib** directory name is special and its purpose will be explained later on in the document.)

The UDO scripts that are placed in syslib/UserDefinedOutputs, userlib/UserDefinedOutputs, or Personallib/UserDefinedOutputs become available to the user to create "User Defined Solutions" through the **Results** > **Create User Defined Solution** menu.

Use **Results** > **Create User Defined Solution** > **Update Menu** to refresh the menu to include the new UDO scripts that might have been copied to syslib, userlib, or Personallib; or exclude them if they have been deleted, after the launch of desktop. Once the user-defined-solution is created, the solution and the calculations defined by UDO become available in Reporter as any other quantities in a new "User Defined" report type.

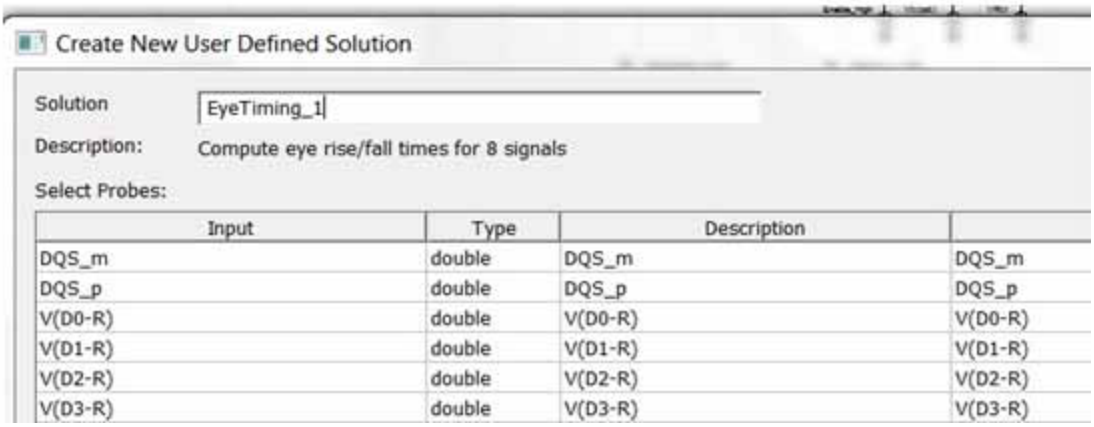


Named Probes and Properties in User Defined Outputs

UDOs allow processing data across traces, solutions, and report types. A UDO specifies the named probes and properties for which user selects or enters the values at the time of creation of user-defined solution. Probes are very similar to traces except that the user selects the values of only intrinsic variables for probes. The values of design or project variables are selected when a trace is created based upon the user defined solution in reporter.

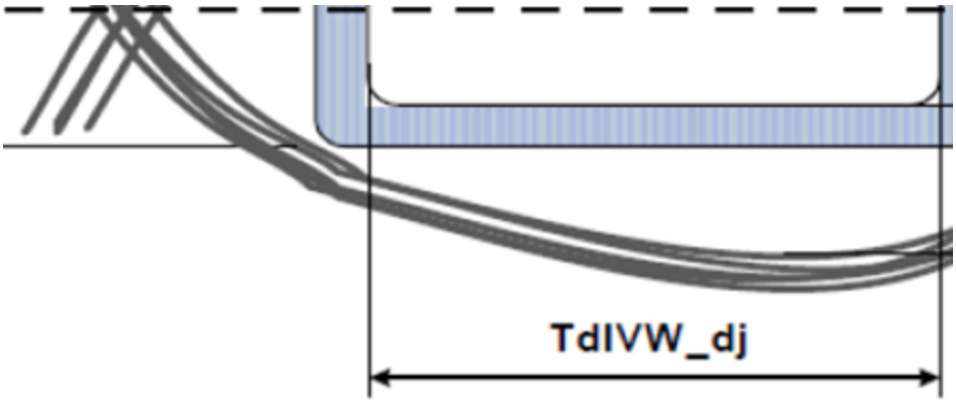
For example, you could create a user-defined solution called EyeTiming_1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Input	Type	Description	
DQS_m	double	DQS_m	DQS_m
DQS_p	double	DQS_p	DQS_p
V(D0-R)	double	V(D0-R)	V(D0-R)
V(D1-R)	double	V(D1-R)	V(D1-R)
V(D2-R)	double	V(D2-R)	V(D2-R)
V(D3-R)	double	V(D3-R)	V(D3-R)

You can then access this solution in the Reporter.



In another example, you could create a user-defined solution called **UDS Distance Trace Arithmetic Result1**:

Then access this solution in the Create User Defined Solution menu:

Computation of Traces Based UDO Calculations

When traces that are based upon UDO outputs are computed, the data for probes is computed and passed to the UDO script for each design variation. Along with the probe data, the values of properties entered by user are also passed. The information about the UDO calculations that need to be computed is also made available. The UDO then performs the computation and passes the results to reporter. Note that UDOs can compute and pass back more calculations than have been requested at that point of time. This allows UDOs to compute a set of calculations that take almost same amount of computational resources as any one calculation in

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that set and cache that with reporter. *When those calculations are subsequently plotted by user, reporter will use the cached results instead of invoking the computation on UDO*.

Dimensions Reduction by UDO Calculations

The probes in a UDO can have heterogeneous dimensions of data, for example, one probe in a UDO can have data that is a function of *n* intrinsic variable, while another probe in same UDO can have data that is function of *m* intrinsic variables, with *n* and *m* are potentially different. UDOs allow you to reduce any number of these intrinsic variables. In the above example, UDO calculations can be a function of any number of intrinsic variables including not being function of any intrinsic variable at all. UDO calculations can also be a function of an intrinsic variable that none of the probes is function of. *The only restriction is that Freq cannot be reduced if any of the probes are on a Fields report type.*

Dynamic Probes

In addition to named probes and properties, UDOs can specify named dynamic probes. The difference between probes and dynamic probes is that while the end user of UDO specifies the complete trace definition for probe, the expression for dynamic probe is specified by UDO code itself and not by end user. This allows UDOs to access the data for probes without requiring the end user to enter each individual probe. For example, a UDO can access data for a huge S matrix for 100 port design without having the end user enter the probe information for each of those 10,000 quantities. Each dynamic probe is associated with a named probe that is entered by user. Information about solution, context, and intrinsic variables is used from selected probes; however multiple dynamic probes can be associated with the same user selected probe. The dynamic probes are acquired from UDOs at the time of trace computation and not at the time of creation of user defined solution.

This means that you select solution, context, and values of intrinsic variables just once, and the same information is used (in this case) for all clock and data signals. The expression for those signals comes from the UDO code.

User Defined Outputs: Python Script API

A User Defined Output (UDO) extension is implemented as an IronPython script that defines a class with a specific name: **UDOExtension**, which derives from a specific base class **IUDOPluginExtension** and implements its abstract methods.

- UDO Extension Implementation
- Data Types Used in UDO Python Scripts
- Working With Properties for UDO
- Other Application Specific Classes Used in Python Scripts
- User Defined Outputs: Messaging Methods
- Using .NET Collection Classes and Interfaces in Python Scripts

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDO Extension Implementation

The purpose, argument list, and expected return types for each of the IUDOPluginExtension abstract methods, which the UDO author is expected to implement are described below.

- Import Statements
- UDOExtension Class
- IUDOPluginExtension Abstract Class

Import Statements

The base class to use and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDO script. The following import statements should be added to the top of the Python script:

```
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
 from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
```

UDOExtension Class

The UDO itself should be implemented as an IronPython class called **UDOExtension** which *must* derive from the **IUDOPluginExtension** abstract base class (from the **Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDOs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDO class must derive from **IUDOPluginExtension**.

The UDOExtension abstract class declares the optional Validate abstract method that may be implemented in the UDOExtension class or one of its base classes.

Example:

```
def BaseClassUDO ((IUDOPluginExtension):
 #base class implementation
 …
 def UDOExtension ((BaseClassUDO):
 #UDO class implementation …
```

Validate

This optional method is used to validate the user choices. The values of the properties entered, the probes etc. can be checked for suitability. This function, while a part of the IUDOExtensionabstract class, has a meaningful default implementation and is therefore optional. However, it can be overridden to take advantage of advanced functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name	Type	Description
	<errorStringList>	List<string>	[out] C# list of Python strings. Should be set only if validation failed; ignored if validation is successful. One error string should be set per each validation error.
	<udsProbParams>	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
	<propList>	IPropertyList object	[in] list of properties
	< userSelectionForDynamicProbes >	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
Return Value	Boolean. True on validation success and false on failure. The default implementation always returns true.		

Python Syntax	Validate (<*errorList*>, <*probeList*>, <*propertiesList*>, <*dynamicProbes*>)

Python Example	`def Validate(self, errorStringList, probeList, propList, dynamicProbes):`
	`  if probeList == None or probeList.Count == 0:`
	`  errorStringList.Add("Empty probe list")`
	`  return False`
	`  return True`

IUDOPluginExtension Abstract Class

The implementation of the IUDOPluginExtension class will be described in this section using a simple UDO example that expects a single probe and reduces its dimension returning as its outputs, the max, min, and average of its input probe data. The script in its entirety will also be listed later on.

Required functions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The IUDOPluginExtension abstract class declares the following abstract methods that must be implemented in the UDOExtension class or one of its base classes. Not implementing any of these methods will result in a run-time error and a non-functioning UDO. The UDS refers to User Defined Solution parameters.

- GetUDSName
- GetUDSDescription
- GetUDSSweepNames
- GetCategoryNames
- GetQuantityNames
- GetQuantityInfo
- GetInputUDSParams
- GetDynamicProbes
- Compute

GetUDSName

Return a string that is used as a prefix for all solution instances created using this UDO.

UI Access	NA
Parameters	None
Return Value	String

Python Syntax	GetUDSName

Python Example	`def GetUDSName(self):`
	`    return "MinMaxAvg"`
	`udsName = UDOPluginEntention.GetUDSName`

GetUDSDescription

Returns a description for the UDO, its purpose, etc. This is used in multiple UDO related dialogs in the application to describe the UDO.

UI Access	NA
Parameters	None
Return Value	String

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	GetUDSDescription
Python Example	`def GetUDSDescription(self):` `    return "Sample UDO for dimension reducing quantities`

GetUDSSweepNames

Returns a list of sweep names to be used for the solution generated by the UDO. These will appear in the sweeps list displayed in the standard reporter dialog when used to create reports from the solution generated by the UDO.

UI Access	NA
Parameters	None
Return Value	List of strings. If the UDO outputs have no sweeps, return the empty list [].

Python Syntax	GetUDSSweepNames

Python Example	`# Returns list of sweeps names`
	`# We have no sweeps as we reduce them.`
	`def GetUDSSweepNames(self):`
	`    return []`

GetCategoryNames

The outputs that the UDO solution provides or generates can be classified into multiple categories, like how the application is displayed in the report creation dialog box.

UI Access	The output will be listed in the categories box in the dialog when creating reports from the UDO generated solution data.
Parameters	None
Return Value	List of strings

Python	GetCategoryNames

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Syntax	
Python Example	`def GetCategoryNames(self):` `  return ["UDOOutputs"]`

GetQuantityNames

For each of the category names returned from the GetCategoryNames method, this function is called to return a list of quantities to be organized under that category name.

> **Note:**
>
> The quantity names must be unique across the categories; that is, no two categories can have quantities with the same name.

UI Access	NA		
Parameters	Name <*categoryName*>	Type String	Description [in] Name of category.
Return Value	List of strings		

Python Syntax	GetQuantityNames(<*categoryName*>)

Python Example	`# returns a list of quantity names for the supplied category name`
	`def GetQuantityNames(self, catName):`
	`  if catName == "UDOOutputs":`
	`  return ["min_val", "max_val", "avg_val"]`
	`  else:`
	`  return []`

GetQuantityInfo

For each quantity that the UDO creates, it must also describe the quantity (unit and other details). This method is called for each quantity name (across all categories) as returned from an earlier call of the GetQuantityNames method.

UI Access	NA		
Parameters	Name	Type	Description

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

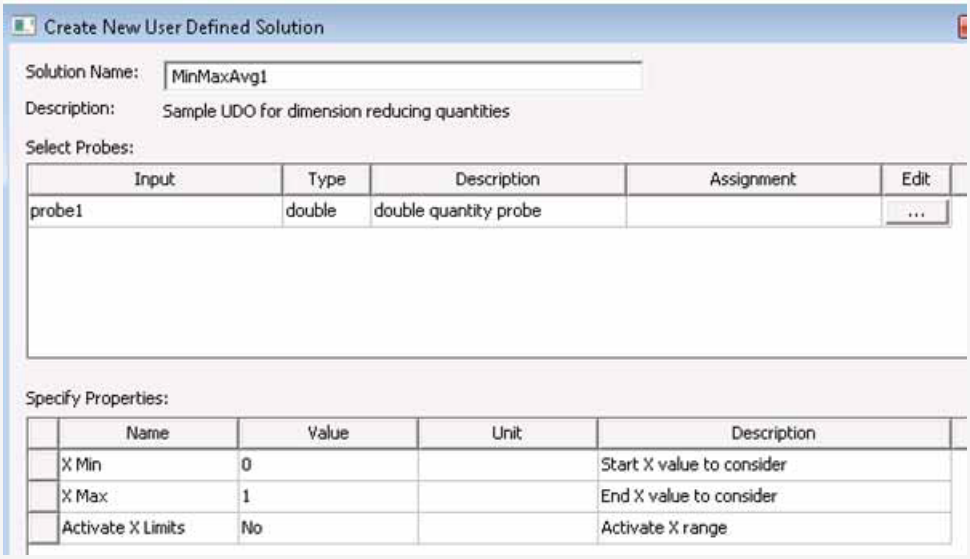
	<quantityName>	String	Name of quantity
Return Value	Object of type QuantityInfo.		

Python Syntax	GetQuantityInfo(*<quantityName>*)

Python Example	`# Returns an instance of QuantityInfo for the qtyName supplied or None if such a`
	`# quantity could not be found`
	`def GetQuantityInfo(self, qtyName):`
	`    # All the quantities we have are simple doubles`
	`    # we can leave them unitless`
	`    return QuantityInfo(Constants.kDoubleParamStr)`

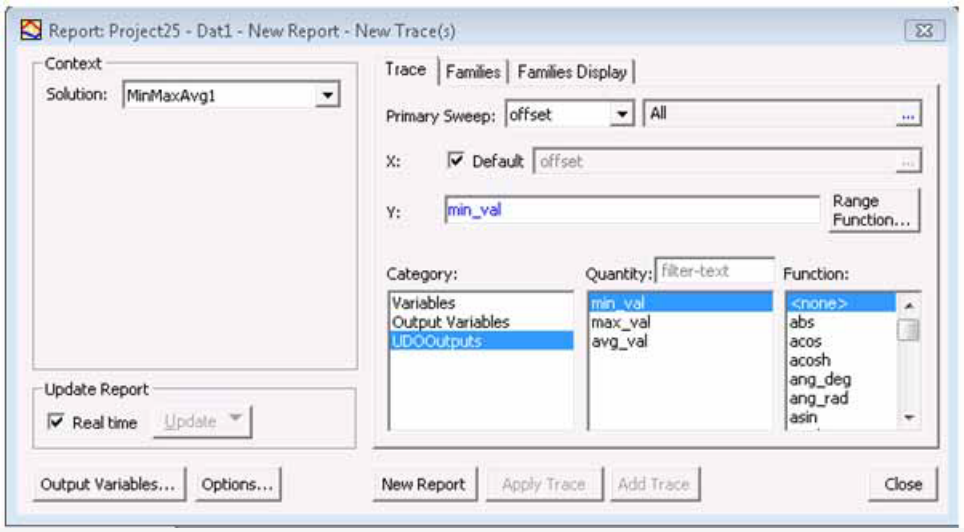
GetInputUDSParams

This is the main definition method of the UDO. The supplied arguments populate details of the parameters to which the UDO user will specify value, specify the probe names and their types as well as the dynamic probe selections.

UI Access	The **GetInputUDSParams** function results in the following dialog when you click **Results**>**Create User Defined Solution**. The mapping from the UDSParams and the properties to the GUI elements should be unambiguous. The name and description of the UDS are also displayed in the **Create New User Defined Solution** window. 

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a report is created from the UDO dialog box, the category and quantity names specified by the UDO are used (as seen below).



Parameters

Name	Type	Description
<udsParams>	List< UDSProbeParams >	Parameters for the probe. The UDO script must add one instance of UDSProbeParams for each probe definition it will display. When creating the UDO solution, the UDO user must assign a matching quantity to each probe. When user define a UDS in the GUI, the user must select a design-solution-quantity for each UDSProbeParam .
<propList>	IPropertyList	The propList object is used to add properties that should be displayed to the user for data collection. These properties with their user-supplied values are returned to the UDO script by the Compute

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	< *userSelectionForDynamicProbes* >	List< UDSProbeParams >	method. When user defines a UDS in the GUI, the user ONLY needs to specify the Design-Solution. This quantity will be assigned dynamically by matching the component-expression defined in each UDSProbeParam.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	GetInputUDSParams(<*udsParams*>, <*propList*>, <*userSelectionForDynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# Adds setup time properties to the propList`
	`def GetInputUDSParams(self, udsParams, propList, userSelectedDynamicProbes):`
	
	`  # Add the probes. We need only one double quantity`
	`  param1 = UDSProbeParams("probe1", "double quantity probe", Constants.kDoubleParamStr, "", "")`
	`  udsParams.Add(param1)`
	
	`  # Add the properties we want the user to supply`
	`  # In this case, we will ask for a start/end range for`
	`  # X parameters. Since we cannot reasonably provide defaults`
	`  # as we have no idea what the sweep limits will be, we will`
	`  # also ask if the limits are to be activated.`
	`  prop = propList.AddNumberProperty("X Min", "0")`
	`  prop.Description = "Start X value to consider"`
	
	`  prop = propList.AddNumberProperty("X Max", "1")`
	`  prop.Description = "End X value to consider"`
	
	`  # For menus, the first option is the default.`
	`  prop = propList.AddMenuProperty("Activate X Limits", ["No", "Yes"])`
	`  prop.Description = "Activate X range"`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    return True
```

GetDynamicProbes

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name <*dynamicProbes*>	Type List< UDSDynamicProbes >	Description [out] List of dynamic probes
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	GetDynamicProbes(<*dynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# output UDSDynamicProbeCollection probes`
	`    def GetDynamicProbes(self, probes):`
	`        pass`

Compute

This is the main computation method which generates the data for the quantities that make up the UDO solution.

UI Access	The data is received from the UI using IUDSInputData. It is processed and the result data is sent to the UI using IUDSOutputData.		
Parameters	Name <*inData*> <*outData*> <*propList*> <*progressMonitor*>	Type IUDSInputData IUDSOutputData IPropertyList IProgressMonitor	Description Used to get the input probe data. Used to set the UDO solution quantity and sweep data. Used to get the user entered values for each of the properties defined during the GetInputUDSParams call. This can be used to set progress for long running calculations, check for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	user initiated abort etc.
Return Value	Boolean. If true, the method was successful. If false, it was not.

Python Syntax	Compute (<*inData*>, <*outData*>, <*propList*>, <*progressMonitor*>)

Python Example	`# IUserDefinedSolutionHandle API implementation.`
	`# Calculates output values and sets them using IUDSInputData/IUDSOutputData API.`
	`def Compute(self, inData, outData, propList, progMon):`
	
	`  # Get the sweeps associated with the probe and validate`
	`  # use the probe name that we had defined earlier`
	`  sweeps = inData.GetSweepNamesForProbe("probe1")`
	`  if (sweeps == None or sweeps.Count > 1):`
	`  AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute")`
	`  return False`
	
	`  # Get the data associated with our probe`
	`  probeData = inData.GetDoubleProbeData("probe1")`
	`  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])`
	
	`  # Get the user specified properties.`
	`  # Note that ideally, these "X Min" etc names should be`
	`  # written as constant members and referred to in both`
	`  # the GetInputUDSParams and in Compute to reduce the`
	`  # change of typos.`
	`  useXRangeProp = propList.GetMenuProperty("Activate X Limits").SelectedMenuChoice`
	`  xRangeStart = propList.GetNumberProperty("X Min").ValueSI`
	`  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI`
	
	`  # At this stage, one can look at the`
	`  # RequestedQuantities and create a dictionary to later`
	`  # check against. However, I am simply computing all`
	`  # the quantities.`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
minVal = 0
maxVal = 0
avgVal = 0

# Check if we need to perform range computation
if useXRangeProp == "Yes":
  seenAny = False
  avgSum = 0
  count = 0

# zip is used since we also need to pull in sweep data
# an index and the array notation could also have been
# used
for probeVal, sweepVal in zip(probeData, sweepData):
  if sweepVal < xRangeStart or sweepVal > xRangeEnd:
    pass

# Note that in a better written script, this code
# would be refactored into its own function to
# avoid code duplication
if not seenAny:
  minVal = probeVal
  maxVal = probeVal
  avgSum = probeVal
  seenAny = True
  count = 1
else:
  if probeVal < minVal:
    minVal = probeVal

  if probeVal > maxVal:
    maxVal = probeVal

  avgSum += probeVal
  count += 1

if seenAny:
  avgVal = avgSum/count

else:
  seenAny = False
  avgSum = 0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDO Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interface for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects must be created in the Python script. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor
- IUDSInput
- IUDSOutput

IUDSInputData

The purpose of this class is to get data (probe and sweep) from Desktop. The IUDSInputData abstract class declares the following abstract methods:

- GetDoubleProbeData
- GetSweepsDataForProbe
- GetComplexProbeData
- GetSweepNamesForProbe
- GetRequiredQuantities

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetVariableValues
- GetInterpolationOrdersData

Examples in this section are just to show proper syntax of the function calls. For actual usage of the class, see the Compute function's example.

GetDoubleProbeData

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array of data for the specified probe if the probe exists or null if the probe is unknown.		

Python Syntax	GetDoubleProbeData (<*probeName*>)
Python Example	`# doubleData is a list of floats` `doubleData = inData.GetDoubleProbeData("probe1")`

GetSweepsDataForProbe

All probe data that is supplied is associated with one or more sweep (an intrinsic quantity like Time, Frequency, Theta, Phi, etc. that is swept) quantities.

UI Access	NA		
Parameters	Name <*probeName*> <*sweepName*>	Type String String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO. Name of the sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value	Double array of data for the specified probe and sweep.

Python Syntax	GetSweepsDataForProbe(<*probeName*>, <*sweepName*>)
Python Example	`# sweepData is C# Array of doubles (floats in python)` `sweepData = inData.GetSweepsDataForProbe ("FarFieldsProbe","Freq"])`

GetComplexProbeData

The primary mechanism by which the UDO retrieves data for its input probes (if it expects complex data for the probe).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array (float in Python) of data for the specified probe. Each pair of floats represents one complex number. The first value is for real part and the second value it the imaginary part. For instance, array [10.0, 0, 5.1, 2.1] represents 2 complex numbers: (10.0, 0) and (5.1, 2.1).		

Python Syntax	GetComplexProbeData(<*probeName*>)

Python Example	`# complexDataAsDouble is C# Array of doubles (floats in Python)`
	`# each pair of floats represents one complex number`
	`complexDataAsDouble = inData.GetComplexProbeData ("FarFieldsProbe")`
	`# creating a list of complex numbers from complexDataAsDouble array`
	`complexData = []`
	`if complexDataAsDouble != None:`
	`  for i in xrange(0,complexDataAsDouble.Count, 2):`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`complexData.append(complex(complexDataAsDouble [i],complexDataAsDouble[i+1]))`

GetSweepNamesForProbe

Retrieves a list of sweep quantity names associated with a given probe. This also indicates the dimensionality of the data. One name implies that the probe-data is 2D (probe-quantity vs Sweep Quantity) and two names implies 3D data (probe-quantity vs Sweep 1 X Sweep 2).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	IList<string> - list of sweep names for the current probe name.		

Python Syntax	GetSweepNamesForProbe(<*probeName*>)
Python Example	`# sweepNames is C# Array of strings` `sweepNames = inData.GetSweepNamesForProbe("FarFieldsProbe")`

GetRequiredQuantities

A given UDO can specify that it provides one or more computed quantities. The user might choose to create a report from only a few among the various available UDO outputs. This function returns the list of UDO output quantities that the user requested. Only these need be computed in the UDO's Compute method.

UI Access	NA
Parameters	None
Return Value	IList<string>

Python Syntax	GetRequiredQuantities()

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`# quantities is C# Array of strings` `quantities = inData.GetRequiredQuantities()`

GetVariableValues

This allows the UDO to obtain the names and values of all the design variables for which the UDO quantities are being requested.

UI Access	NA
Parameters	None
Return Value	IDictionary<string,string> of key-value pairs for variables. Both key and value are strings.

Python Syntax	GetVariableValues()

Python Example	`# theDict is C# Dictionary<string, string>`
	`theDict = inData.GetVariableValues()`
	`if theDict != None:`
	`  #varPair is of .Net KeyValuePair type`
	`  for varPair in theDict:`
	`  varName = varPair.Key #string`
	`  varValue = varPair.Value #string`

GetInterpolationOrdersData

Returns the interpolation orders that are associated with the probe-data. The probe data is specified at each value of the various sweeps. Any value in between the sweep data points, can use the interpolation data to get a possibly more accurate (compared to linear interpolation) inter-sweep value.

UI Access	NA		
Parameters	Name *<probeName>*	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Byte array of interpolation order for the specified probe. These are to be		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	treated as 8-bit signed integers, that is, their values range from 0-127.

Python Syntax	GetInterpolationOrdersData(<*probeName*>)

Python Example	`# interData is C# Array of bytes (integers in Python)`
	`interData = inData.GetInterpolationOrdersData(kProbeNames[0])`
	`for interValue in theDict:`
	`  # interValue and order are integers`
	`  order = interValue`

IUDSOutputData

This type is a twin of the IUDSInputData in that it is used to store the values computed by the UDO's Compute method. The IUDSOutputData abstract class declares the following abstract methods:

- SetSweepsData
- SetDoubleQuantityData
- SetComplexQuantityData

Examples in this section are just to show proper syntax is function calls. For actual usage of the class, see the Compute function example.

SetSweepsData

Each quantity that is computed by the UDO can be associated with a sweep. If it is, the values that make up the sweep's data points must be specified using this call.

UI Access	NA		
Parameters	Name	Type	Description
	<*sweepName* >	String	Name of the sweep
	<*sweepData*>	List<floats>	Sweep data for the specified sweep.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetSweepsData (<*sweepName*>,<*sweepData*>)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`sweepList = [12.3, 14.5, 16.7]` `outData.SetSweepsData("Freq", sweepList)`

SetDoubleQuantityData

This method is used to record the computed quantity data for each output that is computed. Please note that unless all the sweeps are reduced, this should be used in conjunction with SetSweepsData.

UI Access	NA		
Parameters	Name <*qtyName*> <*qtyData*>	Type String List<Floats>	Description Name of the quantity. Quantity data for the specified quantity.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetDoubleQuantityData (<*qtyName*>,<*qtyData*>)
Python Example	`doubleList = [12.3, 14.5, 16.7]` `outData.SetDoubleQuantityData("V1PlusV2", doubleList)`

SetComplexQuantityData

If the quantity computed is a complex quantity, use this method to set the quantity values. Any sweep values must be set separately via the SetSweepsData method.

UI Access	NA		
Parameters	Name <*qtyName* > <*qtyData* >	Type String List<floats>	Description Name of the quantity. Quantity data for the specified quantity. Complex numbers are passed as pairs of floats in a list.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	SetComplexQuantityData (<*qtyName*>,<*qtyData*>)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`doubleFromComplexList=[]`
	`complexList = [(1+1j), (2+4j), (9.1+3.2j)]`
	`for aComplex in complexList:`
	`    doubleFromComplexList.append(aComplex.imag)`
	`    doubleFromComplexList.append(aComplex.real)`
	`    outData. SetComplexQuantityData ("V1PlusV2", doubleFromComplexList)`

Working With Properties for UDO

A property is the unit for collecting and using input from the user that is used to influence the UDO's Compute. These are initially set up when the UDOs **GetInputUDSParams** method is called and are retrieved in the UDO's Compute method.

There are three supported property types that could be used in the UDO script:

- INumberProperty to specify number properties (with unit support).
- IMenuProperty to allow the user to select from a list of options.
- ITextProperty to allow the user to enter text.

The IPropertyList type implements a collection for these properties:

- IPropertyList Abstract class
- IProperty Abstract class
- INumberProperty Abstract class
- ITextProperty Abstract class
- IMenuProperty Abstract class

IPropertyList Abstract Class

Attributes:

- AllProperties (IEnumerable<IProperty> - see IProperty)
- NumProperties (int)

Functions:

- GetProperty(string propName): Returns a named property as an IProperty.
- GetMenuProperty (string propName): Returns the named property as an IMenuProperty.
- GetTextProperty (string propName): Returns the named property as an ITextProperty.
- GetNumberProperty (string propName): Returns the named property as an INumberProperty.
- DeleteProperty (string propName): Deletes an already added named property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- AddNumberProperty(string name, string numberWithUnits): Adds a new number property. If a property with the same name already exists, it is overwritten.
- AddTextProperty(string name, string textValue): Adds a new named text property with the supplied value. Any existing property with the same name is overwritten.
- AddMenuProperty(string name, IList<string> menuChoices): Creates a new named menu property with the supplied list of choices. The default selection is set to item 0 (the first item). Any property with the same name is overwritten.

IProperty Abstract Class

Attributes:

- Name (string)
- Description (string)
- PropType (read-only EPropType - see Constants)

Constructor:

- IProperty(string name, EPropType type)

The class is used as base class for INumberProperty, IMenuProperty, and ITextProperty.

INumberProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- ValueSI (read-only double)
- ValueInUnits (read-only double)
- Units (read-only string)
- HasUnits (read-only bool)

Constructor:

- INumberProperty(string name)

Functions:

- Set(string numberWithUnits)
- SetDouble(double number, string unitString)

ITextProperty Abstract Class

Base class:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- abstract class IProperty

Attributes:

- Text (string)

Constructor:

- ITextProperty(string name)

IMenuProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- MenuSelection (int): This represents the index into the MenuChoices list.
- SelectedMenuChoice (string): This is the item in the MenuChoices list corresponding to the MenuSelection index.
- MenuChoices (IList<string>)

Constructor:

- IMenuProperty (string name)

Example:

```
# adding data to IPropertyList propList; used in Compute function
prop = propList.AddNumberProperty('Offset 1', '0')
prop.Description = 'Trace 1 Offset'
prop = propList.AddNumberProperty("TRATE", "800 MHz")
prop.Description = "Frequency"
prop = propList.AddTextProperty("Text", "The Text")
prop.Description = "Text Property"
prop = propList.AddMenuProperty('Operation', ['Add', 'Subtract',
'Max' ,'Min','Mean'])
prop.Description = 'Operation menu'

# reading data from IPropertyList propList; used in Validate function
numOfNumberProperties = 0
if propList != None and propList.AllProperties != None:
        for prop in propList.AllProperties:
                if prop.PropType == Constants.EPropType.PT_NUMBER:
                        numOfNumberProperties ++
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Application-Specific Classes Used in Python Scripts

This section describes other classes used in Python scripts:

Constants Class

UDSProbeParams Class

UDSDynamicProbes Class

QuantityInfo Class

IProgressMonitor Abstract Class

Constants Class

The constants used in a Python script are defined in the Constants class.

Attributes:

- kDoubleParamStr : string constant used to specify *double* as the type of a quantity
- kComplexParamStr: string constant used to specify *complex* as the type of a quantity
- Enum EPropType: (used to set property type)
  - EPropType.PT_NUMBER
  - EPropType.PT_TEXT
  - EPropType.PT_MENU

Example:

```
paramType = Constants.kDoubleParamStr
propType = Constants.EPropType.PT_NUMBER
```

UDSProbeParams Class

This class defines which data quantity that a UDO will be pulled from the design to compute the UDO output (results). The objects of this class must be created in a Python script with the GetInputUDSParams function. They are supplied to the Validate function if implemented.

Attributes:

- ProbeName (read-only string)
- ProbeDescription (read-only string)
- ParamType (read-only string)
- ReportTypeName (read-only string)
- ComponentExpression (read-only string)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constructor: UDSProbeParams(string probeName, string probeDescription, string paramType, string reportTypeName, string componentExpression);

- probeName - required.
- probeDescription - optional (can be empty string).
- paramType - required; can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- reportTypeName - optional (can be empty string)
- ComponentExpression - optional (can be empty string)

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
```

UDSDynamicProbes Class

Attributes:

- UDSParam (read-only UDSProbeParams)
- UserSelectedProbeName (read-only string)

Constructor: UDSDynamicProbes (UDSProbeParams udsParam, string userSelectedProbeName)

- udsParam - required
- userSelectedProbeName - required

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
selectedName = "probe1"
udsDynamicProbParam = UDSDynamicProbes(udsProbParam, selectedName )
```

QuantityInfo Class

Attributes:

- ParamType (read-only string)
- FullUnitType (read-only string)

Constructors:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- QuantityInfo(string paramType)
- QuantityInfo(string paramType, string fullUnitType)

Parameters:

- paramType can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- fullUnitType is a case insensitive string representing full unit type. It is not defined in Constants. Instead you can use any of the units in string representation - for example, "mm" or"ghz".

Example:

```
quantityInfo1 = QuantityInfo(Constants.kDoubleParamStr)
quantityInfo2 = QuantityInfo(Constants.kDoubleParamStr,"ghz")
```

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

Using .NET Collection Classes and Interfaces in Python Scripts

Some of the API functions specified above use .Net collection classes and interfaces, that is, Array class, IList interface, IEnumerable interface, and IDictionary interface. The following section describes how to work with the .Net collection objects in Python scripts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.NET Array, IEnumerable, and IList objects can be indexed and iterated over as if they were Python lists. You can also check for membership using 'in'. To get .Net Array and IList sizes you can use Python's 'len' or .Net 'Count'.

Example:

Getting size:

```
arraySize = doubleDataArray.Count
   arraySize = len(doubleDataArray)
listSize = sweepsNamesList.Count
 listSize = len(sweepsNamesList)
```

Iterating:

```
for sweep in sweepsNamesList:
   print sweep
for in in xrange(listSize)
      print sweepsNamesList[i]
```

Checking for membership:

```
if 'Time' in sweepsNamesList:
   doThis()
  else:
   doThat()
```

For .NET IDictionary, the same as for Array and IList, you can get size with 'len' or 'Count' and check for membership of the keys using 'in'. Getting values for the keys also works the same way as in Python 'dict'.

Example:

Getting size:

```
varValuesSize = varValues.Count
 varValuesSize = len(varValues)
```

Checking for membership:

```
if 'offset' in varValues:
   print varValues['offset']
```

Getting value:

```
if 'offset' in varValues:
   offsetValue = varValues['offset']
```

As for iteration .NET Dictionary is different from Python dict. While iterating, python dict will return keys, .Net Dictionary will return .Net KeyValuePair.

Example:

Iterating:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for .NET IDictionary:

```
for varPair in varValues: #varPair is of .Net KeyValuePair type
     varName = varPair.Key
    varValue = varPair.Value
```

for Python dict:

```
for varName in varValues:
   varValue = varValues[varName]
```

You can use Python types instead of .NET types if you prefer. For this, you need to cast .NET array and .Net iList to Python list type and .NET Dictionary to Python dict type.

Casting should not be used for data arrays - it can be extremely costly for the memory usage as well as time consuming.

Example:

```
aPythonList = list(dotNetArray)
  aPythonList = list(dotNetList)
   aPythonDict = dict(dotNetDictionary)
```

User Defined Outputs: Messaging Methods

Messaging methods are provided to convey additional information to the user from any of the UDOs methods. The Compute function is the one typically location where such use is anticipated. Any message sent via these functions are displayed in the application's message window using the appropriate icon.

These functions can also be used for debugging purposes.

- AddErrorMessage
- AddInfoMessage
- AddWarningMessage

Example script:

```
############################################################
# Imports
############################################################
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
class UDOExtension(IUDOPluginExtension):
  def __init__(self):
    pass

  #--- IDA IUDOPluginExtension -----------------------
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
def GetUDSName(self):
  return "MinMaxAvg"

#--- ISA IUDOPluginExtension ------------------------
def GetUDSDescription(self):
  return "Sample UDO for dimension reducing quantities"

#--- ISA IUDOPluginExtension ------------------------
# Returns list of category names
 def GetCategoryNames(self):
  return ["UDOOutputs"]

#--- ISA IUDOPluginExtension ------------------------
# returns a list of quantity names for the supplied category name
def GetQuantityNames(self, catName):
  if catName == "UDOOutputs":
    return ["min_val", "max_val", "avg_val"]
  else:
    return []

#--- ISA IUDOPluginExtension ------------------------
# Returns an instance of QuantityInfo for the qtyName supplied or
# None if such a quantity could not be found
def GetQuantityInfo(self, qtyName):
  # All the quantities we have are simple doubles
  # we can leave them unitless
  return QuantityInfo(Constants.kDoubleParamStr)

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# Adds setup time properties to the propList
def GetInputUDSParams(self, udsParams, propList,
userSelectedDynamicProbes):
  # Add the probes. We need only one double quantity
  param1 = UDSProbeParams("probe1",
    "double quantity probe",
    Constants.kDoubleParamStr,
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      "", "")
    udsParams.Add(param1)


    # Add the properties we want the user to supply
    # In this case, we will ask for a start/end range for
    # X parameters. Since we cannot reasonably provide defaults
    # as we have no idea what the sweep limits will be, we will
    # also ask if the limits are to be activated.
    prop = propList.AddNumberProperty("X Min", "0")
    prop.Description = "Start X value to consider"

    prop = propList.AddNumberProperty("X Max", "1")
    prop.Description = "End X value to consider"

    # For menus, the first option is the default.
    prop = propList.AddMenuProperty("Activate X Limits", ["No",
    "Yes"])
    prop.Description = "Activate X range"

    return True

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# output UDSDynamicProbeCollection probes
def GetDynamicProbes(self, probes):
    pass

#--- ISA IUDOPluginExtension ------------------------
# Returns list of sweeps names
# We have no sweeps as we reduce them.
def GetUDSSweepNames(self):
    return []

#-----------------------------------------------------------------
# IUserDefinedSolutionHandle API implementation.
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Calculates output values and sets them using
IUDSInputData/IUDSOutputData API.
def Compute(self, inData, outData, propList, progMon):

  # Get the sweeps associated with the probe and validate
  # use the probe name that we had defined earlier
  sweeps = inData.GetSweepNamesForProbe("probe1")
  if( sweeps == None or sweeps.Count > 1):
    AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or >
    1 in Compute")
    return False

  # Get the data associated with our probe
  probeData = inData.GetDoubleProbeData("probe1")
  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])

  # Get the user specified properties.
  # Note that ideally, these "X Min" etc names should be
  # written as constant members and referred to in both
  # the GetInputUDSParams and in Compute to reduce the change
  # of typos.
  useXRangeProp = propList.GetMenuProperty("Activate X
  Limits").SelectedMenuChoice
  xRangeStart = propList.GetNumberProperty("X Min").ValueSI
  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI

  # At this stage, one can look at the RequestedQuantities and
  # create a dictionary to later check against. However, I am
  # simply computing all the quantities.
  minVal = 0
  maxVal = 0
  avgVal = 0

  # Check if we need to perform range computation
  if useXRangeProp == "Yes":
    seenAny = False
    avgSum = 0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    count = 0


    # zip is used since we also need to pull in sweep data
    # an index and the array notation could also have been used
    for probeVal, sweepVal in zip(probeData, sweepData):
                if sweepVal < xRangeStart or sweepVal > xRangeEnd:
        pass


                # Note that in a better written script, this code
      would be
                # refactored into its own function to avoid code
                # duplication
                if not seenAny:
        minVal = probeVal
        maxVal = probeVal
        avgSum = probeVal
        seenAny = True
        count = 1
                else:
        if probeVal < minVal:
          minVal = probeVal
        if probeVal > maxVal:
          maxVal = probeVal
        avgSum += probeVal
        count += 1
    if seenAny:
                avgVal = avgSum/count
else:
    seenAny = False
    avgSum = 0
    for probeVal in probeData:
                if not seenAny:
        minVal = probeVal
        maxVal = probeVal
        avgSum = probeVal
        seenAny = True
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            else:
        if probeVal < minVal:
          minVal = probeVal
        if probeVal > maxVal:
          maxVal = probeVal
        avgSum += probeVal
    if seenAny:
            avgVal = avgSum/probeData.Count


# Finally set the output values. Note that these are always set
# as lists even if we have just one item.
outData.SetDoubleQuantityData("min_val", [minVal])
outData.SetDoubleQuantityData("max_val", [maxVal])
outData.SetDoubleQuantityData("avg_val", [avgVal])


# And we are done.
return True
```

AddErrorMessage

Call this method to convey an error condition to the user.

UI Access	NA		
Parameters	Name <message>	Type String	Description Error message
Return Value	None		

Python Syntax	AddErrorMessage (<*message*>)
Python Example	`AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute"))`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AddInfoMessage

Call this method to convey an informational message to the user. This is the call to use when outputting messages for debugging purposes.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddInfoMessage (<*message*>)
Python Example	`AddInfoMessage("Enter starting X value.")`

AddWarningMessage

Call this method to convey a warning message, typically used for conditions that are not ideal but can be tolerated by the script.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddWarningMessage (<*message*>)
Python Example	`AddWarningMessage("Unexpected sweep count in Compute")`

User Defined Outputs: Script Organization

As described in the Introduction section, the UDO scripts should all reside under the **UserDefinedOutputs** folder under either of the three library locations (system, user, or personal).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Script Libraries

If you decide that you need base classes, additional data files, and etc., to organize your UDOs better, you can do so. This type of library organization allows code reuse between similar UDOs and can be very helpful. There is special support provided for this type of script-library organization:

- **All script-library and other support files need to be in a *Lib* sub-directory under the *UserDefinedOutputs* directory**. Any *.py* files found in such **Lib** directories are ignored and not displayed in the GUI as a valid UDO choice.
- For a UDO script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level *UserDefinedOutputs* directory can be imported).

Using Additional .NET Assemblies

Because the UDO functionality uses IronPython, we have access to the full .NET eco system. If needed, any subset of the UDO functionality can be implemented in any .NET language and used by the UDO script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any **Lib** directory upstream of the UDO script location: that is, if you have your UDO script in *C:\Users\x\PersonalLib\UserDefinedOutputs\a\b\c\myudo.py* and have a .NET assembly called *com.Acme.UDOLib* You can keep the .NET assembly under:
    - UserDefinedOutputs\Lib
    - UserDefinedOutputs \a\Lib
    - UserDefinedOutputs \a\b\Lib
    - UserDefinedOutputs\a\b\c\Lib
3. Add the following lines to your Python script:
    - `Import clr`
    - `clr.AddReference("com.Acme.UDOLib")`
    - `import com.Acme.UDOLib` -or- `from com.Acme.UDOLib import * etc`

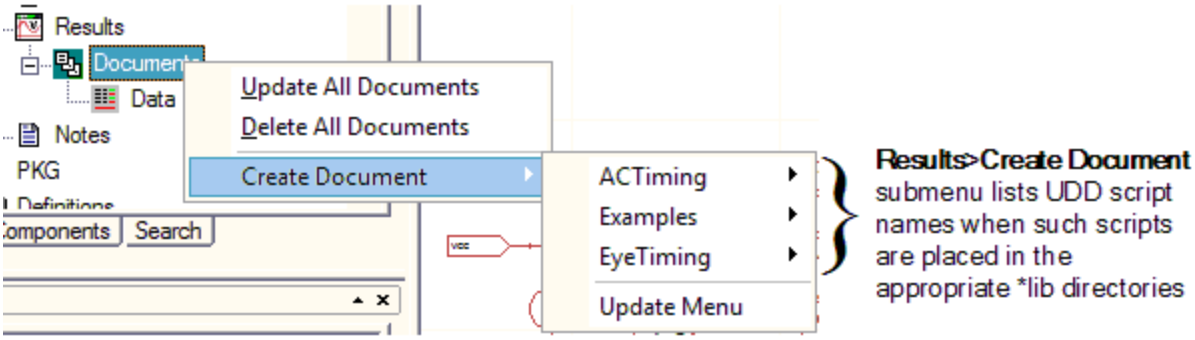
If for some reason you cannot place the .NET assemblies into a Lib directory under *UserDefinedOutputs*, you need to do a couple more steps before step 3 listed above.

```
Import sys
sys.path.append("full path to your .NET assembly location")
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Documents (UDDs)

User-defined documents (UDDs) are custom reports that you define through IronPython scripts. Once placed in a Lib directory, you can access the scripts via the **Create Document** command. The scripts describe a *Create User Defined Document* dialog box that lets you specify trace and solution inputs. After you confirm your input selections, XML, HTML, and PDF documents are generated. A web browser window opens to display the generated HTML file. The created document appears in the Project Manager, under Results in the Documents folder.



The general UDD process flow is as follows:

The UDD Python scripts must be placed in the **UserDefinedDocuments** directory under either of **syslib**, **userlib**, or **Personallib** with any subdirectory structure needed. The Lib directory can contain Python scripts that have common code that other scripts can use.

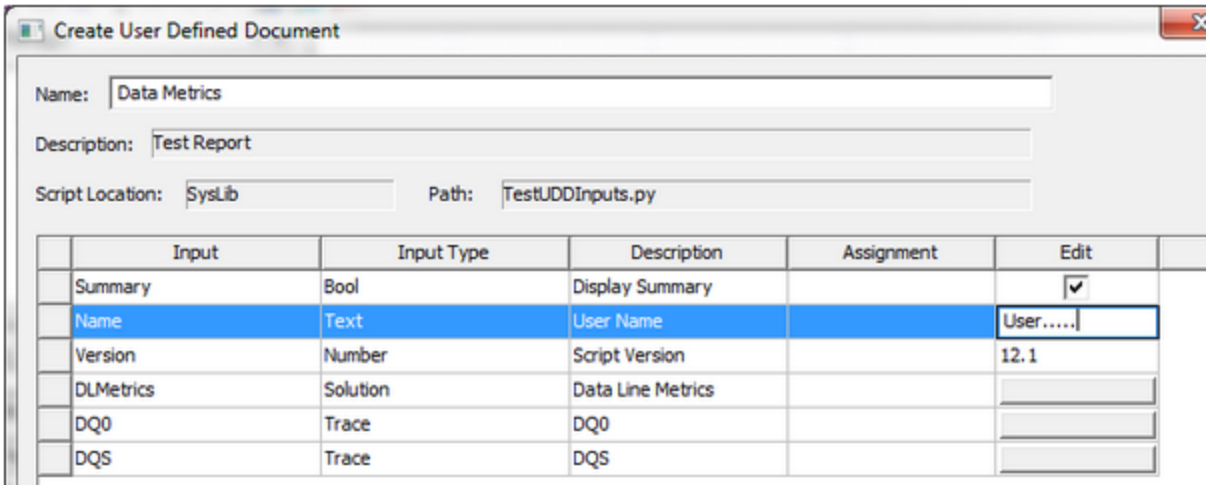
Use **Mechanical** > **Results** > **Create Document**> **Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib, or Personallib, or to exclude them if they have been deleted after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments, or Personallib/UserDefinedDocuments are available through the **Mechanical** > **Results** > **Create Document** menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create User-Defined Document Inputs

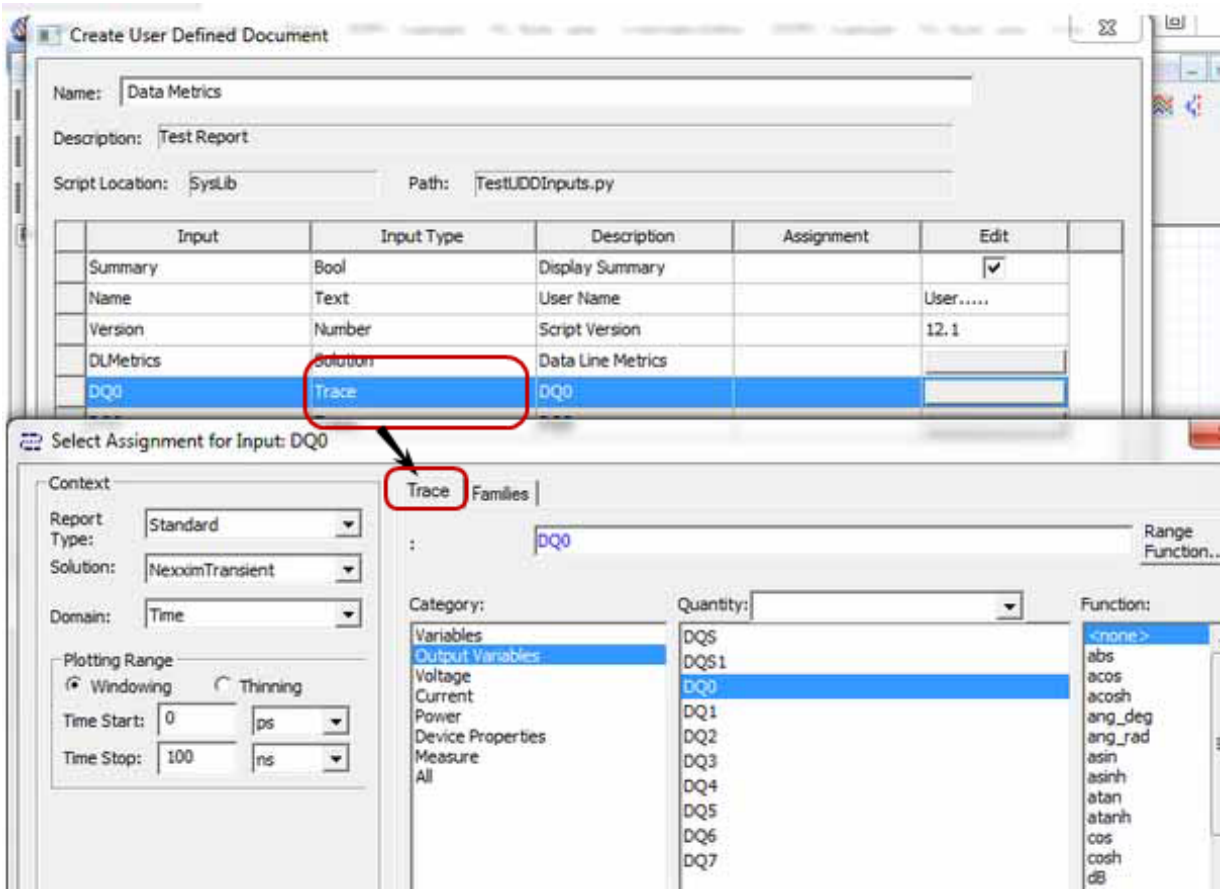
User-defined documents allow data from traces, solutions, and report types as inputs. A UDD can specify the named inputs, which you select or enter the values of in the *Create User Defined Document* dialog box that displays when you run **Mechanical** > **Results** > **Create Document**> <*scriptName*>.



Input Types can be of Boolean, number, text, trace, or solution type. The boolean, number, and text type can be given a default value that you can interactively override when the document is created or modified. For example, you can select a trace when you create or modify a UDD document. The trace data is available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the *Reporter* dialog box the report type (Standard, Eye Diagram, User Defined), solution name, context, and the quantity for which you want the trace data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

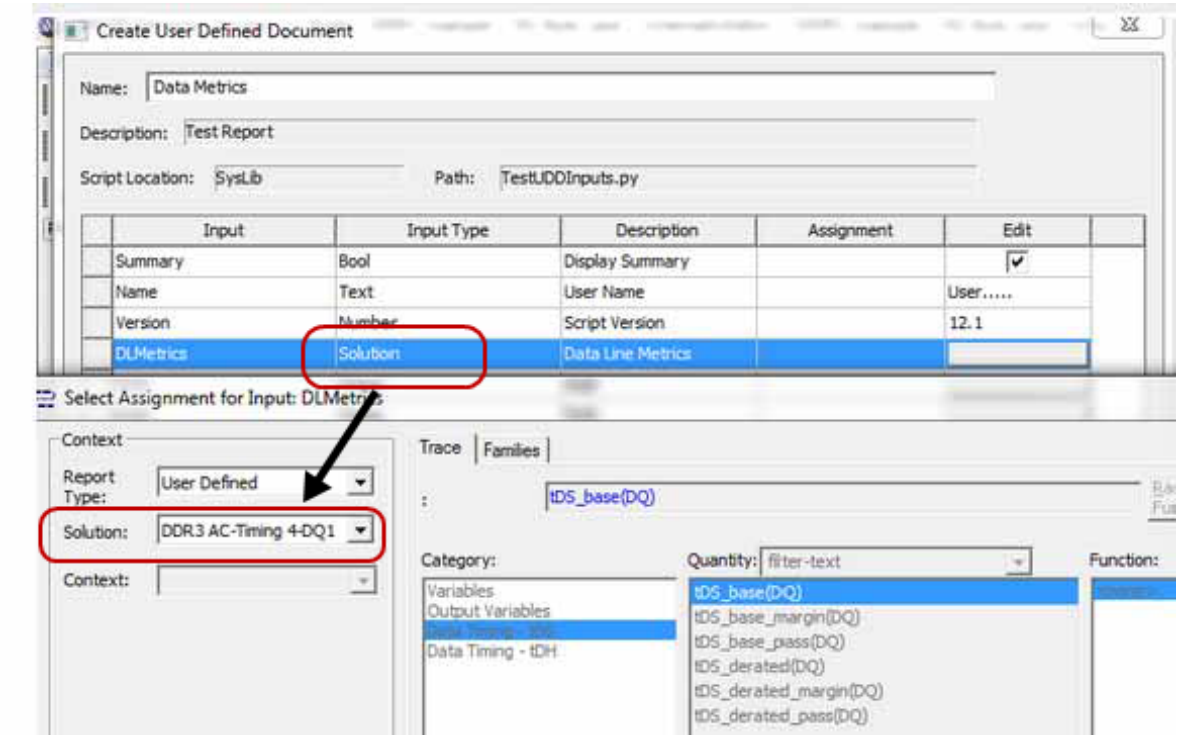


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety is now available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the reporter dialog the report type (Standard, Eye Diagram, User Defined), solution name, and context. A specific quantity cannot be selected since data for all quantities in the solution are available.

> **Note:** The Category/Quantity/Function portion of the dialog is disabled for user input.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

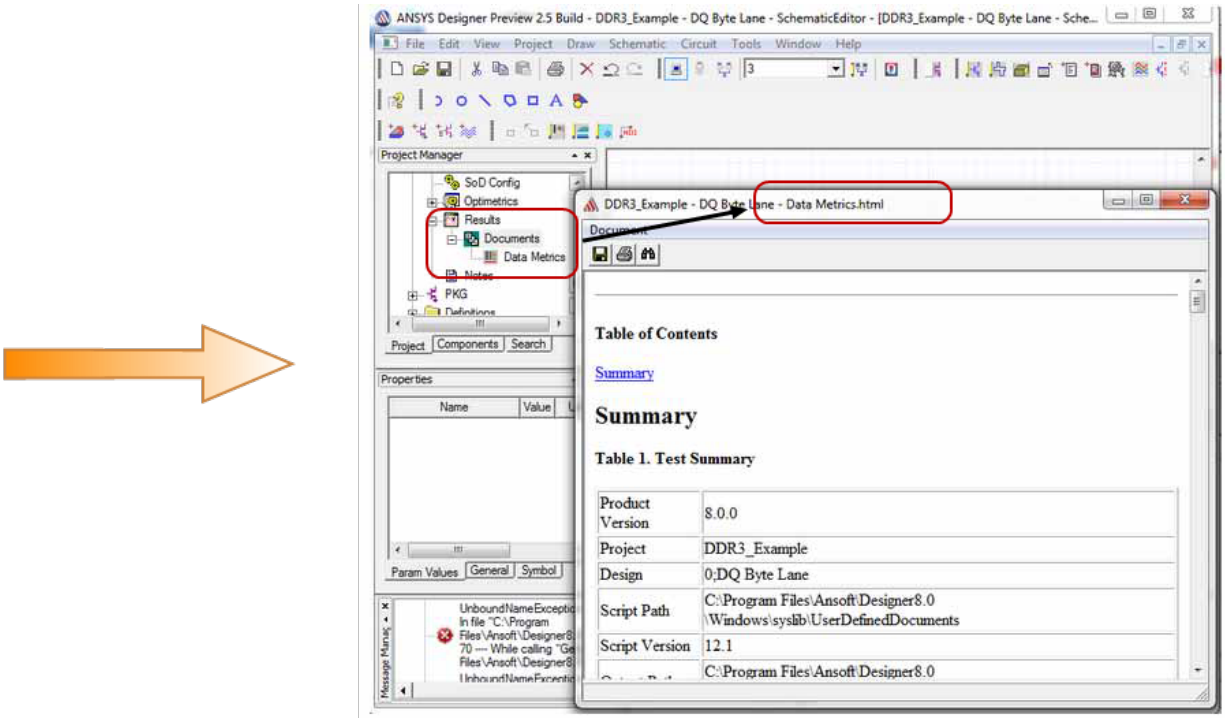


UDD Document Creation and Display

After all the input selections for a UDD are confirmed, based on the script, XML, HTML, and PDF documents are generated based on the inputs provided by the user. (The XML, HTML, and PDF generation is based on specific calls in the Python script, which are explained in a following section.) A web browser window also opens to display the generated HTMLfile.

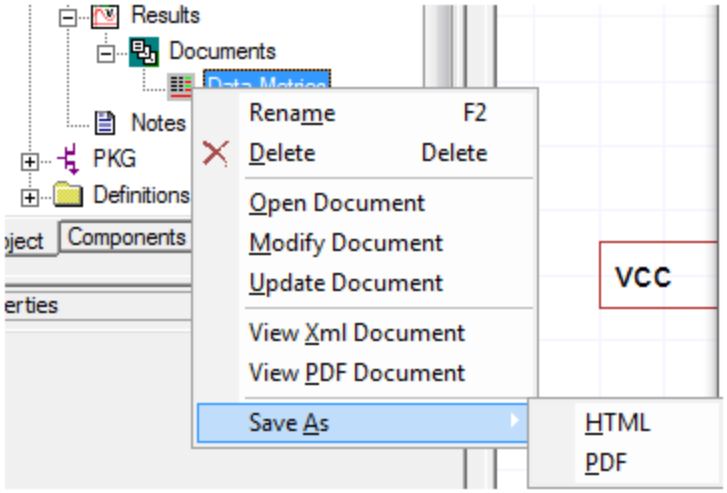
The created document will be placed under a new folder named **Documents** under the **Results** folder. All documents that are created by the user for the design will be placed under this folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Managing Documents Listed in the Project Window Under Results

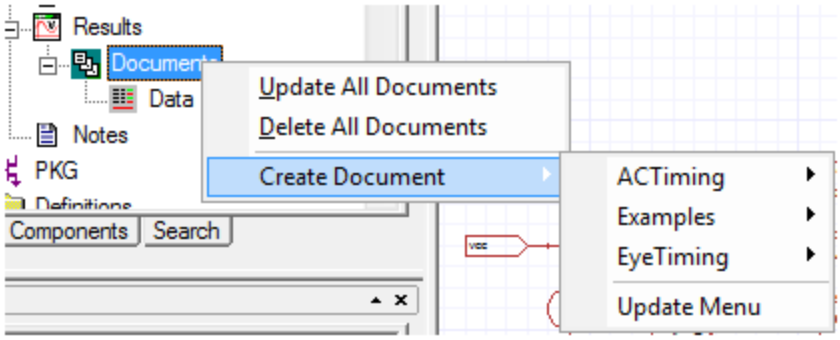
Right-click a user-defined document displayed in the Project Manager tree to bring up a menu where you can rename, delete the document. **Open document** opens the web browser with the html document. **Modify document** opens the setup dialog box, where you can change the selections for the input. To view the XML and the PDF document simply choose the appropriate menu items. There is also a menu item to save the document in a different location.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Documents Folder's Context Menu

Right-click the documents folder has the menu options to **Update All Documents** or **Delete All Documents**. It also provides the option of creating a document from here.

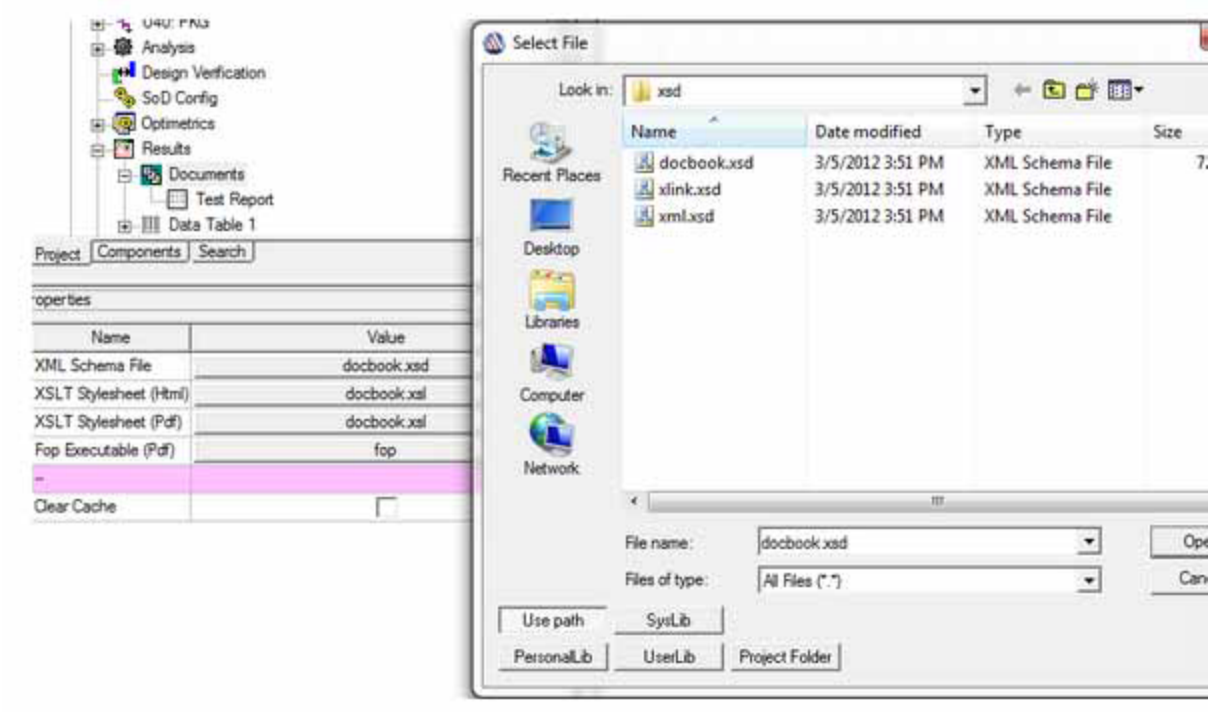


Document Folder's Property Window

When the documents folder is selected, the *Property* window shows the following properties:

- **XML Schema File** - File path to the XML schema file.
- **XSLT StyleSheet (HTML)** - File path to the XSLT stylesheet file used for HTML generation.
- **XSLT StyleSheet (Fo)** - File path to the XSLT stylesheet file used for PDF generation.
- **Fop Executable (PDF)** - File path the Fop executable used for PDF generation.
- **Clear Cache** - Clears the cached XSL transform object and forces creation of a new one. (The caching is done to save time during document generation, so subsequent generation or update of the document can use the cached transform object. But sometimes you may want to force a recompile of the document if you change the stylesheet).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The XML, HTML, and PDF generation require the XML schema file and XSLT stylesheets to generate proper output. In addition, the PDF generation requires a FOP executable. You can use the defaults provided in the installation or provide the file paths of your own preferred stylesheets and fop executable installed in his machine.

Viewing UDDs with an HTML Web Browser

The XML and HTML documents can be viewed in a web browser with some basic functionality like printing the document, searching the document for a phrase or sentence, and saving the document.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

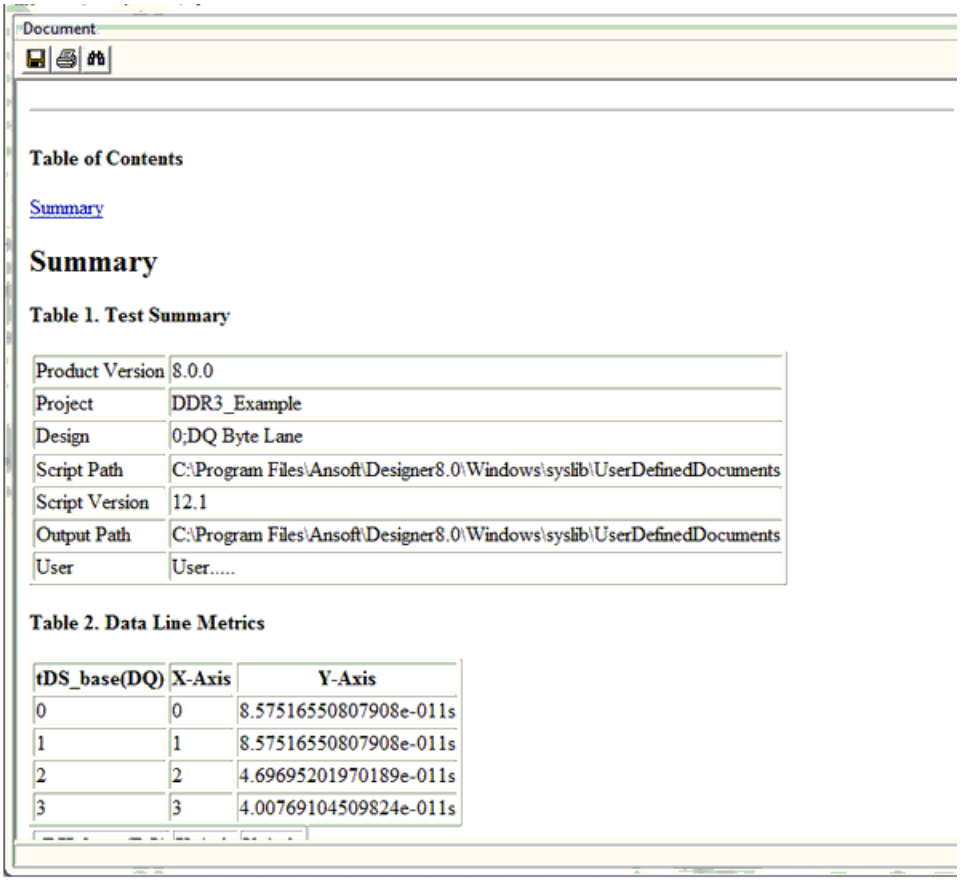

Document

Table of Contents

Summary

Summary

Table 1. Test Summary

Product Version	8.0.0
Project	DDR3_Example
Design	0;DQ Byte Lane
Script Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
Script Version	12.1
Output Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
User	User.....

Table 2. Data Line Metrics

tDS_base(DQ)	X-Axis	Y-Axis
0	0	8.57516550807908e-011s
1	1	8.57516550807908e-011s
2	2	4.69695201970189e-011s
3	3	4.00769104509824e-011s

UDD Script Libraries

Base classes and data files shared between similar UDDs can be organized to reuse the code in a better way. All script-library and other support files need to be in a Lib sub-directory under the UserDefinedDefinitions directory. Any .py files found in such Lib directories are ignored and not displayed in the GUI as a valid UDD choice. For a UDD script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level UserDefinedDefintions directory can be imported).

The UDD functionality uses IronPython so we have access to all the .NET assembles. If needed, any subset of the UDD functionality can be implemented in any .NET language and used by the UDD script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any Lib directory upstream of the UDD script location: that is, if you have your UDD script in C:\Users\x\PersonalLib\UserDefinedDefintions\a\b\c\myudd.py and have a .NET

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assembly called com.Acme.UDDLib You can keep the .NET assembly under:

- UserDefinedDefintions\Lib
- UserDefinedDefintions\a\Lib
- UserDefinedDefintions\a\b\Lib
- UserDefinedDefintions\a\b\c\Lib

3. Add the following lines to your Python script:

```
Import clr

clr.AddReference("com.Acme.UDDLib")

import com.Acme.UDDLib -or- from com.Acme.UDDLib import * etc
```

If for some reason you cannot place the .NET assemblies into a Lib directory under UserDefinedDefintions, you need to do a couple more steps before step 3 listed above:

```
Import sys

sys.path.append("full path to your .NET assembly location")
```

User-Defined Documents: Python Script API

A User-Defined Documents (UDD) extension is implemented as an IronPython script that defines a class with a specific name: UDDExtension which derives from a specific base class IUDDPluginExtension and implements its abstract methods.

This API supports multiple data types in the forms of constants and classes. It also has several input interfaces.

User-Defined Document scripting commands are provided in this product's scripting guide. An complete example with a line by line explanation of how to use these methods is available.

Import Statements

The base class to be used and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDD script: the following import statements should be added to the top of the python script:

```
from Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Data
import *
from
Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces
import *
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDD Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interfaces for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects of the C# class must be created in the Python script using their C# constructors. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor Abstract Class
- UDDExtension Class
- UDDInputData class
- UDDInputParams class

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

IUDDPluginExtension Abstract Class

The **IUDDPluginExtension** abstract class declares the following abstract methods which must be implemented in the UDDExtension class or one of its base classes. If any of these methods are not implemented, a run-time error will occur and the UDD will not function.

- GetUDDName
- GetUDDDescription
- ShowDefaultSetupDialog
- GetUDDInputParams
- Generate

This class also provides for the following optional methods:

- SetupUDDInputParams
- HandleUDDEvents
- GetUDDSchema
- GetUDDStyleSheetForHtml
- GetUDDStyleSheetForPdf
- GetFopExecutable
- GetUDDAppContext
- GetUDDDesignContext

GetUDDName

Retrieves a prefix to use for all solution instances created using this UDD.

UI Access	NA
Parameters	None
Return Value	String that is the prefix

Python Syntax	GetUDDName()

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`def GetUDDName(self):` `    return "MinMaxAvg"`

GetUDDDescription

Retrieves a description for the UDD, its purpose, etc.

UI Access	NA
Parameters	None
Return Value	String that is the description

Python Syntax	GetUDDDescription()
Python Example	`def GetUDDDescription(self):` `    return "Sample UDD"`

ShowDefaultSetupDialog

Retrieves whether to show the default setup dialog box. If not, the user might want to implement or show a customized setup dialog.

UI Access	NA
Parameters	None
Return Value	Boolean. If true, the default dialog should be shown. If false, the user does not want to see the default dialog.

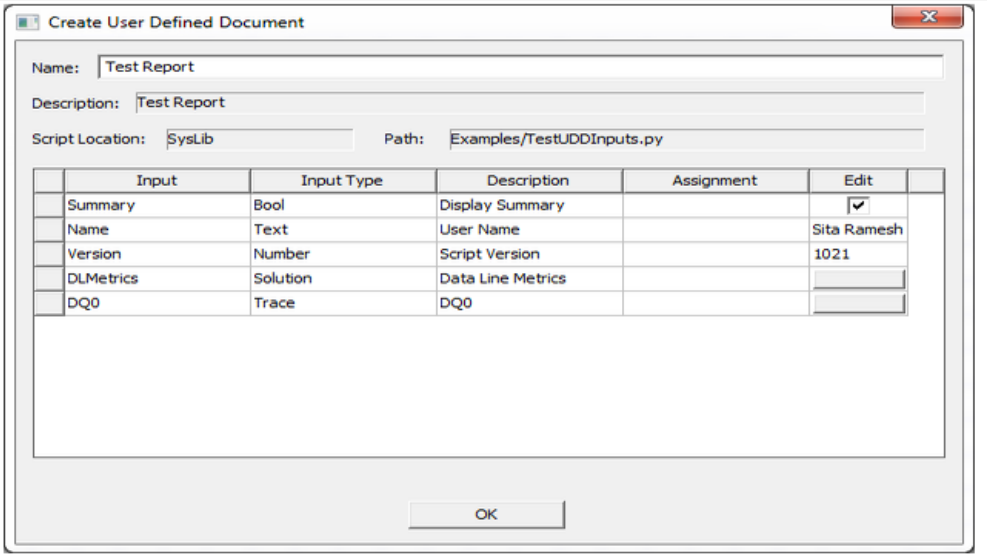
Python Syntax	ShowDefaultSetupDialog()
Python Example	`def ShowDefaultSetupDialog(self):` `    return True`

GetUDDInputParams

Retrieves inputs parameters for the User Defined Document. These parameters are the input's name, description, and data type. The UDD user specifies the value of the inputs through a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog or within the script.

UI Access	Depending on the input parameters, the following dialog may be displayed when you click **Mechanical**> **Results**> **Create Document**. The name and description of the UDD are also displayed in this dialog. 
Parameters	Name: *<uddInputs>* Type: List< UDDInputParams >* Description: The UDD script is expected to add one instance of UDDInputParams for each input definition that it will to display. When creating the UDD, the user will assign a matching value to each such input.
Return Value	Boolean. If true, the method was successful. If false, it was not.

Python Syntax	GetUDDInputParams(*<probeName>*)

Python Example	`def GetUDDInputParams(self, uddInputs)`
	`    # Boolean input`
	`    param1 = UDDInputParams("Summary","Display Summary", Constants.kBoolTypeStr, True)`
	`    uddInputs.Add(param1)`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    # Text input
    param2 = UDDInputParams("Name","User Name",
Constants.kTextTypeStr, "Sita Ramesh")
    uddInputs.Add(param2)
    # Number input
    param3 = UDDInputParams("Version","Script Version",
Constants.kNumberTypeStr, 1021)
    uddInputs.Add(param3)
    # Solution input
    param5 = UDDInputParams("DLMetrics","Data Line
Metrics",Constants.kSolutionTypeStr)
    uddInputs.Add(param4)
    # Trace input
    param5 = UDDInputParams
("DQ0","DQ0",Constants.kTraceTypeStr)
    uddInputs.Add(param5)
    return True
```

Generate

This is the main method which accesses the data from the uddInputs and generates the document.

UI Access	NA		
Parameters	Name	Type	Description
	<uddInputs>	List< UDDInputData>	The list of inputs that the user setup in the dialog box. They are now available to query for data.
	<generator>	IUDDGenerator	This is the document generator object which used to create different elements of the document like titles, sections, tables, images and write the data too. See the Document Generator Interfaces.
	<progressMonitor>	IProgressMonitor	This can be used to set progress for long running calculations and check for any user-initiated aborts.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python	Generate(*<uddInputs>*, *<generator>*, *<progressMonitor>*)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Syntax	

Python Example	def Generate(self, input, docgen, progMon):
	
	    # Gather data from inputs
	    boolinput = input[0].Data()
	    textinput = input[1].Data()
	    dblinput = input[2].Data()
	
	    # Get document root
	    docroot = docgen.GetDocumentRoot()
	
	    # Add Section
	    section1 = docroot.AddSection("Summary", "Overall Results ")
	
	    # Add a table
	    table1 = section1.AddTable("Test Summary")
	
	    #Add a table group with 2 columns
	    tgroup1 = table1.AddTableGroup(2)
	
	    # get desktop application
	    oApp = self.GetUDDAppContext()
	    if oApp != None:
	        oDesktop = oApp.GetAppDesktop()
	    if oDesktop != None:
	        # version number
	        version = oDesktop.GetVersion()
	        text1 = tgroup1.AddContent()
	        text1 .Add(0, "Product Version")
	        text1 .Add(1, version)
	
	    oProject = oDesktop.GetActiveProject()
	    if oProject != None:
	        projectname= oProject.GetName()
	        text1 = tgroup1.AddContent()
	        text1 .Add(0, "Project")
	        text1 1.Add(1, projectname)
	
	    oDesign = self.GetUDDDesignContext()
	    if oDesign != None:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        designname = oDesign.GetName()
        text1 = tgroup1.AddContent()
        text1 .Add(0, "Design")
        text1 .Add(1, designname)

    # Provides a script path
    scriptpath = docgen.GetScriptPath()
    text1 = tgroup1.AddContent()
    text1 .Add(0, "Script Path")
    text1 .Add(1, scriptpath)

    #Provides the script version
    text1 = tgroup1.AddContent()
    text1 .Add(0, "Script Version")
    text1 .Add(1, str(dblinput))

    #Provides the output xml path
    outputpath = docgen.GetOutputFilePath()
    text1 = tgroup1.AddContent()
    text1.Add(0, "Output Path")
    text1.Add(1, outputpath)

    #Provides the user information
    text1 = tgroup1.AddContent()
    text1 .Add(0, "User")
    text1 .Add(1, textinput)

    # Generate Xml output
    docgen.Write(False)

    # Generate Html output
    docgen.WriteHTML()

    # Generate PDF output
    docgen.WritePDF()

    return True
```

SetupUDDInputParams

This optional method displays a customized dialog and returns the user choices for the input params.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name *<uddInputs>*	Type List<UDDInputParams>	Description .NET list of UDDInputParams objects with values for each of them. These can be the user choice for each input obtained through a custom dialog or some other non-graphical assignment. This method cannot process trace and solution types of input with a custom dialog because there is no way of assigning solution data to the input without the invocation of the reporter dialog.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetupUDDInputParams(*<uddInputs>*)

Python Example	`def SetupUDDInputParams(self, uddInputs)`
	`    udddialog = BaseExampleUDDDialog()`
	`    if udddialog.ShowDialog() == Forms.DialogResult.OK:`
	`        # Boolean input`
	`        param1 = udddialog.GetInput("Summary")`
	`        uddInputs.Add(param1)`
	
	`        # Text input`
	`        param2 = udddialog.GetInput("Name")`
	`        uddInputs.Add(param2)`
	
	`        # Number input`
	`        param3 = udddialog.GetInput("Version")`
	`        uddInputs.Add(param3)`

HandleUDDEvents

This optional method is the event handler for all link events set by the SetEventLink() method on a IUDDText. Refer to the definition of the IUDDText object in the Document Generator Interface. The tags associated with the event are received by plugin using this abstract class.

UI Access	NA

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters	Name Type Description <*eventTags*> List<string> Event tags.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.

Python Syntax	HandleUDDEvents(<*eventTags*>)

Python Example	`def HandleUDDEvents(self, uddLinks):`
	`  if uddLinks[0] == "Open Report":`
	`  # Get Design Name`
	`  oDesign = self.GetUDDDesignContext()`
	`  if oDesign != None:`
	`  oDesign.OpenReport(uddLinks[1])`
	`  return True`

GetUDDSchema

This optional method retrieves the file path of the schema to validate the XML. This will override the default schema used.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the schema.

Python Syntax	GetUDDSchema()

Python Example	`def GetUDDSchema(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\schema\\xsd\\docbook.xsd"`
	`schemaPath = pluginExt.GetUDDSchema()`

GetUDDStyleSheetForHtml

This optional method retrieves the file path of the style sheet used to generate the HTML document. This will override the default stylesheet for HTML.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the style sheet.

Python Syntax	GetUDDStyleSheetForHtml()

Python Example	`def GetUDDStyleSheetForHtml(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\"`
	`styleSheet = pluginExt.GetUDDStyleSheetForHtml()`

GetUDDStyleSheetForPdf

This optional method retrieves the file path of the style sheet used to generate the PDF document. This will override the default stylesheet for PDF.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the style sheet.

Python Syntax	GetUDDStyleSheetForPdf()

Python Example	`def GetUDDStyleSheetForPdf(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\xsl\\fo\\docbook.xsl"`
	`styleSheet = pluginExt.GetUDDStyleSheetForPdf()`

GetFopExecutable

This optional method retrieves the file path of the fop executable used to generate the PDF document.

UI Access	NA

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters	None
Return Value	String containing the full file path of the fop executable.

Python Syntax	GetFopExecutable()

Python Example	`def GetFopExecutable(self):`
	`    return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\ApacheFOP\\fop-1.0\\fop"`
	`fopExe = pluginExt.GetFopExecutable()`

GetUDDAppContext

This optional method retrieves the UDD Owner if it is set. This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the desktop app.

UI Access	NA
Parameters	None
Return Value	the IDispatch for the desktop object

Python Syntax	GetUDDAppContext()
Python Example	`oDesktop = self.GetUDDAppContext()`

GetUDDDesignContext

This optional method retrieves the UDD Owner if it is set. This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the design.

UI Access	NA
Parameters	None

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value	IDispatch for the specified design object

Python Syntax	GetUDDDesignContext()
Python Example	`oDesign = self.GetUDDDesignContext()`

UDDExtension Class

The UDD itself should be implemented as an IronPython class called **UDDExtension** which must derive from the IUDDPluginExtension abstract base class (from the **Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDDs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDD class must derive from **IUDDPluginExtension**.

Example:

```
def BaseClassUDD ((IUDDPluginExtension):
#base class implementation
…
def UDDExtension ((BaseClassUDD):
#UDD class implementation
…
```

> **Note:**
>
> This class is modeled after the UDO class UDOExtension, therefore the usage is similar.

UDDInputData Class

This class contains user input data and can be used in the generated document. UDDInputData has the constructor UDDInputData(string name) and the following properties accessors:

Property accessor	Purpose	Form
Name	Get/Set the name of an input.	string Name();
Type	Get/Set the type of an input	string Type();

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data can be any of the following types, based on the UDDInputParams type specified in the script.

- UDDInputBool
- UDDInputDouble
- UDDInputSolution
- UDDInputText
- UDDInputTrace

UDDInputBool

Method purpose	Return Value	Data type of the returned value in C#
Retrieves boolean value of the input data.	a boolean value of input data	string Data();

UDDInputDouble

Method purpose	Return Value	Data type of the returned value in C#
Retrieves double data.	a double value of input data	double Data();

UDDInputSolution

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data for a probe	Data collection	IDictionary<double, double> DoubleData (string name);
Retrieves x and y double data for a probe for a particular variation	Data collection	IDictionary<double, double> DoubleData (string name, IDictionary<string, string> variation);
Retrieves x and y complex data for a probe	Data collection	IDictionary<double, double[]> ComplexData (string name);
Retrieves x and y complex data for a probe for a particular variation	Data collection	IDictionary<double, double[]> ComplexData (string name, IDictionary<string, string> variation);
Retrieves x and y text data for a probe	Data collection	IDictionary<string, string> TextData(string name, IDictionary<string, string> variation);

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retrieves x and y text data for a probe for a particular variation	Data collection	IDictionary<string, string> TextData(string name);
Retrieves a list of category names	A string collection	IList<string> CategoryNames();
Retrieves a list of quantity names given a category name	A string collection	IList<string> QuantityNames(string category);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();

UDDInputText

Method purpose	Return Value	Data type of the returned value in C#
Retrieves string data	a string value of input data	string Data();

UDDInputTrace

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data	Double	IDictionary<double, double> DoubleData();
Retrieves x and y double data for a particular variation	Double	IDictionary<double, double> DoubleData (IDictionary<string, string> variation);
Retrieves x and y complex data	Double	IDictionary<double, double[]> ComplexData();
Retrieves x and y complex data for a particular variation	Double	IDictionary<double, double[]> ComplexData (IDictionary<string, string> variation);
Retrieves x and y text data	Text	IDictionary<string, string> TextData();
Retrieves x and y text data for a particular variation	Text	IDictionary<string, string> TextData (IDictionary<string, string> variation);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();
Retrieves an image of the trace data in a plot	An image	string Image();

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDDInputParams Class

The objects of this class must be created in Python script in the GetUDDInputParams and SetUDDInputParams functions.

Attributes :

- Input Name (string)
- Input Description (string)
- Input Type (string) (This can be boolean, number, text, trace, or solution.)
- BoolData (boolean)
- DoubleData (double)
- TextData (string)
- ReportType (string)
- SolutionName (string)
- DomainName (string)

Constructors:

- UDDInputParams(string name, string description, string type)
- UDDInputParams(string name, string description, string type, bool data)
- UDDInputParams(string name, string description, string type, double data)
- UDDInputParams(string name, string description, string type, string data)
- UDDInputParams(string name, string description, string type, string reportType, string solutionName, string domainName)

Property Accessors :

- Name : Get/Set the name of an input
- Description : Get/Set the description of an input
- Type : Get/Set the type of an input
- BoolData : Get/Set the data of a boolean input
- DoubleData : Get/Set the data of a number input
- TextData : Get/Set the data of a text input
- ReportType : Get/Set the report type
- SolutionName : Get/Set the name of the solution
- DomainName : Get/Set the name of the domain

UDD Input Interfaces

The Generate function takes in a list of inputs which allows the user to access data from the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IUDDInputBool : This interface exposes 3 methods.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the boolean data, set by the user in the setup dialog.

IUDDInputDouble : This interface exposes 3 methods.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the double data, set by the user in the setup dialog.

IUDDInputText : This interface exposes 3 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the text data, set by the user in the setup dialog.

IUDDInputTrace : This interface exposes 9 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData() : Method used to return x and y double data as a IDictionary<double, double>
- DoubleData(IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a variation.
- ComplexData() : Method used to return x data and y complex data as a IDictionary<double, double[]>
- ComplexData(IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a variation.
- TextData() : Method used to return x data and y data as a IDictionary<string, string>
- TextData(IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a variation.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

IUDDInputSolution : This interface exposes 11 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData(string name): Method used to return x and y double data as a IDictionary<double, double>, given a quantity name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- DoubleData(string name, IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a quantity name and a variation.
- ComplexData(string name) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name.
- ComplexData(string name, IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name and a variation.
- TextData(string name) : Method used to return x data and y data as a IDictionary<string, string> given a quantity name.
- TextData(string name, IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a quantity name and a variation.
- CategoryNames() : Method to return a list of category names in the solution as an IList<string>
- QuantityNames(string category) : Method to return a list of quantity names in the solution as an IList<string>, given a category.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

Examples

```
def Generate(self, input, docgen, progMon):
# Getting the boolean data set by the user
boolinput = input[0].Data()
# Getting the double data set by the user
dblinput = input[1].Data()
# Getting the text data set by the user
textinput = input[2].Data()
# Getting the category names in a solution
categories = input[3].CategoryNames()
# Getting the quantity names based on a category
quantities = input[3].QuantityNames(categories[0])
# Getting the XY data from the trace
xydata = input[4].DoubleData()
```

Explication of a Sample UDD Script

This VB script defines a document. It is a portion of one of the UDD example VB scripts.

```
Array("NAME:Test Report",          ' Name of the document
"Test Report",                     ' Description of the document
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"SysLib",                                   ' Location of the python script: Syslib, Userlib, PersonalLib, etc.
"TestUDDReport",                            ' Relative path of the script in the UserDefinedDocuments folder

' This array is the start of the input definition.
Array("NAME:Inputs",                        ' Document Inputs keyword

' This array contains the Solution input.
Array("NAME:DLMetrics",                     ' Input name
"Solution",                                 ' Solution Input Type
"Data Line Metrics",                        ' Input Description
-1,                                         ' Solution ID
-1),                                        ' Report ID

' This array contains the trace input.
Array("NAME:DQ0",                           ' Input name
"Trace",                                    ' Trace Input Type
"DQ0",                                      ' Input Description
-1,                                         ' Solution ID
-1),                                        ' Report ID

' This array contains the text input.
Array("NAME:Name",                          ' Input name
"Text",                                     ' Text Input Type
"User Name",                                ' Input Description
Array("Sita Ramesh")),                      ' Default Value

' This array contains the Bool input.
Array("NAME:Summary",                       ' Input name
"Bool",                                     ' Boolean Input Type
"Display Summary",                          ' Input Description
Array(true)),                               ' Default Value

' This array contains the number input.
Array("NAME:Version",                       ' Input name
"Number",                                   ' Number Input Type
"Script Version",                           ' Input Description
Array(1021)))),                             ' Default Value
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
' This array contains trace selection for the solution and trace inputs.
Array("NAME:DocTraces",                    ' Document traces keyword

' This array has input for "DLMetrics".
Array("NAME:DLMetrics",                    ' Input name

' This array defines a trace similar to the UDO. This trace definition is a User defined solution
Array("User Defined", "", "DDR3 AC-Timing 4-DQ1", Array("Context:=",
"", Array("Index:=", Array("All"), "Trise:=", Array("Nominal"),
"Tfall:=", Array("Nominal"), "Pulse_Width:=", Array("Nominal"),
"Data_Rate:=", Array("Nominal"), "Length:=", Array("Nominal")), Array
("Probe Component:=", Array("")), Array())),

' This array is for input "DQ0".
Array("NAME:DQ0",

' This array defines a trace similar to the UDO. This trace definition is a Standard solution.
Array("Standard", "DQ0", "NexximTransient", Array("NAME:Context",
"SimValueContext:=", Array(1, 0, 2, 0, false, false, -1, 1, 0, 1, 1,
"", 0, 0, "DE", false, "0", "DP", _
false, "20000000", "DT", false, "0.001", "WE", false, "100ns", "WM",
false, _
"100ns", "WN", false, "0ps", "WS", false, "0ps")), Array("Time:=",
Array("All"), "Trise:=", Array( _
"Nominal"), "Tfall:=", Array("Nominal"), "Pulse_Width:=", Array
("Nominal"), "Data_Rate:=", Array("Nominal"), "Length:=", Array
("Nominal")), Array("Probe Component:=", Array( _
"DQ0")), Array())))
```

Document Generator Interfaces

This document briefly describes the API interfaces available in the document generator plugin.

(Ansys.Ansoft.DocGeneratorPluginDotNet.dll)

Scripting objects available in the script for the Generate function:

- oApp = self.GetUDDAppContext()

  Gets the application context. Use this interface to get the active project ad the version of the product.

  ```
  oDesktop = oApp.GetAppDesktop()
  ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if oDesktop != None:
vr = oDesktop.GetVersion()
oProject = oDesktop.GetActiveProject()
```

- oDesign = self.GetUDDDesignContext()

  Gets the design context. Use this interface to get the design name.

```
oDesign = self.GetUDDDesignContext()
  if oDesign != None:
    nm = oDesign.GetName()
```

- IUDDGenerator interface

  This interface available in the Generate method of the UDDPluginExtension. This interface can be used to:

  1. Set the document output file path.
     ```
     docgen.SetOutput("C:\\Examples\\DocumentOutput.xml")
     ```
  2. Get the document root.
     ```
     docroot = docgen.GetDocumentRoot()
     ```
  3. Write out to the output file.
     ```
     docgen.Write()
     ```
  4. Write HTML document
     ```
     void WriteHTML();
     ```
  5. Write PDF document
     ```
     void WritePDF();
     ```
  6. Load the HTML transform object
     ```
     void LoadHTMLTransform();
     ```
  7. Load the cached PDF transform object
     ```
     void LoadPDFTransform();
     ```
  8. Get script path
     ```
     string GetScriptPath();
     ```
  9. Get output file path
     ```
     string GetOutputFilePath();
     ```

- IUDDRoot interface

  Calling GetDocumentRoot() on the IUDDGenerator interface provides you with the this interface. This interface can be used to:

  1. Add a new section to the document. Provide a section title.
     ```
     section1 = docroot.AddSection("Section title")
     ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Add a new section to the document. Provide a section title and subtitle.
   ```
   section1 = docroot.AddSection("Section title", "Section subtitle")
   ```
3. Add a new title
   ```
   section1 = docroot.AddTitle("Title")
   ```
4. Add a new subtitle
   ```
   section1 = docroot.AddSubtitle("Subtitle")
   ```

- IUDDSection interface

  Calling AddSection() on the IUDDRoot interface provides you with this interface. This interface can be used to:

  1. Set an ID for the section for internal links.
     ```
     section1.SetID("id")
     ```
  2. Add a new table to the document. Provide a table title.
     ```
     table1 = section1.AddTable("Table title")
     ```
  3. Add a new image to the document. Provide an image title and a file path to the image file.
     ```
     image1 = section1.AddImage("Image title")
     ```
  4. Add text to the document.
     ```
     text1 = section1.AddText("Random text........")
     ```

- IUDDImage interface

  Calling AddImage() on the IUDDSection interface provides you with this interface. In this interface, you can call the following methods:

  1. Set an ID for the image for internal links.
     ```
     image1.SetID("id")
     ```
  2. Set alignment information. Can be "center", "left", and "right".
     ```
     image1.SetAligment("center")
     ```
  3. Set the file path of the image file. Not necessary if image file path is set through the AddImage() method.
     ```
     image1.SetFileRef("Image path")
     ```
  4. Set the format of the image file. Can be "BMP", "PNG", "JPEG", "JPG", "DVI", etc.
     ```
     image1.SetFormat("format")
     ```

- IUDDText interface

  Calling AddText() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set an ID for the text for internal links.
   ```
   text1.SetID("id")
   ```
2. Set the emphasis attribute on the text.
   ```
   text1.SetEmphasis()
   ```
3. Set the quotes attribute on the text.
   ```
   Text1.SetQuotes()
   ```
4. Set the block quotes attribute on the text.
   ```
   text1.SetBlockquotes()
   ```
5. Set quotes on the text.
   ```
   Text1.SetQuotes()
   ```
6. Set the wordsize attribute on the text
   ```
   text1.SetSize(size as an integer)
   ```
7. Set a link to an ID of any element to provide internal links
   ```
   text1.SetLink("linkname")
   ```
8. Set an event link to handle an event. The HandleUDDEvents method should be implemented in the script to handle the event.
   ```
   text1.SetEventLink("linkname")
   ```

- IUDDTable interface

  Calling AddTable() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the table for internal links.
     ```
     table1.SetID("id")
     ```
  2. Set alignment information. Can be "center", "left", and "right".
     ```
     table1.SetAlignment("center")
     ```
  3. Set the background color of the table
     ```
     table1.SetBgColor(string bgcolor)
     ```
  4. Set the frame type. Can be "all", "bottom", "top", "sides", and "topbot".
     ```
     table1.SetFrame(string frame)
     ```
  5. Add a table group and specify the number of columns. A table can have multiple table groups.
     ```
     IUDDTableGroup table1.SetTableGroup(int columns)
     ```

- IUDDTableGroup interface

  Calling AddTableGroup() on the IUDDTable interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the table group for internal links.
     ```
     tgroup1.SetID("id")
     ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set alignment information. Can be "center", "left", and "right".

   ```
   tgroup1.SetAlignment("center")
   ```

3. Set the column width of a column given the index of the column and the required width. Width can be set in 2 ways.
   - Width can be set relative to 1. E.g Setting it to "2*" makes the column width double the width of the others.
   - If the entire table width is considered to be 99.99 units. Width can be a number relative to this.

   ```
   tgroup1.SetColumnWidth(int index, string width)
   ```

4. Add a header to the table group

   ```
   IUDDTableRow tgroup1.AddHeader()
   ```

5. Add a header with multiple rows to the table group. Takes number of sub rows.

   ```
   IUDDTableRow tgroup1.AddHeader(int rows)
   ```

6. Add a row of content to the table group

   ```
   IUDDTableRow tgroup1.AddContent()
   ```

7. Add content with multiple rows to the table group. Takes number of sub rows.

   ```
   IUDDTableRow tgroup1.AddContent(int rows)
   ```

- IUDDTableRow interface

  Calling AddHeader() & AddContent() on the IUDDTableGroup interface provides you with the this interface. In this interface, you can call the following methods:

1. Set an ID for the table row for internal links.

   ```
   trow1.SetID("id")
   ```

2. Set alignment information. Can be "center", "left", and "right".

   ```
   trow1.SetAlignment("center")
   ```

3. Set cell text. Can be cell content or header text. Takes a column index and a text string. It is added to the first row.

   ```
   IUDDTextElement trow1.Add(int column, string text)
   ```

4. Set cell text. Can be cell content or header text. Takes a column index, row index and a text string. Takes in a row number because a table row can have multiple sub rows.

   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```

5. Set cell content. Takes a column index and an int value. It is added to the first row.

   ```
   IUDDTextElement trow1.Add(int column, int value)
   ```

6. Set cell content. Takes a column index, row index and a int value.

   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Set cell text. Takes a column index and a double value. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, double value)
   ```
8. Set cell text. Takes a column index, row index and a double value.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, double value)
   ```
9. Set cell text spanning 2 columns. Can be cell content or header text. Takes a sub row index , starting column index., ending column index and a text string.
   ```
   IUDDTextElement trow1.AddSpanningcolumnst(int subrow, int columnstart, int columnend, string text)
   ```
10. Set cell text. Can be cell content or header text. Takes a column index, starting sub row index, ending sub row index and a text string.
    ```
    IUDDTextElement trow1.AddpanningRows(int column, int subrowstart, int subrowend, string text)
    ```

- IUDDTableRow interface

  Calling Add() on the IUDDTableGroup interface provides you with the this interface. In this interface, you can call the following methods :

  1. Set an ID for the table row for internal links.
     ```
     trow1.SetID("id")
     ```
  2. Set alignment information . can be "center", "left", and "right".
     ```
     trow1.SetAlignment("center")
     ```

Includes all the methods exposed by the IUDDText interface.

Chip Model Analyzer (CMA)

Ansys Chip Model Analyzer (CMA) is an advanced Chip Power Model (CPM) creation tool for performance checking and failure diagnosis of the Power Delivery Network (PDN). CMA can use either HSpice or Nexxim simulators. HSpice and external Nexxim require licenses, but the Nexxim Internal simulator does not.

CMA does not currently interact with Electronics Desktop. However, you can launch CMA from within Electronics Desktop by clicking **Tools** > **Chip Model Analyzer (CMA)**.

From there, you can import CPM files for analysis or create a pseudo CPM.

For more information about using CMA, consult the CMA training documentation by launching CMA and selecting **Help** > **Content**.

PinToPinUtility

The PinToPin utility can quickly extract HFSS or SIwave models for specified nets of package and PCB geometries. It establishes a repeatable process for robust geometry extraction, port

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

configuration, and passive model assignment. Process execution can be done via the user interface (Windows only) or in non-graphical mode (Windows or Linux).

For more information, consult the SIwave and HFSS 3D Layout help.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19 - Meshing

Once you have finished designing your model, you need to generate the computational mesh that is used as the basis of solutions.

A good computational mesh is essential for a successful and accurate solution. If the overall mesh is too coarse, the resulting solution may be inaccurate. If the overall mesh is too fine, the computational cost may become prohibitive. In summary, the cost and accuracy of the solution are directly dependent on the quality of the mesh.

Windows® HPC Non-Exclusive Jobs

Windows HPC allows three choices for the Cluster AffinityType setting:

- No Jobs: Affinity is not set for any jobs
- Non Exclusive Jobs (default): Affinity is not set for exclusive jobs
- All Jobs: Affinity is set for all jobs

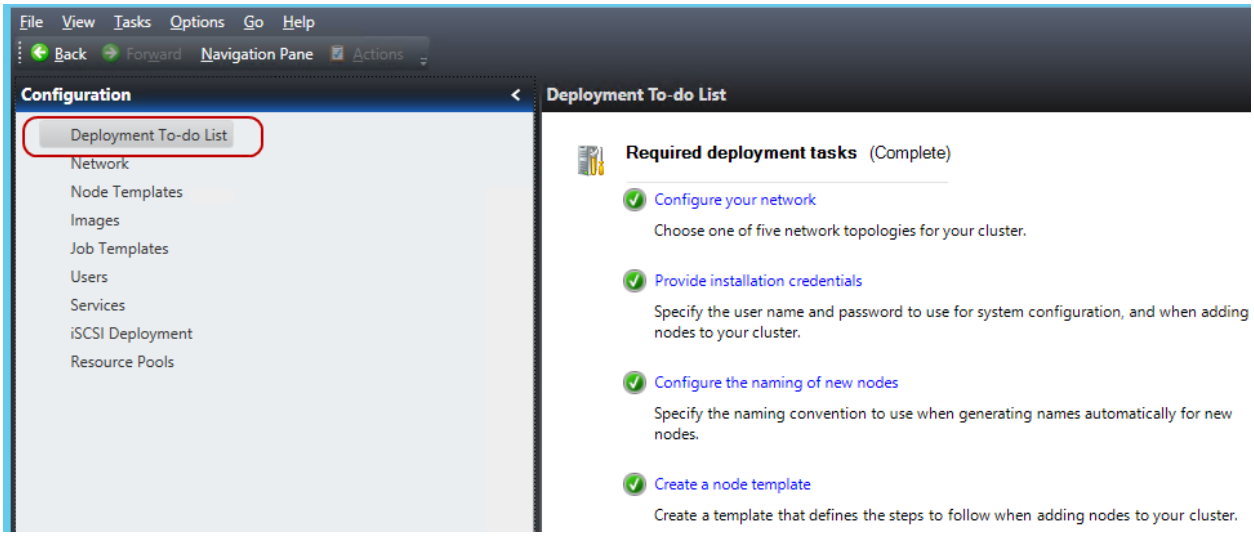
If the AffinityType is set to “No Jobs,” then we can allow non-exclusive jobs in many cases, but non-exclusive jobs will be disallowed if you specify an explicit host list (because all resources on all specified hosts are used), if you specify nodes and cores (to ensure that the job has exclusive access to the specified number of nodes), or if you specify multi-threading (to prevent a single task from being split across hosts). If the AffinityType is set to “Non Exclusive Jobs,” then exclusive jobs will be disallowed for all cases, the same as the restrictions for the previous release.

The previous job submission dialog always has the “exclusive” check box disabled for Windows HPC jobs, so that “exclusive” jobs cannot be submitted to a Windows HPC cluster. The “exclusive” check box is now enabled for Windows HPC jobs. This allows you to submit exclusive jobs to a Windows HPC cluster. If this check box is checked, but the submission parameters and Windows HPC cluster AffinityType do not allow submission of an exclusive job, then an error message is displayed when you submit or preview the job.

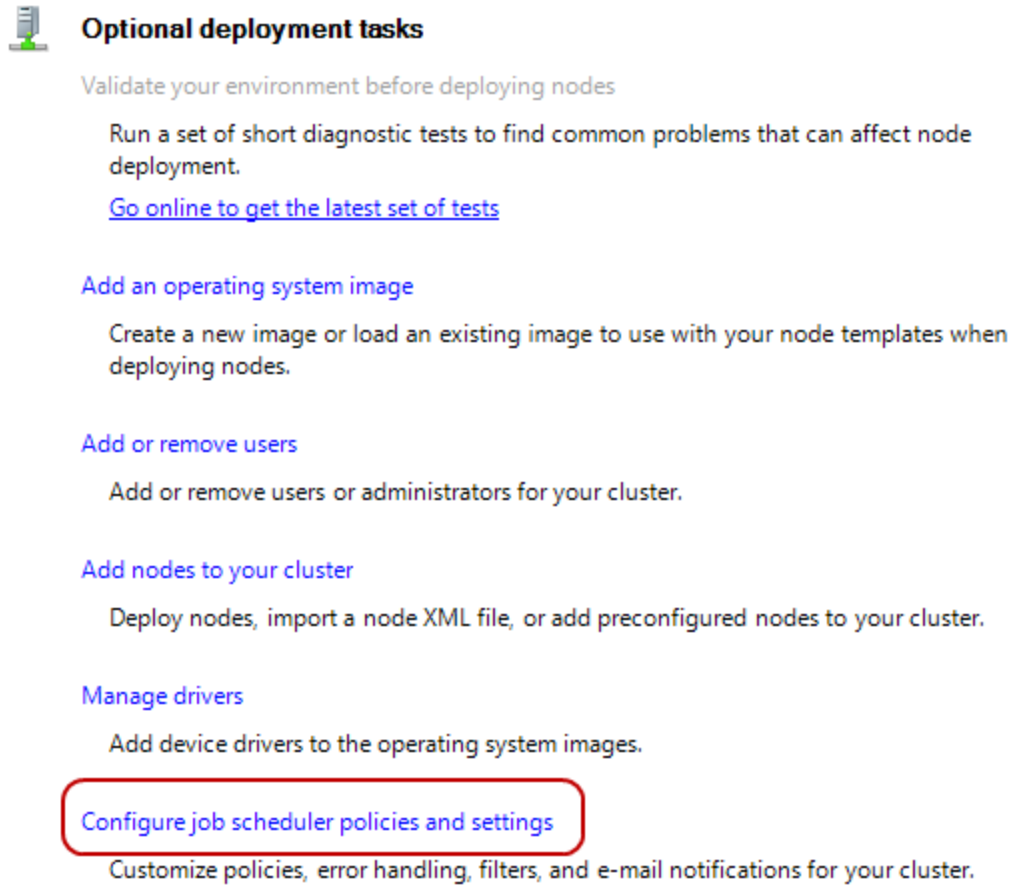
Setting the Windows HPC Cluster Affinity Type

A Windows HPC Cluster administrator is able to modify the Affinity Type setting for the cluster. The administrator should start the Windows HPC Cluster Manager. The first step is to select the Configuration button on the lower left pane, then select the Deployment To-do List in the upper left pane. Then, the upper right pane displays the Deployment To-do List, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

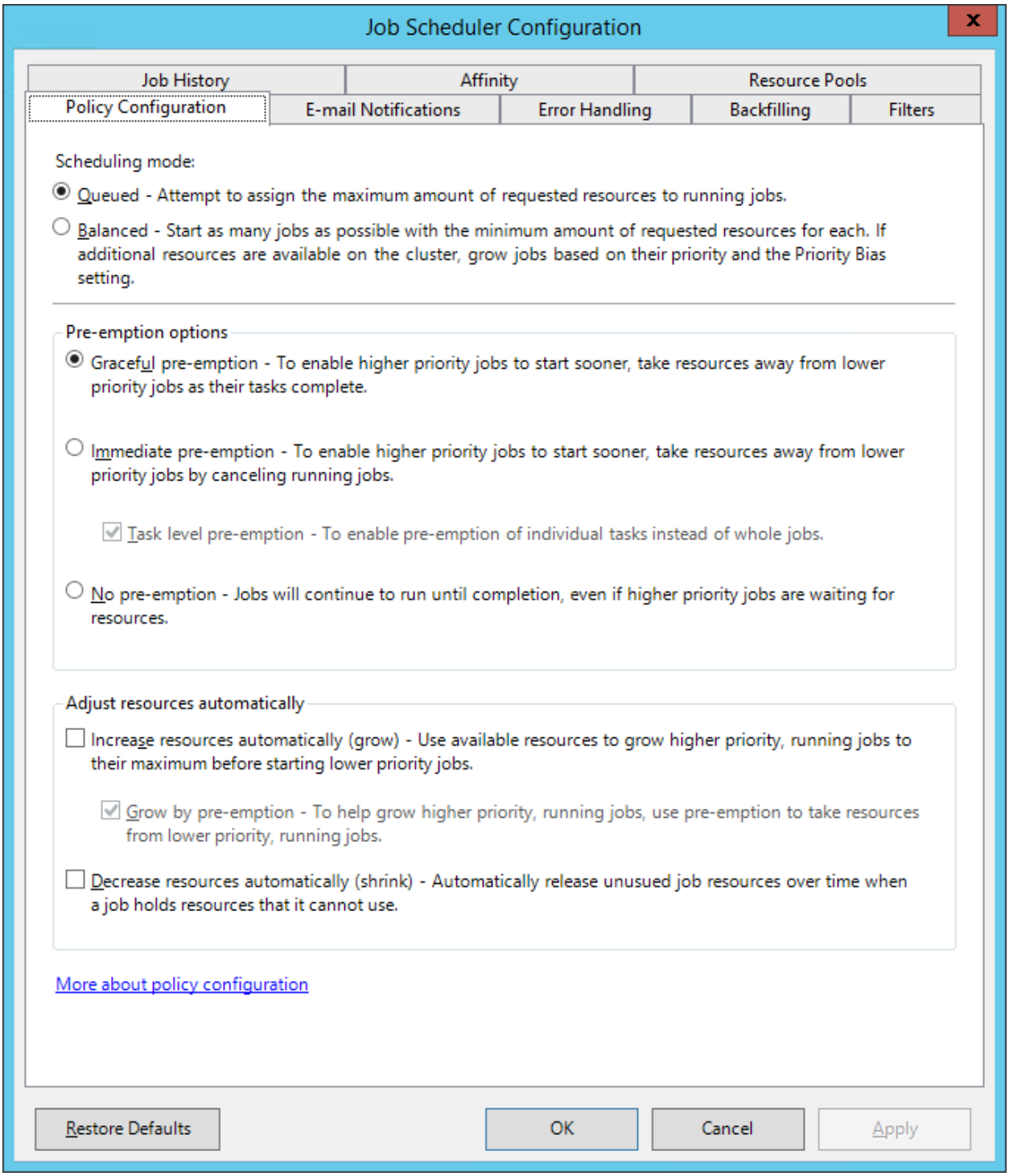


Next, the "Configure job scheduler policies and settings" should be selected. This is under the heading "Optional deployment tasks."



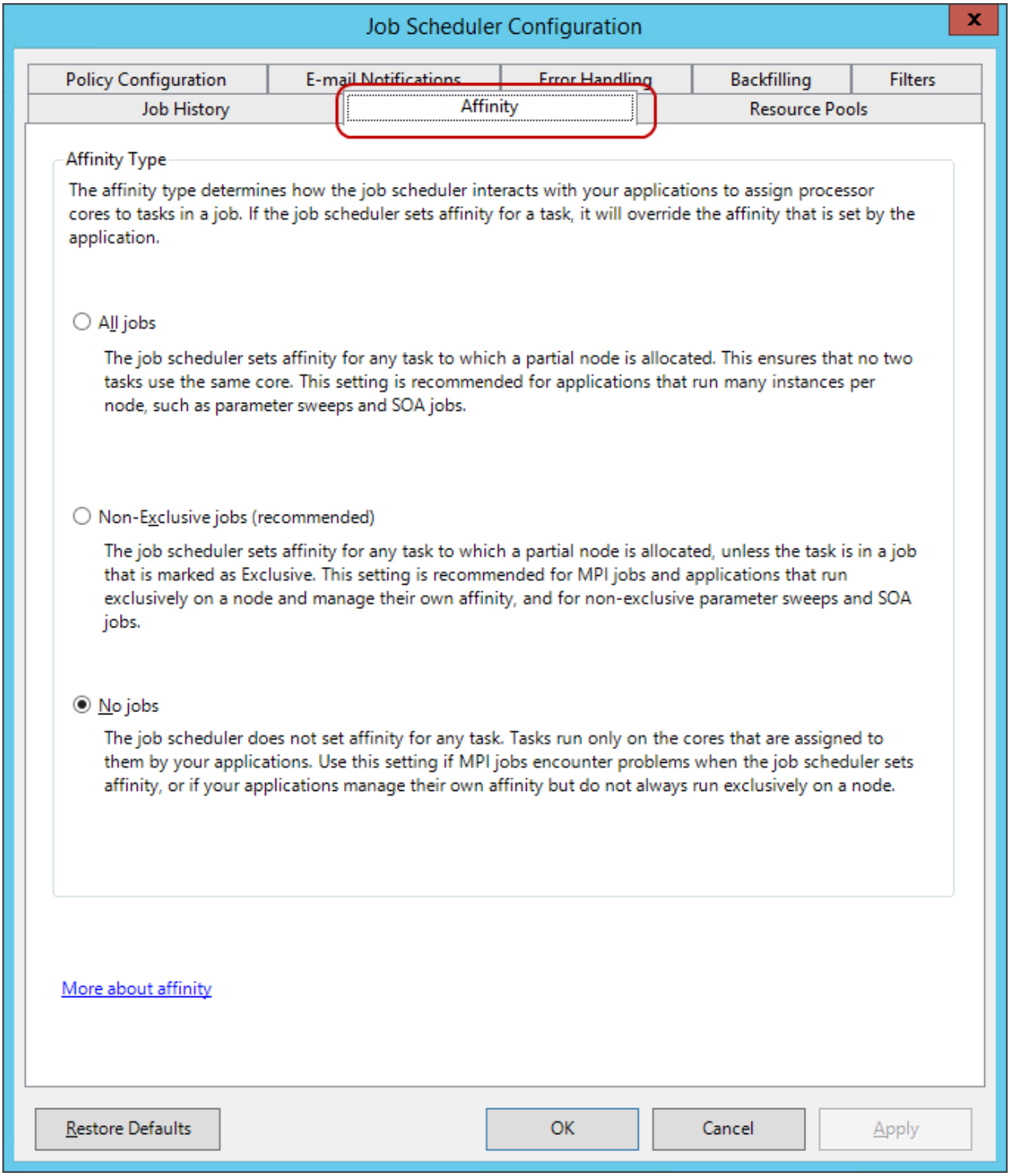
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Job Scheduler Configuration* dialog box pops up, as shown below.



Next, select the "Affinity" tab should be selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The cluster administrator may now select a radio button to select the desired Affinity Type:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs
- Non-Exclusive jobs
- No jobs

In order to submit non-exclusive jobs, the "No jobs" option should be selected. The "Non-Exclusive jobs" option may be selected, but this will not allow non-exclusive Ansys Electromagnetics jobs to be submitted to the cluster. The "All jobs" option is not supported for Ansys Electromagnetics jobs. The OK or Apply buttons may be used to apply the selected option to the cluster.

Examples with "Submissions with Nodes are for exclusive usage by this job" Unchecked and Checked

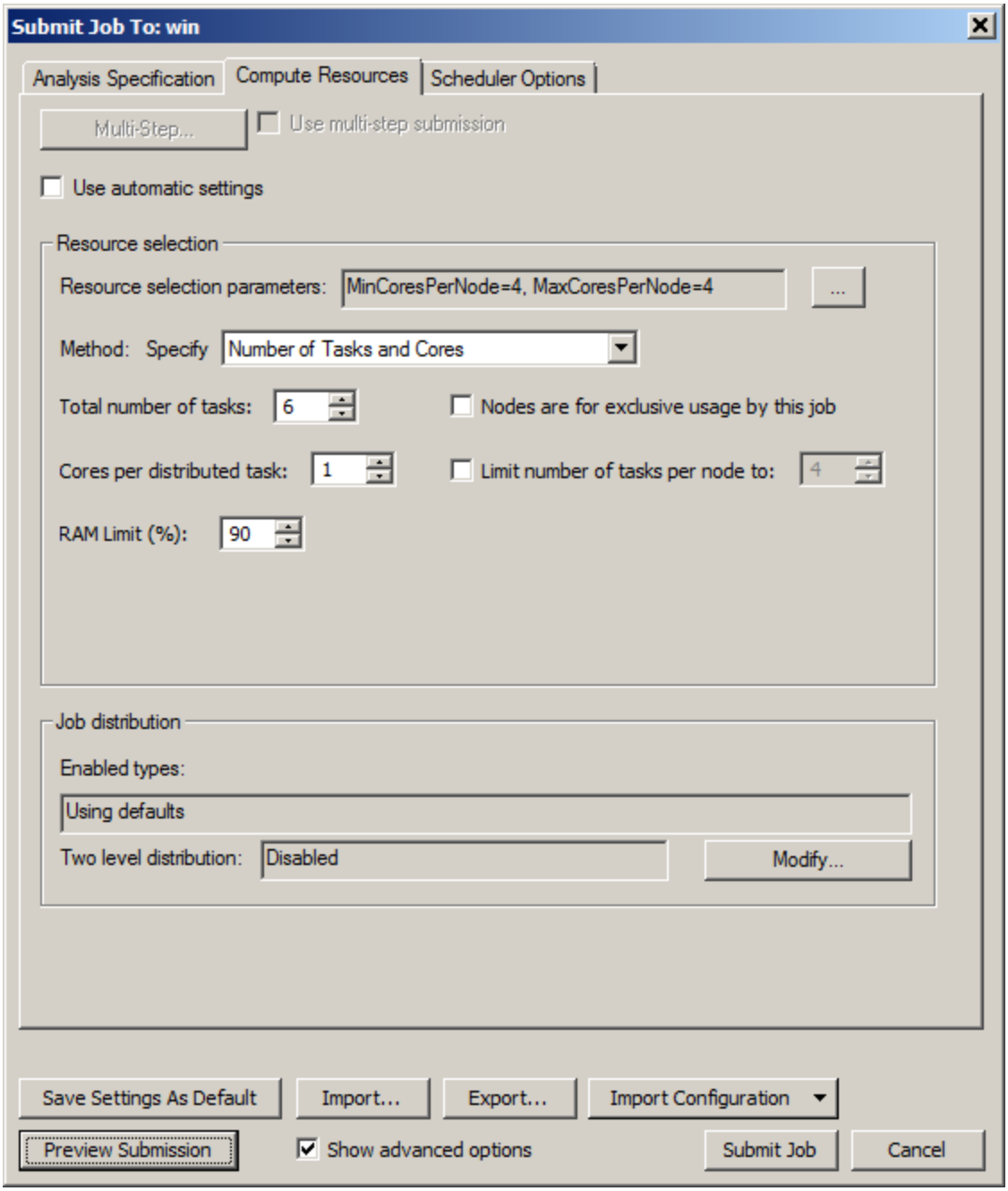
Most of the following screen shots are for the same project file submitted to a Windows HPC cluster with the AffinityType set to "No Jobs". The scheduler resource selection parameters are the same for all of these screen shots:

- MinCoresPerNode=4
- MaxCoresPerNode=4

The first set of screen shots show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is not checked, and the resource selection method is "Specify Number of Tasks and Cores". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. If it is unchecked, then the job is submitted with the exclusive parameter for the job set to false.

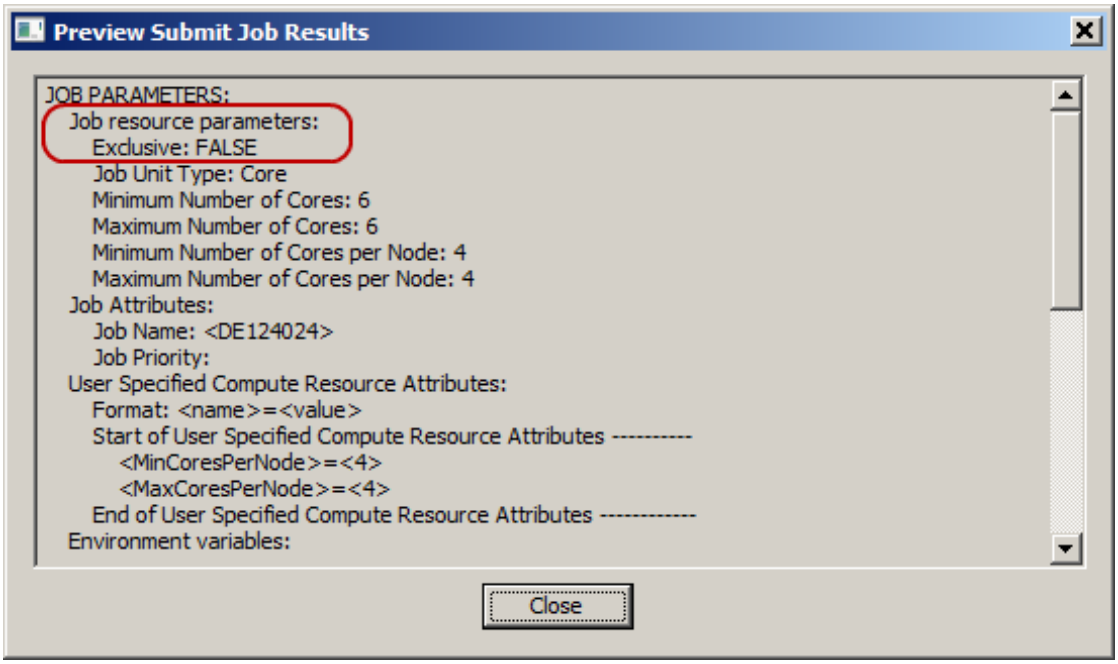
The following screen shot shows this configuration with a Submit Job with Task and Cores specified and nodes as non-exclusive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows Preview for the job submission indicating that the exclusive parameter for the job is false.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

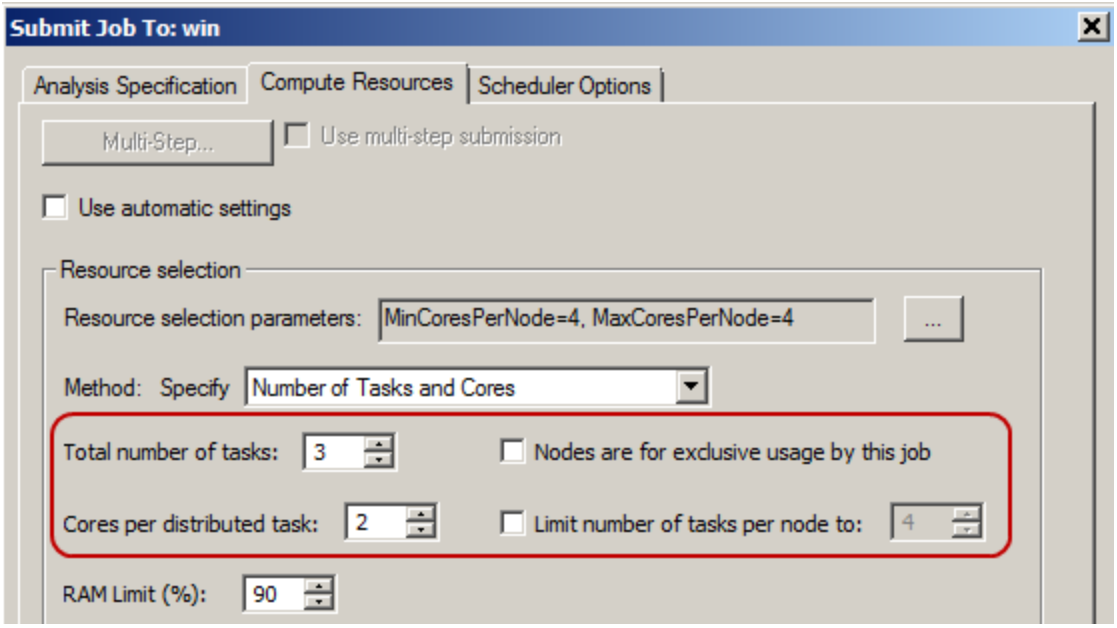


Jobs may only be submitted using this resource selection method with the exclusive parameter set to false if all three of the following conditions are met:

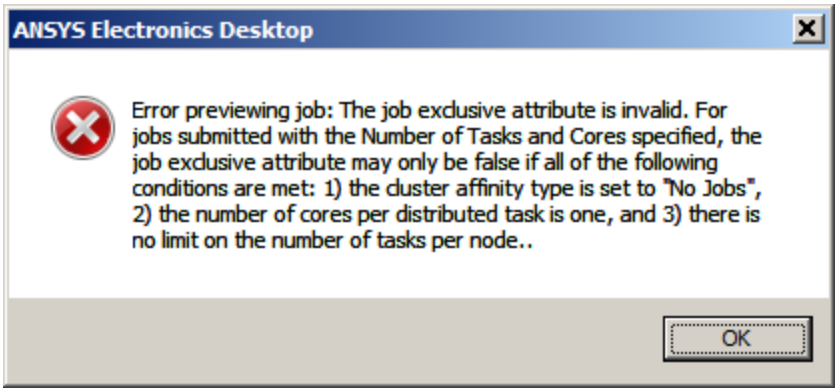
1. The cluster AffinityType is "No Jobs",
2. There is no multiprocessing, and
3. There is no limit on the number of tasks per node.

If any of these requirements are not met, then the job is not submitted or previewed, and an error message appears. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



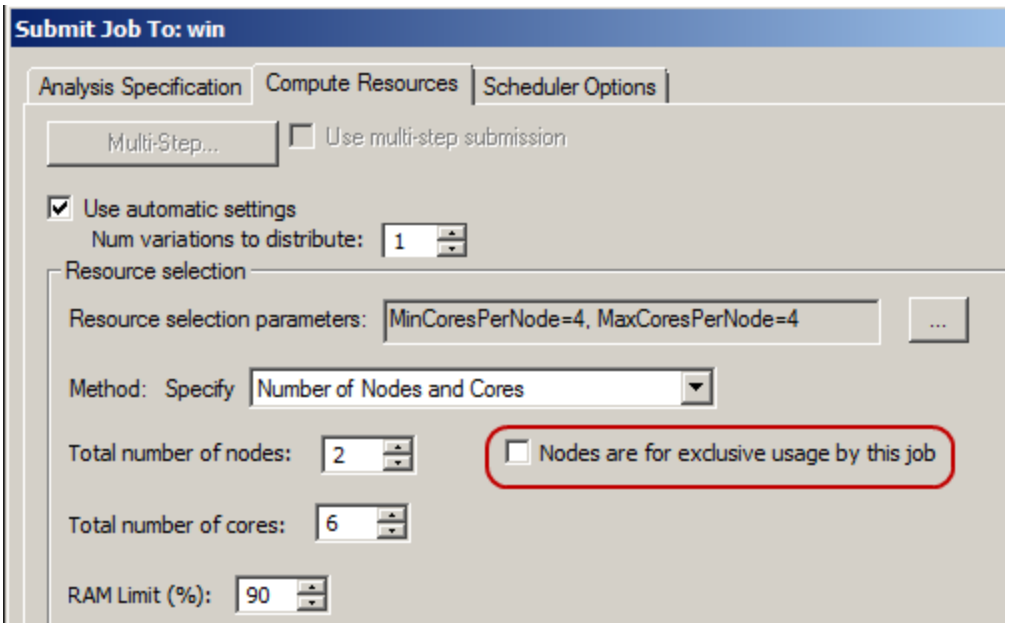
The following figure and screen shot ErrorMsgTasksAndCoresNonExMP.png shows the error message box shown to the user.



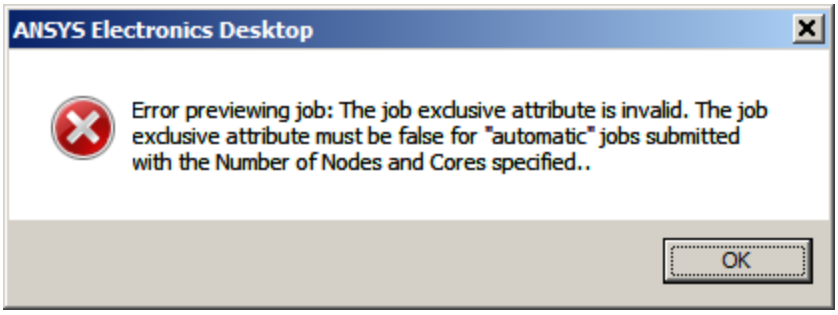
The next example shows submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Nodes and Cores". If you select *Use automatic settings* with *Num variations to distribute* set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set Num variations to distribute to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically. The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

behavior is the same as previous releases. If it is unchecked, then the job is not submitted or previewed because only exclusive jobs are allowed when using this resource selection method, and an error message box is shown to the user in this case. The following figure shows this configuration:

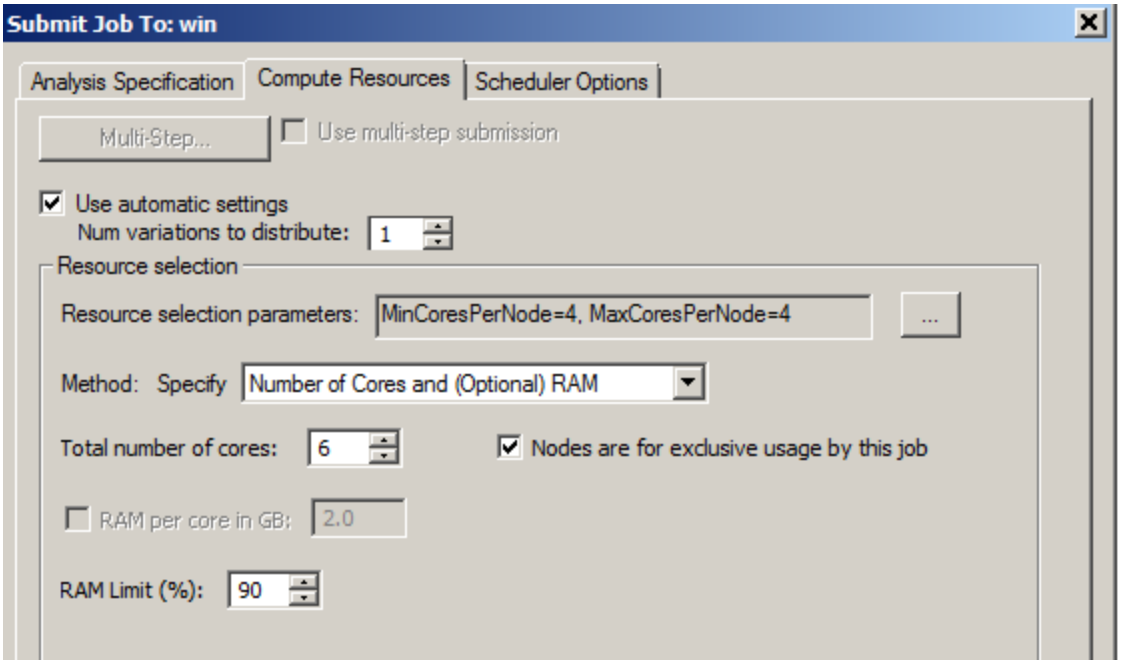


The following figure shows the error message.

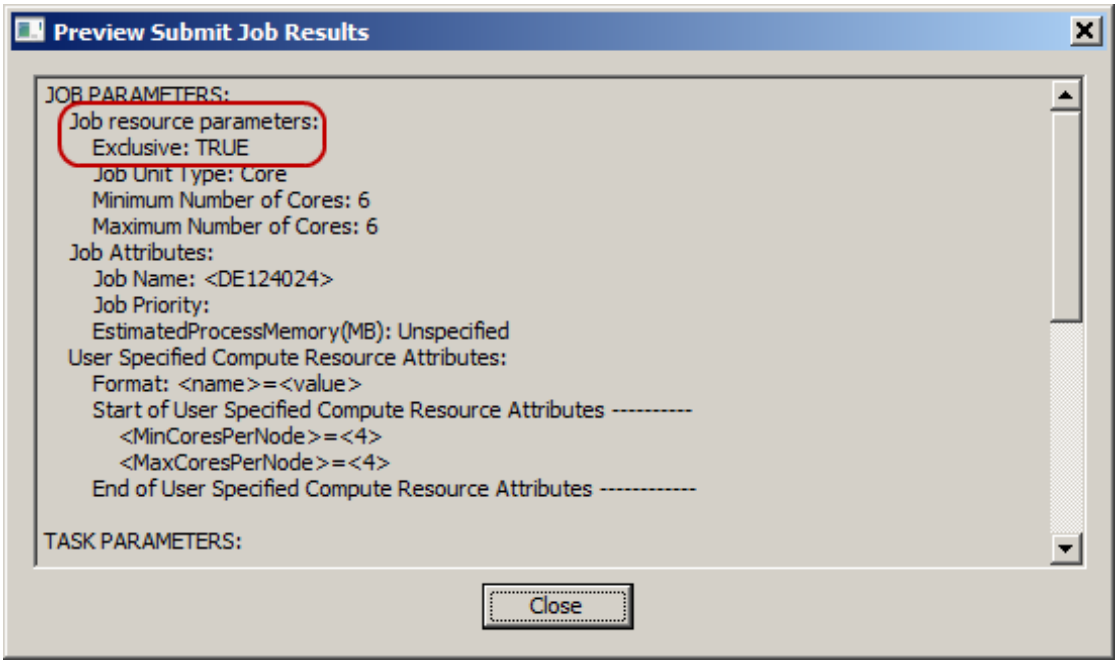


The next example show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Cores and (Optional) RAM". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and you may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

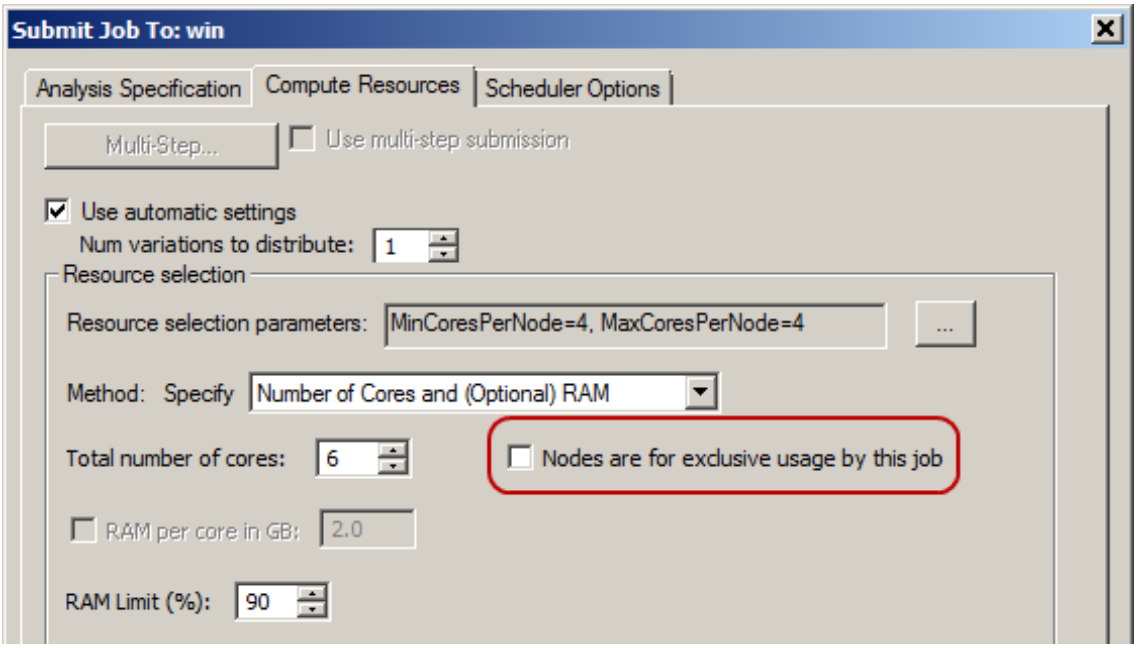


The following figure shows the Job Preview for this case, indicating that the exclusive parameter for the job is true.

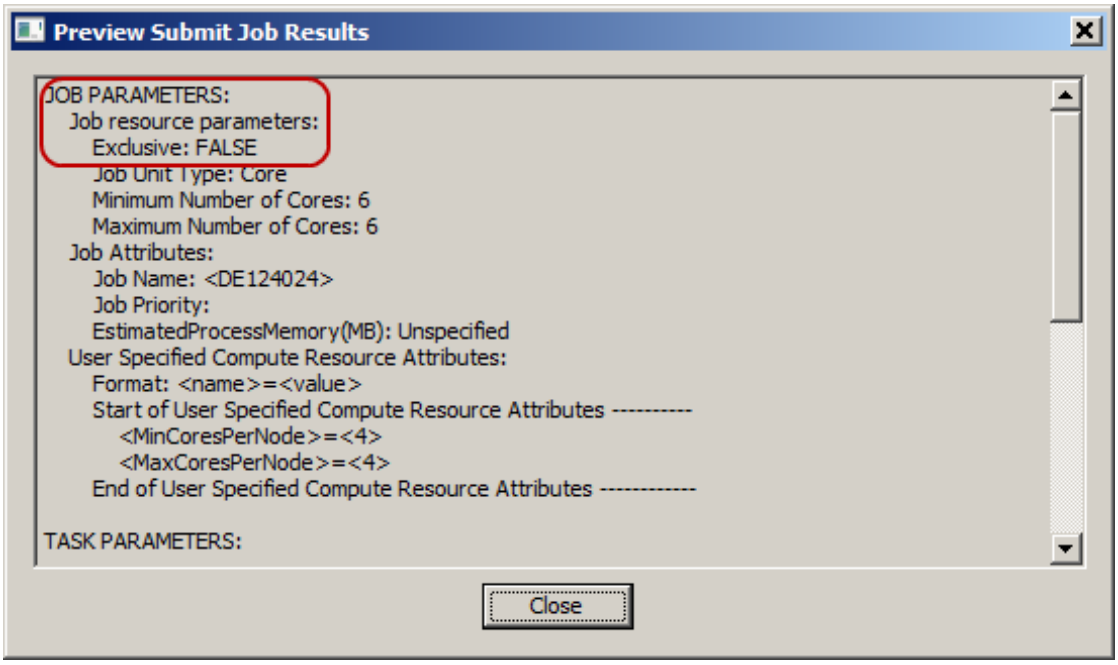


If the exclusive check box is unchecked, then the job is submitted with the exclusive parameter set to false. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



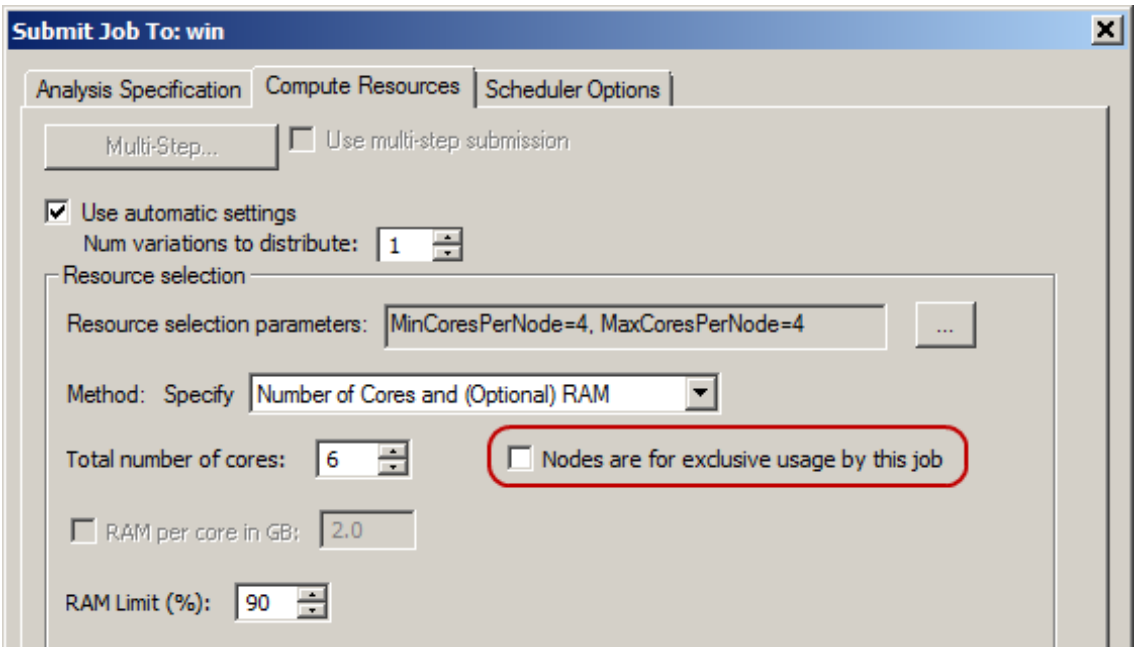
The following figurescreen shot PreviewCoresAndRAMNonEx.png shows the Job Preview for this case, indicating that the exclusive parameter for the job is false.



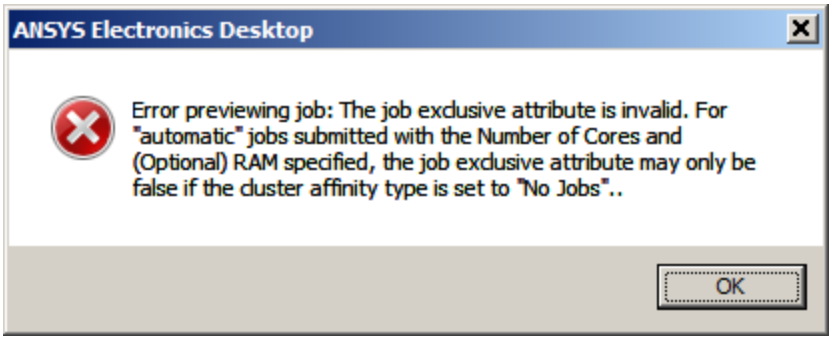
The last example resembles the previous one, except that the Windows HPC cluster AffinityType is set to "Non Exclusive Jobs" in this case. The job is submitted with the "Use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatic settings" check box in the checked state, and the resource selection method set to "Specify Number of Cores and (Optional) RAM". Because the AffinityType is not "No Jobs", only exclusive jobs may be submitted to this cluster. If the exclusive check box is not checked, then the job is not submitted or previewed, and an error message box is displayed to the user. The following figure shows this configuration.



The following figure shows the error message.



Mesh Linking

Importing a mesh allows a Mechanical simulation (target design) to be solved using the meshes created from a previously completed simulation (source design). The Mechanical simulation time can be reduced by eliminating the time spent on creating a mesh. Additionally, matched

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

meshes improve the accuracy of mapping for imported EM Loss excitations in Thermal solutions.

You can import a mesh from another design in the same project file or from a different project.

Mesh linking will work if the following two conditions are satisfied:

- Both designs have matching object names.
- Both designs have identical geometry (based on each object having the same model history).

See Mesh Linking Requirements below for more information.

You can import meshes from the following types of source designs into a Mechanical target design:

- **Mechanical**
- **HFSS**
- **Maxwell 3D**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- You cannot import a mesh from a Mechanical–Thermal source design (either Steady-State or Transient) if it includes any Thermal Contact assignments, unless the target design is also one of the two Mechanical–Thermal solution types. The specific source and target solution types do not have to match. That is, one can be steady-state and the other transient. An additional requirement is that the target Thermal design must have Contact assignments at the same faces as the source design, though the thermal resistance values can differ between the source and target designs.
- You cannot import a mesh from an HFSS source design that includes an IE or PO Region.
- Source designs can include sheet objects, and the sheet objects must remain active model objects in the Mechanical target design for mesh linking to work.

  For all Mechanical designs (Modal, Thermal, and Structural solutions), you cannot assign boundaries or excitations to sheet objects. These objects are ignored by the solver.

  For Thermal solutions only, you can select the **Shell Element** attribute in a sheet object's properties, and then you can assign it boundaries and excitations. Shell object participate in thermal solutions.
- HFSS **FE-BI** and **Dielectric Cavity** types of **Hybrid Regions** and other **vacuum** or **air** regions in source designs are supported for mesh linking.
  - **Thermal Solutions:** For such regions in **Thermal solutions**, clear the **Solve Inside** attribute in the region's properties.

    The ambient air or fluid around solid objects is generally represented by Convection boundaries. Specified or imported convection film coefficients take into account the effect of fluid motion along the convection surfaces. Clearing the **Solve Inside** attribute prevents redundant heat conduction through the region, which would occur in parallel with the convection boundary's effects.

    Mesh linking will work when you choose not to solve inside an object. Conversely, deleting the region or setting it as a non-model object (by clearing the **Model** attribute), would prevent mesh linking from working, because the active source and target geometry would no longer be identical.
  - **Modal and Structural Solutions:**, Avoid mesh linking from source design that contain such air or vacuum regions. The reasons are as follows:
    - If you were to include an air or vacuum region, you would have to define very small, arbitrary, non-zero Mass Density and Young's

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modulus values in the material properties, along with an arbitrary Poisson's Ratio value. For Modal solutions, the region would move with the solid objects as they vibrate, and the additional mass and stiffness, though small, could somewhat skew the Modal results. You might also encounter vibration modes in the air or vacuum regions themselves.

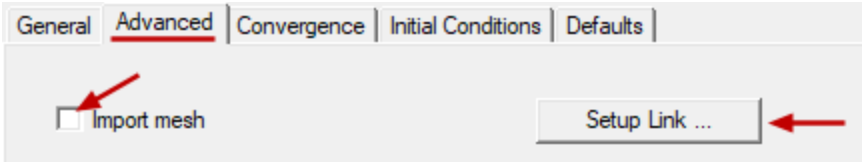
- Mesh linking will fail if you set included vacuum or air regions to non-model objects (that is, if you clear the **Model** attribute in the object properties). It will fail because the active geometries are no longer identical between the two designs.
- Clearing the **Solve Inside** attribute would prevent mesh linking from failing. But, if you don't solve inside the air or vacuum region, no displacement can occur. That is, the region will behave as a rigid body. Therefore, nothing in contact with the region would be able to deform. Mode shapes and frequencies could not be determined, thermal expansion or contraction would be prevented, and thermal stress results would be incorrect.

- The source and/or target designs can contain 3D Components if they're not encrypted. When an encrypted 3D Component is involved, mesh linking is only supported when the source and target design type and solution type are the same (for example, Mechanical – Thermal source and Mechanical – Thermal target).

How to Import a Mesh:

Prior to importing a mesh, ensure that you have created an exact duplicate of the source model geometry and object names by following the steps outlined under Mesh Linking Requirements.

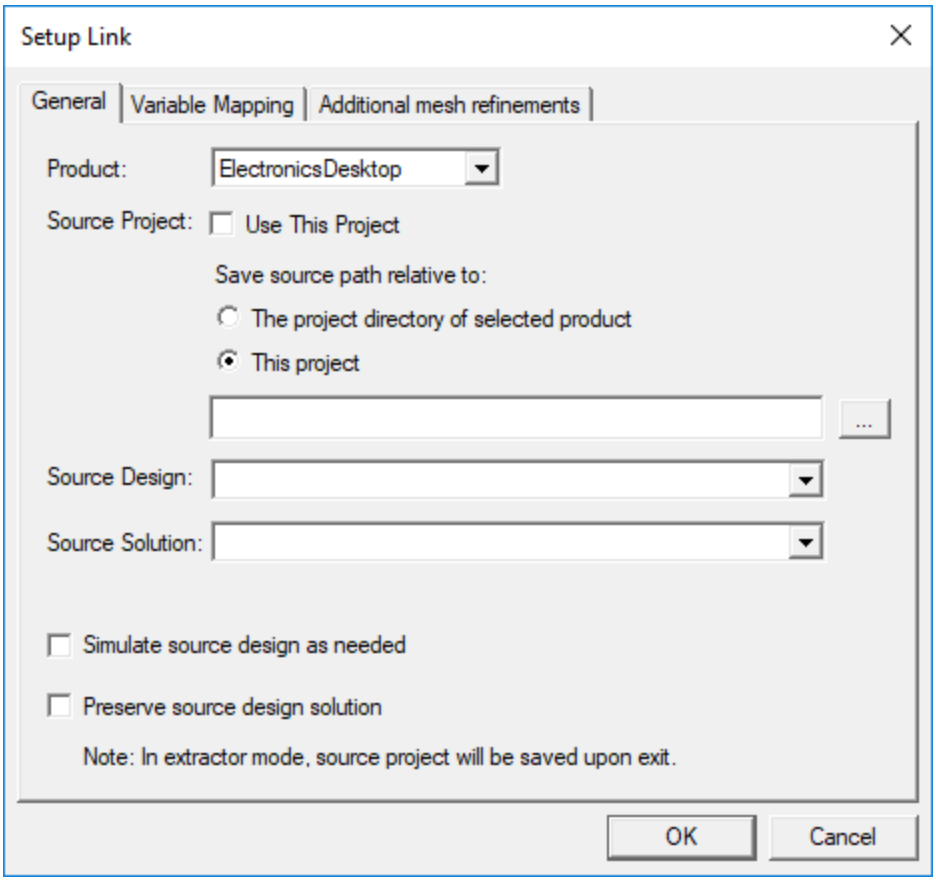
Mesh linking is set up as part of specifying the *Solution Setup*. The settings for importing a mesh are found under the **Advanced** tab of the *Modal Solve Setup, Thermal Solve Setup,* or *Structural Solve Setup* dialog box:



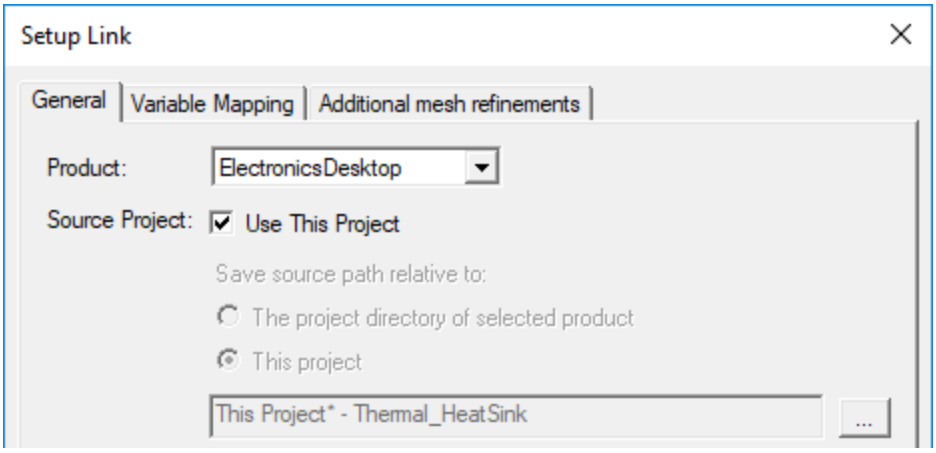
1. Select the **Import mesh** option.

   The *Setup Link* dialog box appears if a link hasn't been specified previously:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If the mesh is to be imported from a design in the same project:
   a. Select **Use This Project**:



   b. Skip to **step 5**.

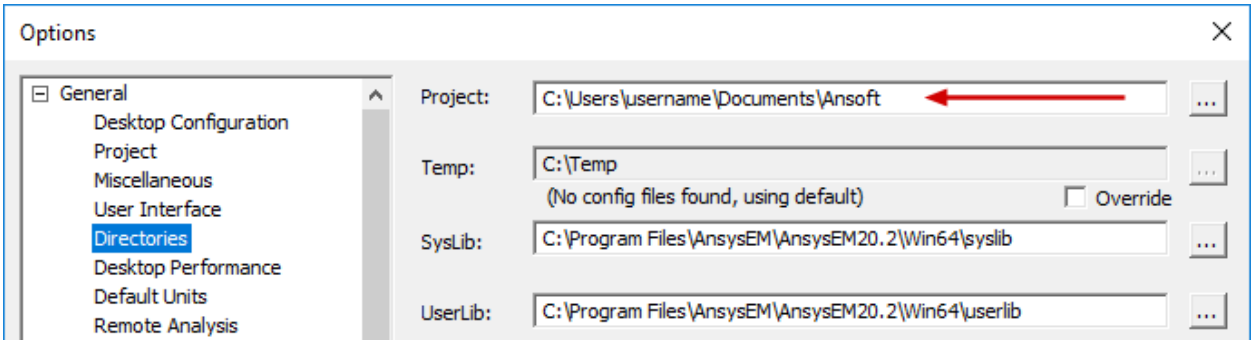
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> It is convenient to include source and target designs within the same project. Doing so keeps all project files in the same location and is the most convenient and straightforward way to organize linked designs. However, this approach may be impractical when there is a large number of designs in the project.

3. Otherwise, if the mesh source is to be from a different project, clear **Use This Project**. The Mechanical design saves the relative path to the source project in one of two ways.

   Choose one of the following two options under *Save source path relative to*:

   - **The project directory of selected product:** Save source project path relative to the default *Project* folder specified in the *Options* dialog box (*Tools* > *Options* > *General Options*):

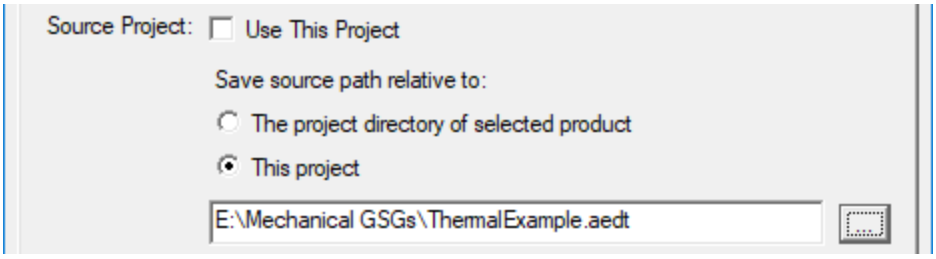
   - **This project:** Save source project path relative to the target project location.

4. Click the ellipsis button (**...**).

   The *Open* dialog box appears. Then:

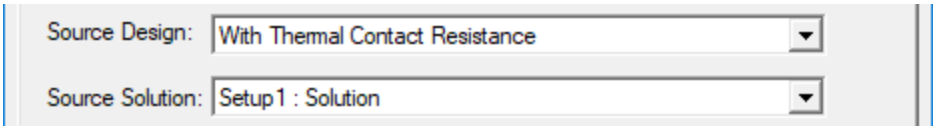
   a. Locate the source project file and select it.
   b. Click **Open**.

   The selected project is listed in the *Setup Link* dialog box:

5. Choose the **Source Design** and **Source Solution** from the provided drop-down menus:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> These two settings are automatically populated with an available design and solution in the source project. However, if more than one other design or solution exists in the source project, you can manually select the desired ones (if different from the automatically selected design or solution).

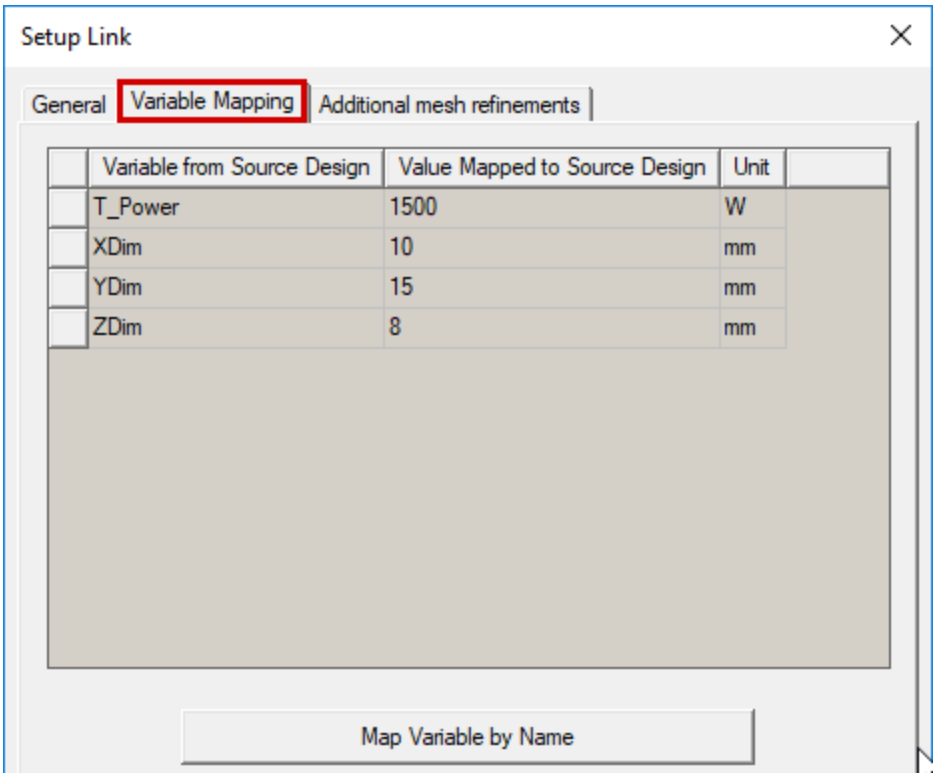
6. Optionally, select one or both of the following two options:
   - **Simulate source design as needed:** Select this option if you want an updated source mesh to be linked to the *target* design when the *source* design has been modified.
   - **Preserve design solution:** When the *source* design is in a separate project, and that project is closed, the source design will not be saved (preserving the preexisting solution data). This option is not applicable if the source design is in the same project as the *target* design.

     > **Note:**
     >
     > In the *Extractor mode*, the source project will be saved upon exit. *Extractor mode* means that the source project is opened during the link solely for the purpose of solving.

7. Click the **Variable Mapping** tab.

   If design variables have been defined in the source design, they are listed in the *Variable from Source Design* column. Initially, the nominal value of each variable is listed in the *Value Mapped to Source Design* column:

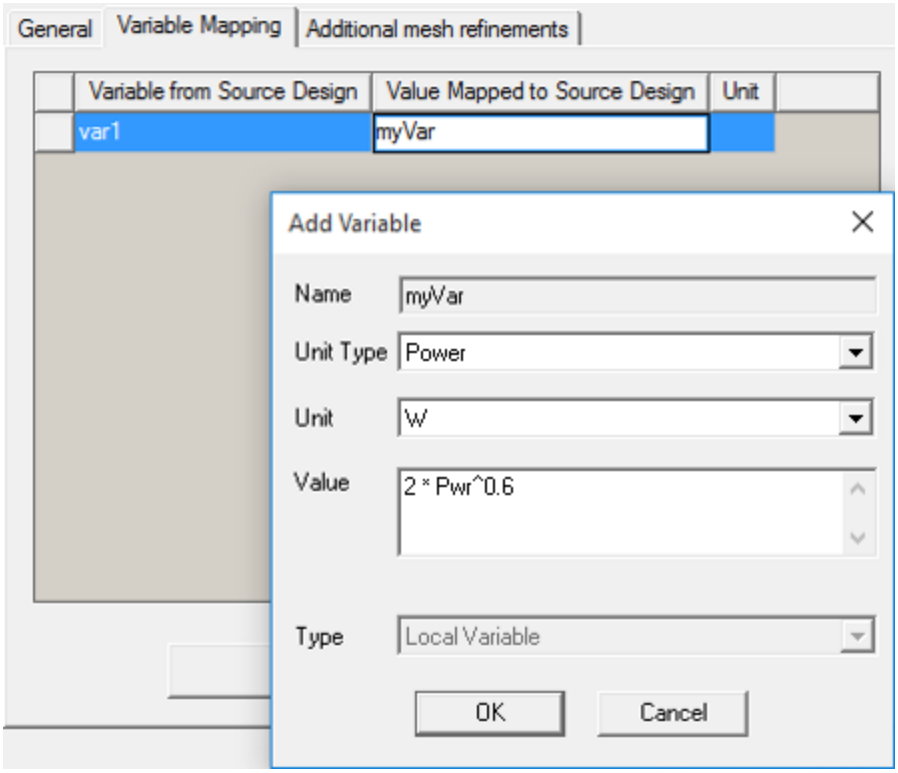
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



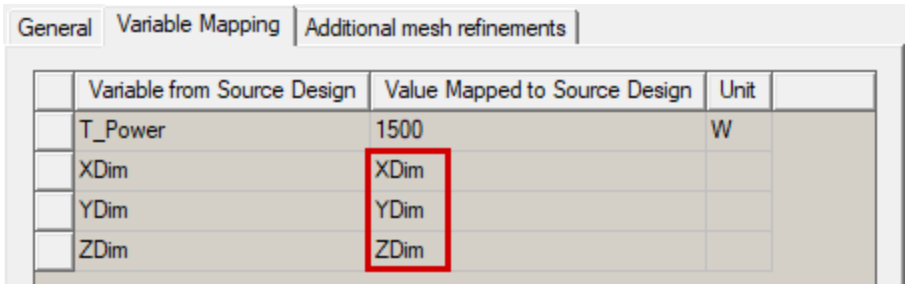
The values passed back to the source design from the mechanical design (second column) can either be constants (numerical values), variables, or expressions. The specified values determine the variation from which the source design provides the linked data.

> **Note:**
>
> Automatic variable creation is supported in the *Variable Mapping* tab. You can directly type a new variable name into the *Value Mapped to Source Design* column, and the *Add Variable* dialog box will appear, in which you can define the **Units** and **Value** of the variable. The specified value can be a constant, a variable name, or an expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Optionally, to keep the source and target design variables synchronized, click **Map Variable by Name**. Then, the same variable names will appear in both columns, and the values will remain synchronized between the two mesh-linked analyses.



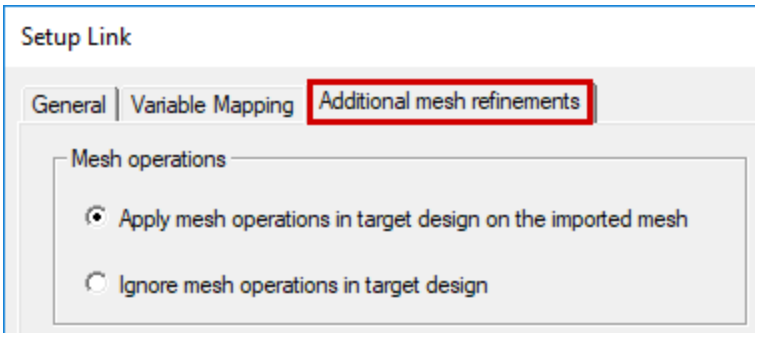
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- Variables can only be mapped by name if they already exist in the Mechanical design. When you copy and paste geometry from the source, or import it from the clipboard, variables used in the definition of the geometry automatically come into the Mechanical design. However, for non-geometry-related variables (such as a variable voltage or wattage applied to an excitation in the HFSS *Edit post process sources* dialog box), the variables will not automatically appear in the Mechanical design. There is no consequence to this limitation though, since only variables that affect the geometry will affect the modal results.
- When the *Value Mapped to Source Design* column contains constants, those constant values will be passed to the source design even if you change the associated variable's value in the target design's *Properties* dialog box (*Mechanical* > *Design Properties*) or in the docked *Properties* window (with the target design selected in the Project Manager).
- If you have not mapped the detected variables by name, you will be prompted with a warning upon attempting to close the *Setup Link* dialog box. The warning will allow you to return to the *Variables Mapping* tab to click the *Map Variable by Name* button. Alternatively, you can accept the unsynchronized variables and close the dialog box.

9. In the **Additional mesh refinements** tab of the *Setup Link* dialog box, choose one of the following two options:
   - **Apply mesh operations in target design on the imported mesh**
   - **Ignore mesh operations in the target design**

   (depending on whether or not you want the target Mechanical solution to perform further mesh refinement on the imported mesh).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Changing the *Additional mesh refinement* settings does not invalidate the cached link data (meshes from source design).
- When the source and target for mesh linking are both Mechanical–Thermal designs in which one or more Thermal Contact assignments exist, mesh operations cannot be performed in the target design. If you attempt to do so by selecting the first option, a warning message is produced stating that mesh operations cannot be performed on the target design. The warning message won't prevent the solution from running. The solution will be based on the unmodified imported mesh.

10. Click **OK** to close the *Setup Link* dialog box.
11. If you are done with the thermal solution setup, click **OK** to close this dialog box too.

12. Optionally, for Thermal solution models including an imported vacuum, fluid, or air region that you want to exclude from participating in the thermal solution, select that object in the History Tree. Then, in the docked *Properties* window, clear the **Solve Inside** option.

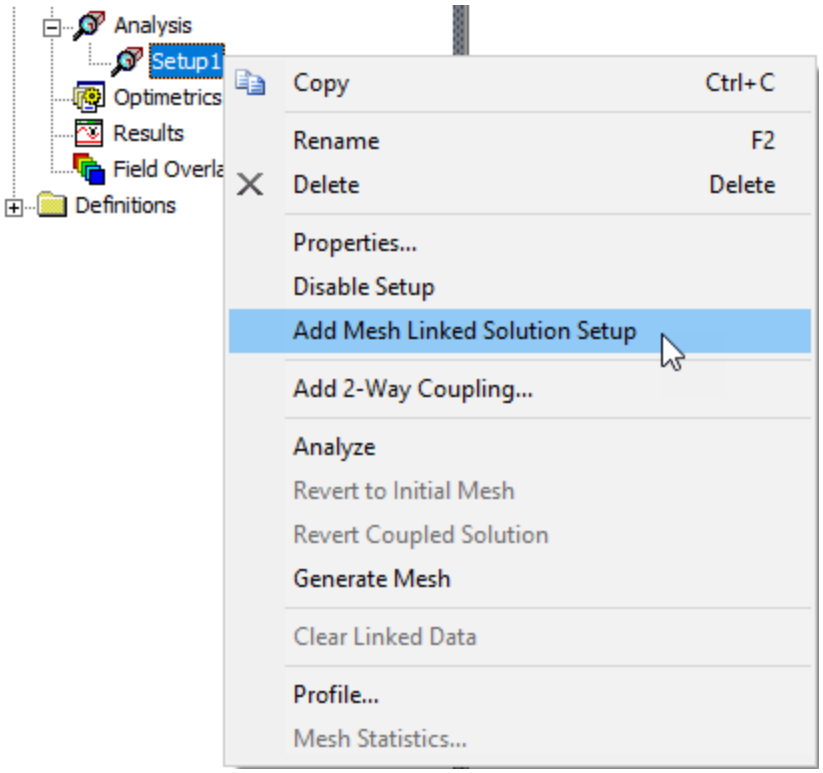
Tip:

If you define a link setup for importing a mesh and later revisit the *Link Setup* dialog box, it will be in *View Only* mode. The settings will be displayed but unavailable for editing. Select the **Edit Link** option under the *General* tab to return the dialog box to the editing mode.

If you want to import the mesh from an existing solution setup to a new solution setup in the same design, a command is available to automate the process. The program can automatically create a new target solution setup that is configured to link to, and import the mesh from, the source solution setup. To use this feature, do the following:

1. Under *Analysis* in the Project Manager, right-click the existing solution setup that you want to use as a mesh source for a new setup.
2. Choose **Add Mesh Linked Solution Setup** from the shortcut menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The new setup is added under *Analysis* in the Project Manager. If you look at the properties of this new setup, you will see that the link to the setup that you initially right-clicked is already defined.

3. Optionally, edit the settings in the new analysis setup as desired.

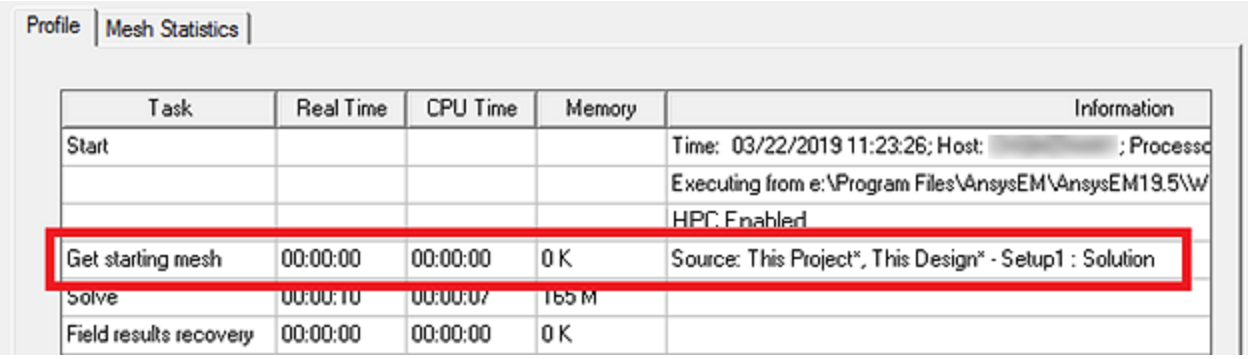
Verifying Mesh Linking

There are two ways to verify that mesh linking has been performed after the source and target setups have been analyzed:

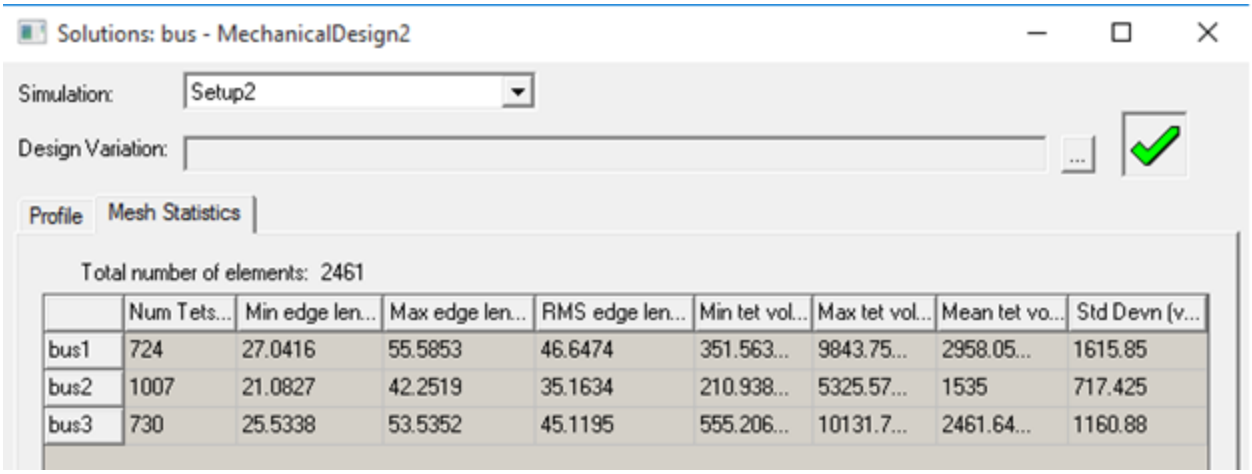
- Under *Analysis* in the Project Manager, right-click the target setup and choose **Profile** from the shortcut menu.

  The mesh import step is listed in the profile:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Under *Analysis* in the Project Manager, right-click each of the setups in turn (source and target) and choose **Mesh Statistics**. The mesh statistics should be identical between the two designs, providing you haven't solved the second setup with additional mesh operations (refinements) defined.



Mesh Linking Requirements

As stated previously, you can reuse a mesh from an existing source design in a target design if all objects have matching names and identical geometry (based on each object having the same model history in both designs). The easiest way to achieve this result is to copy and paste the items from one design into the other. The objects will be supported for mesh linking regardless of the order of the copy/paste operations.

In the following procedure, all objects are copied and pasted from one design to the other in a single operation:

- If your source and target model are both Mechanical designs, copy and paste the complete design, as follows:
  1. In the *Project Manager*, right-click the ***source design*** and choose **Copy**.
  2. Also in the *Project Manager*, right-click the ***project name*** and choose **Paste**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Edit the **Solution Type** if needed (for example, if the source design is a Mechanical – Thermal solution, and the target is to be a Structural solution).

- If your source and target model are different design types:
  1. Select all of the objects in the source design using one of the following three methods:
     - In *Object* selection mode, click in the *Modeler* window of the *source* design to ensure that it's the active window and then press **Ctrl+A**.
     - In the History Tree of the source design, right-click **Model** and choose **Select All**.
     - In the *Object selection mode*, click and drag the mouse to draw a selection box enclosing all source model objects.

       > **Note:**
       >
       > If you draw the selection window from left-to-right, you must fully enclose every object. If you draw it from right-to-left, every object that the window encloses and those that it crosses through are selected. So, in the latter case, the selection window does not have to fully enclose every object in the model to select them all.

  2. Copy the selected objects to your Clipboard using one of the following four methods:
     - Press **Ctrl+C**.
     - On the **View, Draw,** or **Model** ribbon tab, click **Copy**.
     - Right-click the Modeler window and select **Edit** > **Copy** from the short-cut menu.
     - Using the menu bar, click **Edit** > **Copy**.
  3. In the *target* design, use one of the following four methods of pasting the objects:
     - Click in the display area of the Modeler window and press **Ctrl+V**.
     - On the **View, Draw,** or **Model** ribbon tab, click **Paste**.
     - Right-click the Modeler window and select **Edit** > **Paste** from the short-cut menu.
     - Using the menu bar, click **Edit** > **Paste**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The *Import from Clipboard* command in the *Modeler* menu is still available for copying source geometry into a target design. However, it is no longer required to make mesh-linking-compliant geometry. Using this command can have unexpected consequences, as it clears the History Tree.

After you create the duplicate model using one of the above methods, create a *Solution Setup* in the target design. Use the procedure outlined previously under How to Import a Mesh.

Cached Link Data and Solution Invalidation

The meshes from the source design are cached in the target design after its simulation. You can remove these cached meshes (instructions below).

If you've updated the source design and want the imported mesh in the target design to reflect the changes in the source project, you need to clear the linked data. Doing so invalidates the previous solutions. Clear the linked data using one of the following two methods:

Method 1: (to clear linked data and/or solutions)

1. Using the menu bar, click **Mechanical** > **Results** > **Clean Up Solutions**.

   The *Clean Up Solutions* dialog box appears.

2. In the Solutions section of the dialog box, choose one of the following sets of options:
   - Select the **Linked Data Only** option.
   - Select the **Fields and Meshes** and **Include Linked Data** options.
   - Select the **All Solutions** and **Include Linked Data** options.
3. Click **Do Deletions**.

Method 2: (to clear linked data only)

- Under *Analysis* in the Project Manager, right-click a solution setup and choose **Clear Linked Data** from the shortcut menu.

  This action will clear all linked data in the target design. For example, if your design has two EM Loss links and three solution setups that each include a mesh link, data from all five links will be cleared.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Current meshes and previously solved results are invalidated when the cached mesh is cleared.
- Meshes are cached for each solution setup with an imported mesh specified.
- The cached mesh is automatically cleared when the target Mechanical design is edited in a way that invalidates its mesh (such as adding, removing, or modifying geometry).
- Cached meshes in the target (Mechanical) design are *not* automatically removed when the *source* design is edited.

Specifying Initial Mesh Settings

You can specify the initial mesh settings, including the surface approximation and the meshing approach. Initial Mesh Settings apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Note:

When you have defined mesh linking between a source and target design, all parameters within the *Initial Mesh Settings* dialog box are ignored for the target design. The mesh is copied from the source design without modification.

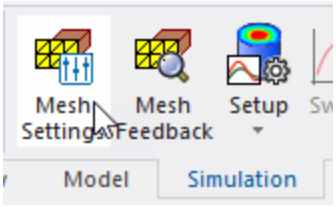
For most designs, you can let the solver automatically choose which of two meshing approaches to take. Each solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics.

In a few cases, you may decide to override the automatic choice and designate the mesher to use. To do so:

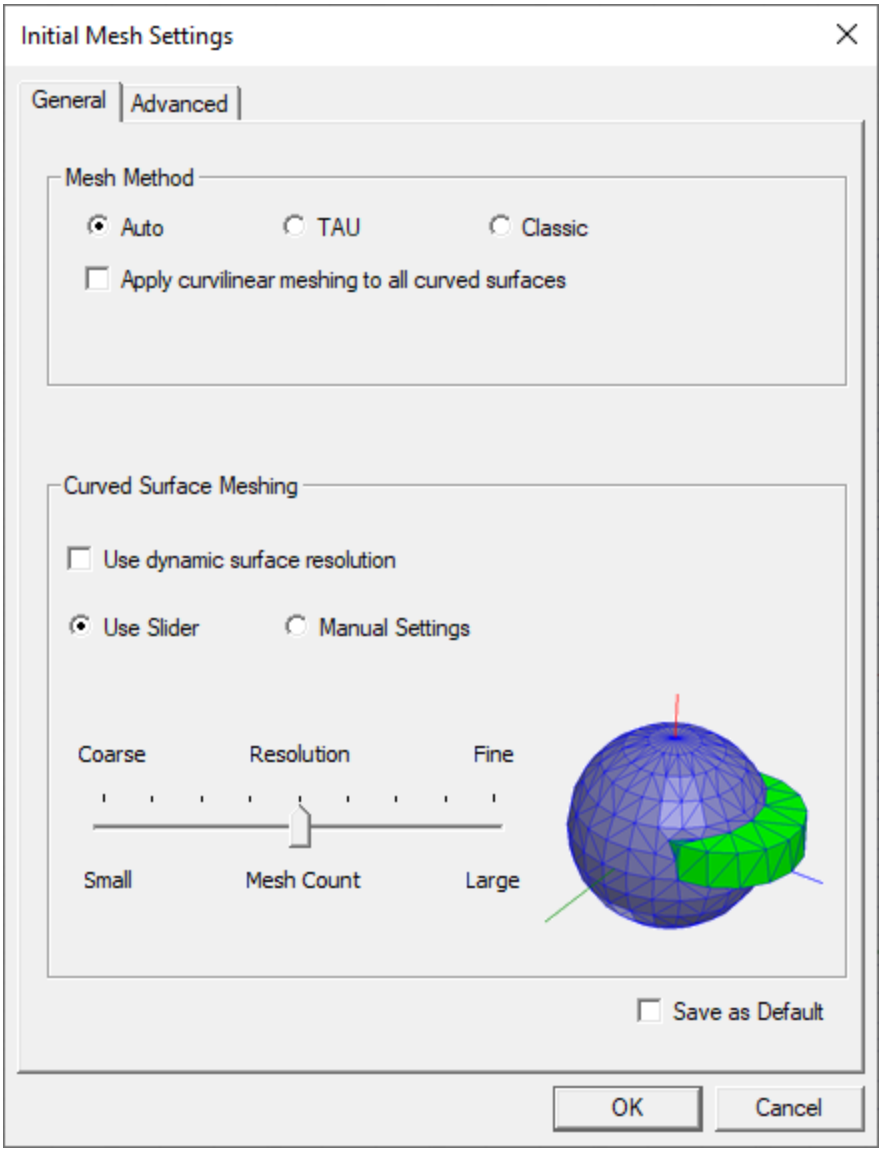
1. Access the **Initial Mesh Settings** one of two ways:
   - Select **Mechanical** > **Mesh** > **Initial Mesh Settings...**
   - In the Project Manager, right-click **Mesh**, and select **Initial Mesh Settings** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Simulation** tab of the ribbon, select **Mesh Settings**.



The **Initial Mesh Settings** window appears, with the **General** tab selected.



2. The **General** tab contains **Mesh Method** options for:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Auto** – the solver automatically selects the mesher. This is the default setting.
- **TAU** – only specific curve faces will be remeshed (for example, equation-based axisymmetric faces). If a curve face connects to non-remeshed curve faces, the curve face will not be remeshed. If a curve face cannot be remeshed, faceting triangles will be used as the surface mesh. Mesh quality depends on how the faceting triangles are made. If the surface mesh is generated by TAU, it shows in the profile as Mesh TAU (Surface). This can fall back to Mesh TAU (Wrapper), which can fall back to the classic Mesh (Stitch).
- **Classic** – Classic mesh uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately. Depending on the model, Classic mesh may converge using fewer resources whereas TAU or Phi Plus may converge faster.

3. For Mechanical models with curved surfaces, you can select or deselect **Apply curvilinear meshing to all curved surfaces**. This increases accuracy, but costs more memory.
4. Under **Curved Surface Meshing**, you can choose to **Use dynamic surface resolution**, select **Use Slider** or specify **Manual Settings**.

   If you select **Use dynamic surface resolution**, the **Manual Settings** option is disabled.

   The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*. Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

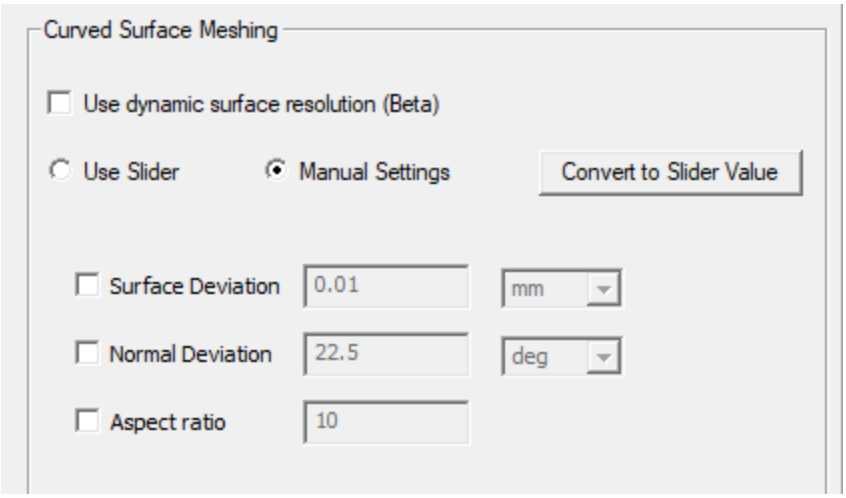
   Selecting **Use dynamic surface resolution** in the *Curved Surface Meshing* section specifies the best-practice mesh operations over the geometric models. This mesh operation will support 3D volume mesh and surface mesh in all products. The default mesh operations or user-defined mesh operations may or may not be replaced by the optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get more smooth curvature representation while small curve faces, such as fillets, small curve objects, and so forth, get relatively coarse triangulations so that the overall mesh count will be reduced.

   If you select dynamic surface resolution, you can optionally use the slider bar to adjust surface representation levels as: coarse (1-3), normal (4-6), and fine (7-9). You can also adjust the surface representation level for a specific group of faces by specifying slider bar, as described in Modifying Surface Approximation Settings.

   > **Note:**
   >
   > For Mechanical–Thermal and Mechanical–Structural solutions only, the effects of the mesh slider are different than for other design and solution types. Please see the Mechanical Mesh Slider topic for more information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you choose **Manual Settings**, the window changes to show text fields. See: Modifying Surface Approximation Settings.

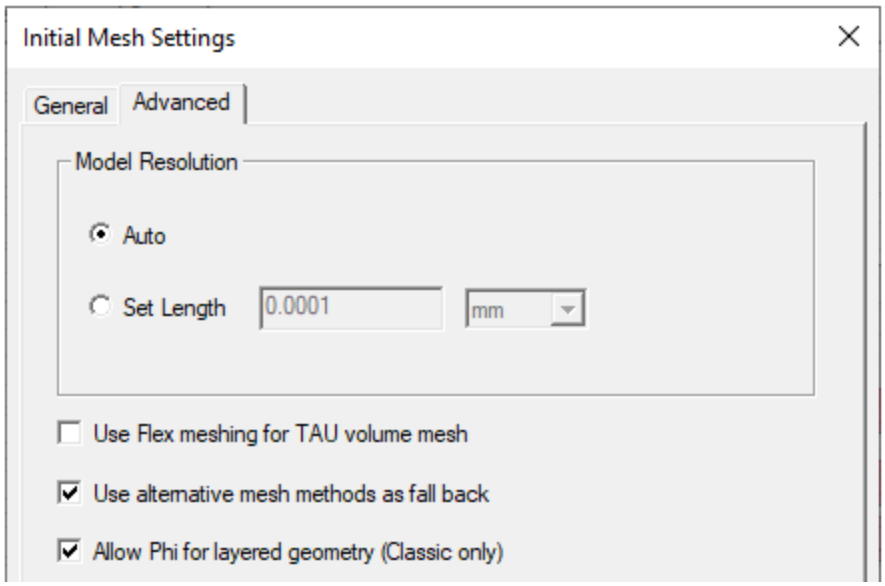


   Use the check boxes to enable the fields and specify values:

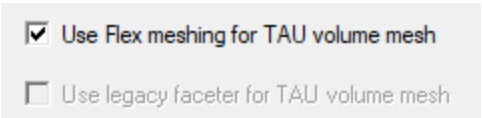
   - **Surface Deviation** – the distance between the true surfaces of the selected faces and the meshed faces.
   - **Normal Deviation** – the angular distance between the normal of the true surface and the corresponding mesh surface.
   - **Aspect Ratio** – determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
   - Clicking **Convert to Slider Value** converts the manually entered values to an equivalent slider setting, and returns the panel to the slider view.

6. To make your choices the default, use the **Save as default** check box.
7. The **Advanced** tab lets you optionally specify a **Set Length** for **Model Resolution**. This is for experienced users who have a good understanding of how particular values will affect their models. In general, the **Auto** setting provides good results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

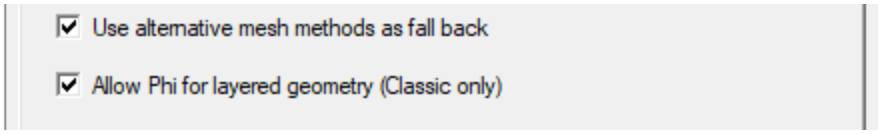


8. **Use Flex meshing for TAU volume mesh** enables a version of the TAU mesher that will rarely fail to generate a mesh. In most cases, the TAU Flex mesh is as accurate as traditional TAU or Classic meshes. However, for some complex models with bad translation or poorly defined surfaces that would fail to produce a strict mesh on all objects, relaxed tolerances will be applied. In these cases the user should review the mesh to evaluate whether it is acceptable for simulation. See TAU Flex meshing for more details.



9. **Use legacy faceter for TAU volume mesh** – By default, TAU uses the latest faceter to generate geometric model for meshing. Occasionally, the mesher may fail, and TAU will automatically fall back to use the legacy faceter to make a second attempt. In such cases, directly using the legacy faceter can get the mesh in the first attempt so total meshing time will be saved. The legacy faceter is not actively maintained by Spatial and will be retired soon. This option should be used only as a last resort.
10. If your selected mesh method (Classic/TAU) failed to generate initial mesh, the solver can either stop or fall back on alternative mesh methods if you select **Use alternative methods as fall back**. You can also select **Phi for layered geometry (Classic only)**. For example, specifically, the mesher might go like this: When mesh method is TAU: TAU (>Phi)>Classic. When mesh method is Classic: Classic(>Phi)>TAU. Note: the options do not affect the Auto mesh setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



11. Click **OK** to apply your choices.

The settings will be applied to the initial mesh generated.

Mechanical Mesh Slider

There is a mesh size slider in the *Initial Mesh Settings* dialog box. Though this setting is shared by multiple design types, the effect of the slider has been enhanced for three of the four *Mechanical* solution types, as follows:

- Steady-State Thermal
- Transient Thermal
- Structural

The following enhancement has been implemented for the three above-listed Mechanical solution types:

- The mesh slider controls the initial mesh density and element quality for **all** faces (flat and curved) and also for the interior mesh of all solid objects.

The intent of the enhanced mesh slider is to greatly reduce the necessity of manually assigning mesh operations to obtain accurate results from thermal and structural solutions. Even at the default slider position, midway between Coarse and Fine, the mesh quality is significantly improved relative to earlier versions of the software.

For *Modal* solutions, the behavior of the slider remains the same as it is for other design types in the Ansys Electronics Desktop application. Specifically, the slider continues to affect the initial meshing behavior only at *curved* surfaces. Flat faces are unaffected.

In all cases, the mesh slider control is global, affecting all applicable objects in the model.

> **Note:**
>
> A similar mesh slider appears in the *Surface Approximation* dialog box. That instance of the slider is unmodified and affects all products and solution types in the same way, including thermal and structural solutions. That is, it continues to affect only curved faces and to use the original refinement scheme.

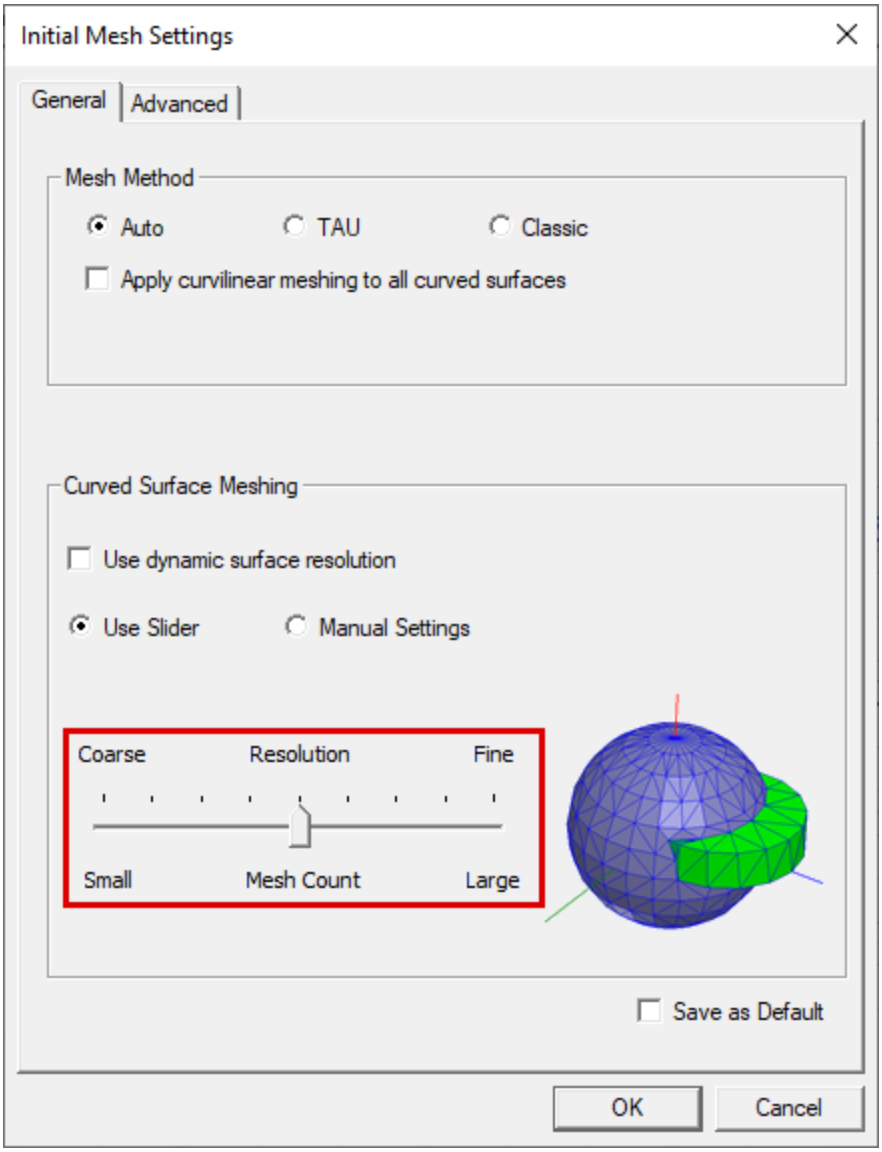
User-defined mesh operations subsequently refine the initial mesh locally (for specified objects or faces), providing that the mesh operation dictates smaller elements than the initial mesh

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

includes. In other words, a mesh operation will not force a coarser local element size, only a finer size.

Access the mesh slider using one of the following two methods:

- In the Project Manager, right-click **Mesh** and choose **Initial Mesh Settings** from the shortcut menu.
- Using the menu bar, click **Mechanical** > **Mesh** > **Initial Mesh Settings**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When you set up Mesh Linking to duplicate the mesh of a source design in your Mechanical design, the mesh slider and all other parameters within the *Initial Mesh Settings* dialog box are ignored. The mesh is copied from the source design without modification. However, you can optionally choose whether or not to ignore mesh operations defined in the target design. If not ignored, target design mesh operations will produce further local mesh size refinement.

Adjusting the Mesh Density

Adjust the slider position towards the right to produce a finer mesh (smaller elements, larger element count). A finer mesh generally improves the quality of the elements and the accuracy of the solution. However, excessive refinement leads to greater solution time and greater memory/storage requirements, without significant, additional improvement of the results.

Adjust the slider position towards the left to make a coarser mesh (larger elements, smaller element count). Sometimes a relatively coarse mesh is adequate for simple shapes and simple thermal or structural loads and boundaries.

Based on characteristics of the geometry, and on the slider position, the program will automatically define on-selection and inside-selection mesh controls. This process happens in the background, and the automatic mesh operations are not listed in the *Mesh* branch of the Project Manager, nor are they accessible elsewhere.

Mesh operations that you manually assign to selected faces or objects, whether on or inside the selection, will override the automatic mesh controls for the selected entities, providing the effect is to make the local mesh finer. Mesh operations will not force a coarser mesh than the global sizing parameters dictate.

Tip:

To more quickly see the effect of any given slider adjustment, you can generate the mesh without running the solution. Simply right-click the solution setup, under *Analysis* in the Project Manager, and choose **Generate Mesh**.

TAU Flex Meshing

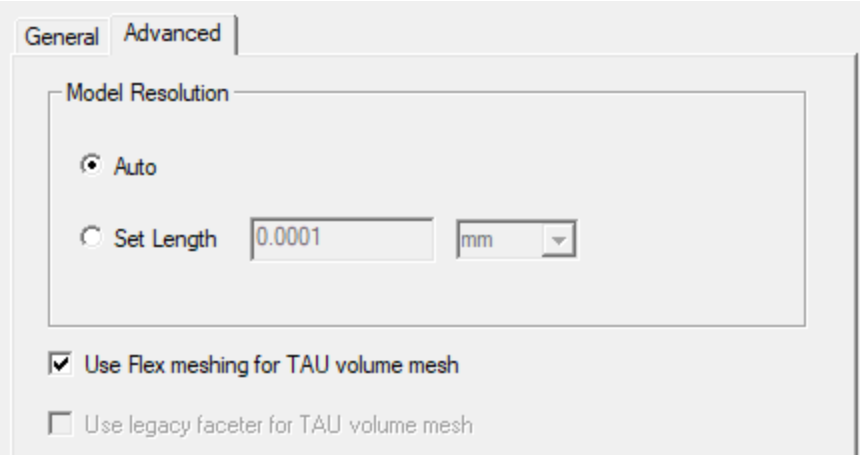
TAU Flex is designed for guaranteed mesh success, and may also reduce meshing time. It provides a mesh even for complex and “dirty” geometry model without any preliminary geometric healing or repair.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

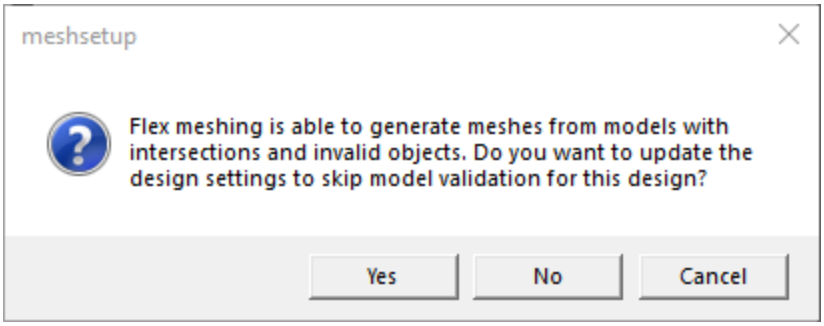
TAU Flex is a multi-domain, priority mesher which provides more efficient and reliable meshing. The priority mesh generation is driven by solver requirements for materials (metal/non-metal), boundary conditions (ports, radiation,...), conduction paths, and so forth. The constraint-tolerant meshing method gives the TAU Flex mesher flexibility to excuse mesh difficulties in lower-priority regions so that the resulting mesh can still provide acceptable simulation results.

Steps for Using TAU Flex

1. Navigate to the **Initial Mesh Settings** window.
2. Click the **Advanced** tab and select **Use Flex meshing for TAU volume mesh**.



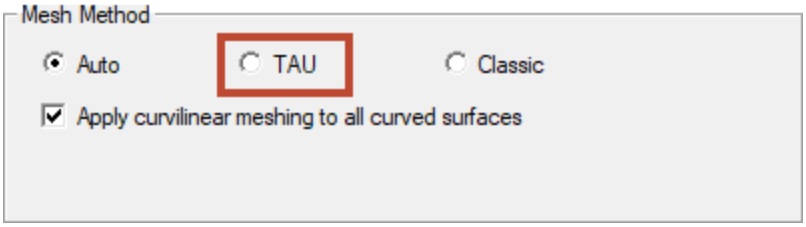
Selecting this option opens a dialog box. Select **Yes** to continue.



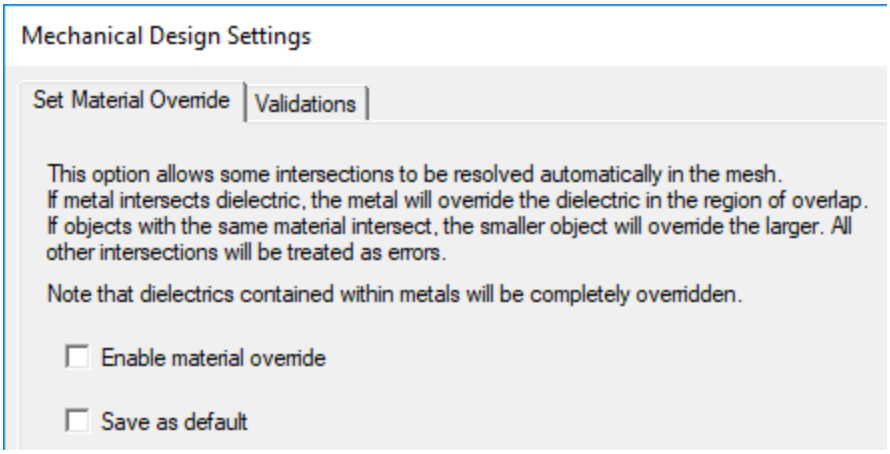
To get the full benefits of TAU Flex, especially for complex or "dirty" models, follow these recommended steps:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **General** tab of the **Initial Mesh Settings** window, select **TAU**.



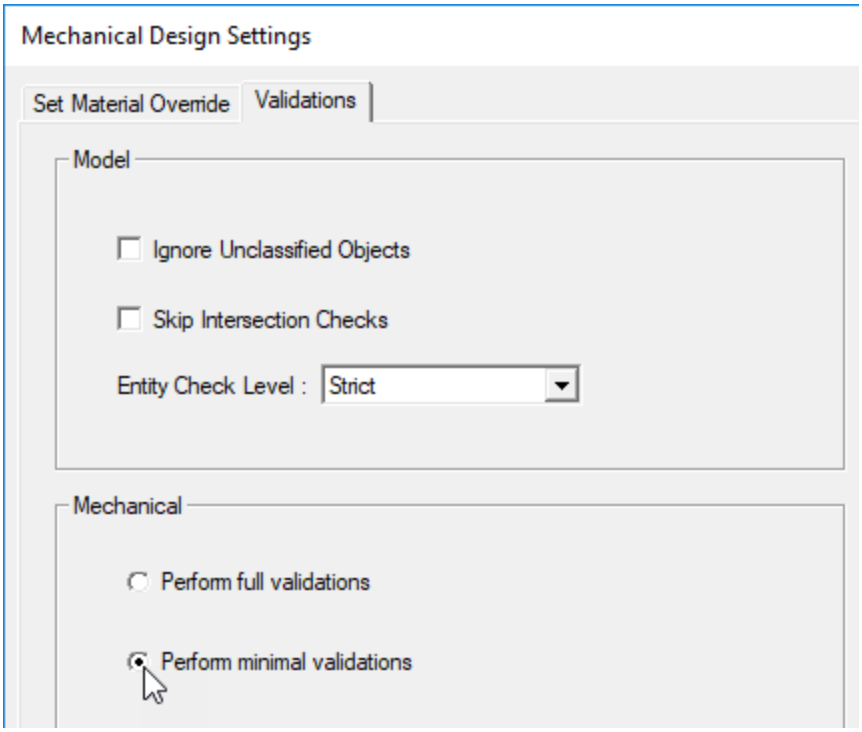
2. Click **Mechanical** > **Design Settings** to open the **Design Settings** window.
3. On the Set Material Override tab, select **Enable material override**.



This activates TAU's priority meshing method based on metal/non-metal materials.

4. Still in the Design Settings window, click the **Validations** tab and select **Perform minimal validations**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

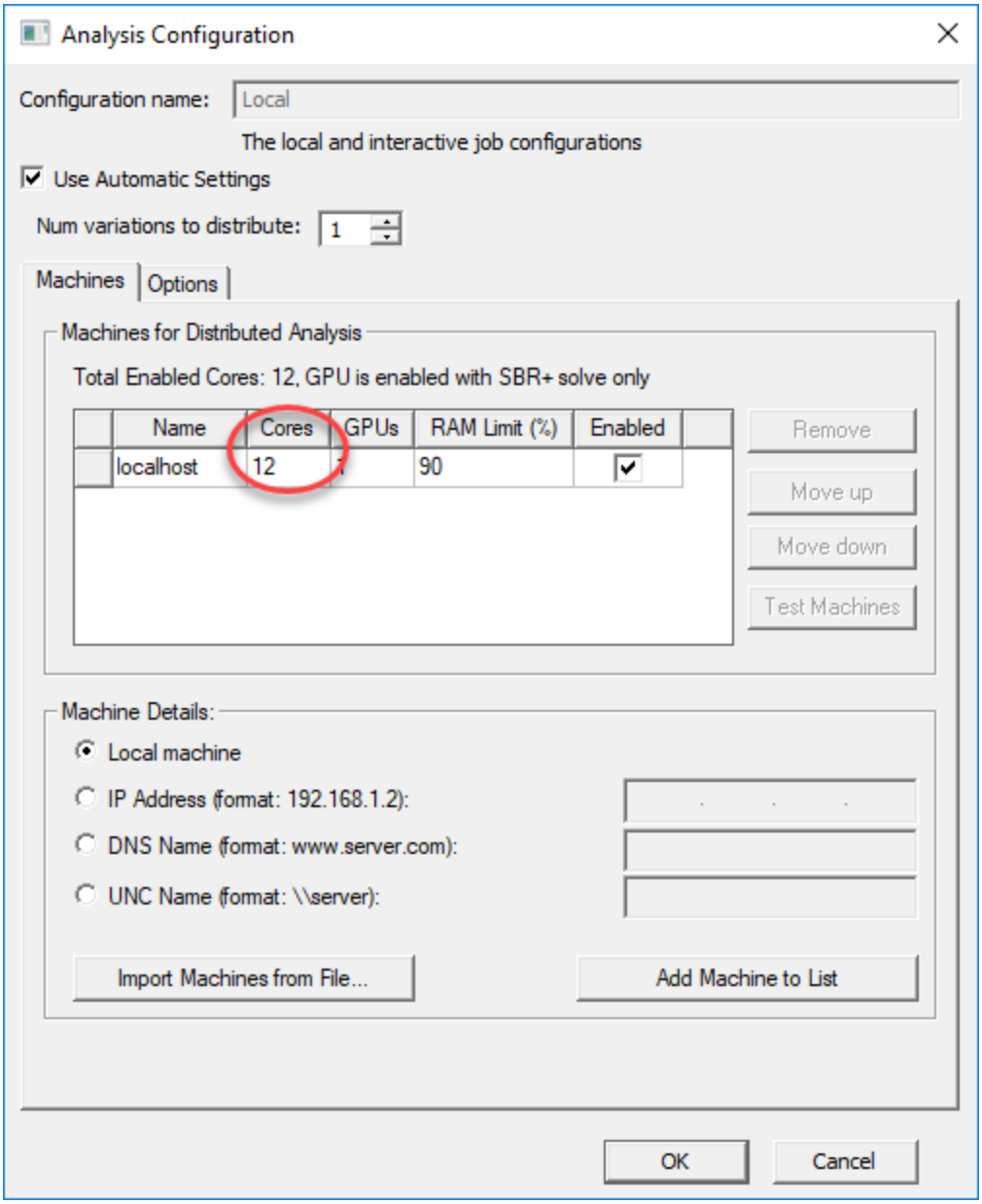


This reduces preprocessing time for complex models, and allows simulation of "dirty" geometry for which TAU Flex can provide a mesh.

5. Because TAU Flex is a parallel mesher for many processes, you should specify multiple cores in **HPC and Analysis Options** to speed up the meshing process. Access the Analysis Configuration window one of two ways:
   - Select the Simulation tab on the ribbon and click **Analysis Config**.
   - Use **Tools** > **Options** > **HPC and Analysis Options** or select the **Simulation** tab on the ribbon and click **HPC Options**. Then double-click a configuration.

   The **Analysis Configuration** window appears.

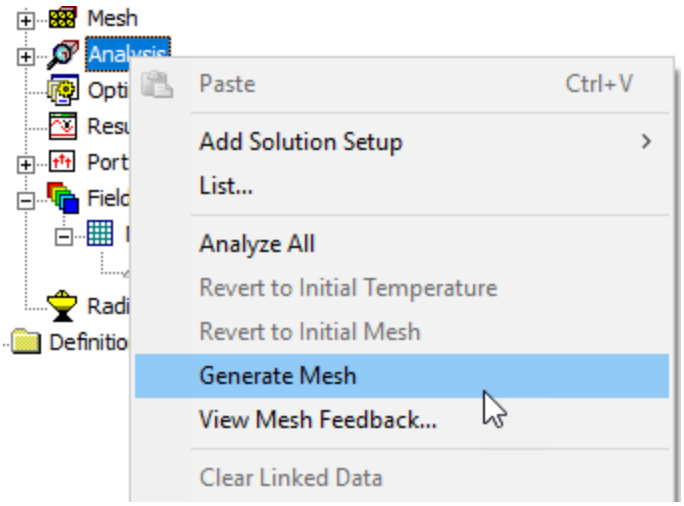
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify the number of cores to use.

6. In most cases, TAU Flex can provide mesh as accurate as TAU or Classic, but not always. Therefore, we recommend that you **Generate Mesh** without solving, and that you review the mesh before proceeding the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



TAU Flex Warning Messages

In cases where TAU Flex has difficulty making a strict mesh on some objects, mesh warnings are given for those objects. You should review the mesh on those objects before proceeding to the simulation. You can exercise discretion in deciding whether a given mesh issue needs to be addressed (for example, critical antenna geometry) or can be ignored (for example, a small "leak" in a dielectric substrate far from the source).

There are three types of warnings:

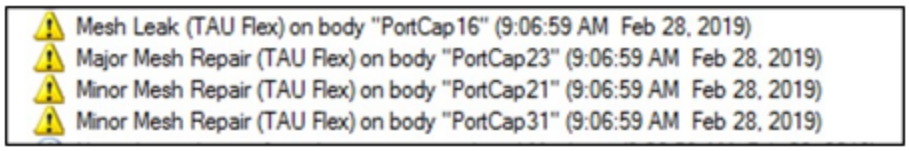
- **"Mesh Leak"** – *The most severe warning*. A body does not have any mesh element, or it has mesh elements with a significant mismatch of surface area between the mesh and geometry – larger than 10% of planar area, or 20% of curved area.
- **"Major Mesh Repair"** – A body has major surface area mismatch between the mesh and geometry – larger than 5% of planar area, or 10% of curved area.
- **"Minor Mesh Repair"** – *The least severe warning*. A body has minor surface area mismatch between the mesh and geometry – less than 5% for planar area, or 10% for curved area.

Reviewing TAU Flex Mesh Warnings

TAU offers three ways to view mesh warnings.

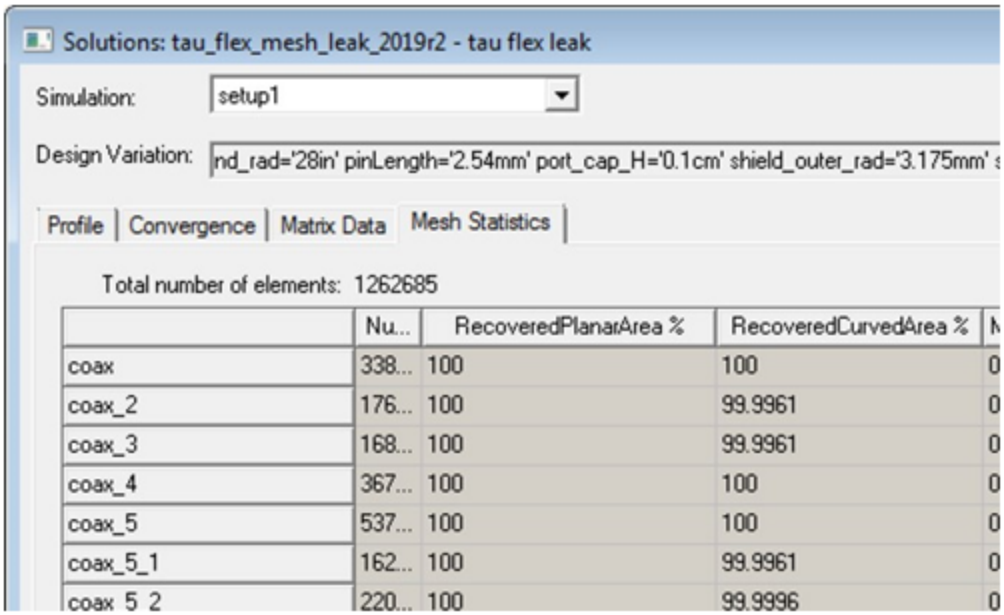
1. **Message Manager** – Immediately after the mesh generation is done, if a tolerant mesh is given, warnings display in the Message Manager. If the window is hidden, click **Show Messages**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Mesh Statistics** – On the **Results** tab of the ribbon, click **Solution Data** to open a **Solutions** window.

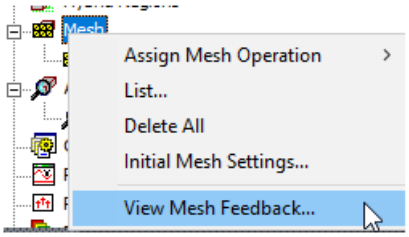
   On the **Mesh Statistics** tab of the **Solutions** window, if a tolerant mesh is generated, the surface area difference between the mesh and geometric model at body level is listed as "RecoveredPlanarArea" and "RecoveredCurveArea". 100% means that the body has a strict mesh. You can click the surface area tab to sort the body list and see the tolerant mesh bodies first.

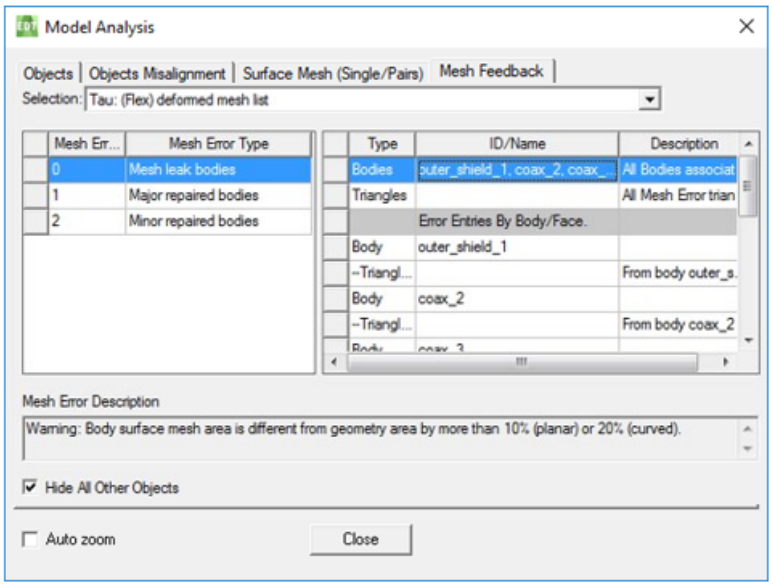


	Nu...	RecoveredPlanarArea %	RecoveredCurvedArea %	
coax	338...	100	100	0
coax_2	176...	100	99.9961	0
coax_3	168...	100	99.9961	0
coax_4	367...	100	100	0
coax_5	537...	100	100	0
coax_5_1	162...	100	99.9961	0
coax_5_2	220...	100	99.9996	0

3. **Model Analysis** – To view the details of the tolerant mesh, click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback** to launch the **Model Analysis** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Click on the top list to display objects that have tolerant mesh.

Click the "Triangles" to display the mesh on the tolerant objects.

Individual objects are listed under "Error Entities By Body/Face". You can select one or multiple bodies to view the mesh at a more detailed level.

Addressing TAU Flex Mesh Issues

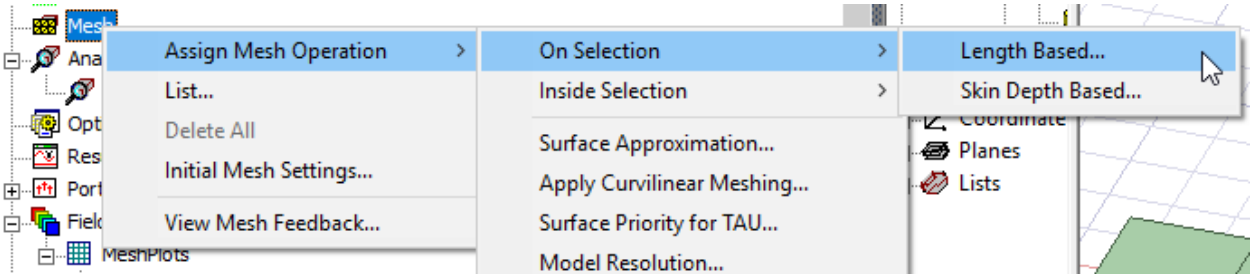
In cases where TAU Flex has difficulty making a strict mesh on some objects, you can address the problem using one of the following techniques:

1. Redraw the geometry in question in the native modeler.
2. Heal the geometry, either with integrated healers or perhaps in Discovery.
3. If the object has curved or true surfaces, consider applying a localized surface mesh operation which creates a finer than default mesh.
4. Remove difficult objects that are not critical to the simulation.

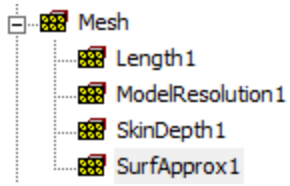
Defining Mesh Operations

In Project Manager, mesh operations are optional mesh refinement settings that provide the solvers with mesh construction guidance. This technique of guiding mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Assign Mesh Operation** commands on the respective solver menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each mesh operation you define appears in the Project tree.



Note:

There are no restrictions against assigning multiple mesh operations to the same entities. The *type* of mesh operation can differ between overlapping assignments. For example, length-based refinement could be assigned inside a body, and skin-depth seeding on one or more of its faces. Additionally, overlapping mesh operations of the *same type* can have differing, or even redundant, parameter values (such as element length) without causing an error condition.

Typically, the operation that dictates the finest local meshing results will control the outcome in overlapping assignment areas or volumes.

You can instruct the solvers to reduce the maximum size of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can instruct solvers to refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement). These types of mesh operations are performed on the *current mesh*, that is, the most recently generated mesh.

In a few circumstances, you may also want to create a mesh operation that modifies a solver's surface approximation settings for one or more faces. Surface approximation settings are only applied to the *initial mesh*, that is, the mesh that is generated the first time a design variation is solved.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also choose to override automatic choice of which mesher a product uses, by using the [*solverName*] > Mesh > Initial Mesh Settings command.

See the technical notes for more details about Ansys Electronics Desktop's application of mesh operations.

What do you want to do?

- Length Based Mesh Refinement
- Perform length-based mesh refinement on object faces
- Perform length-based mesh refinement inside objects
- Perform skin depth-based mesh refinement on object faces
- Modify surface approximation settings for one or more faces
- Apply Curvilinear Elements
- Specify automatic or specified model resolution for a selection
- Specify the initial mesh settings
- TAU Flex Meshing

Length Based Mesh Refinement

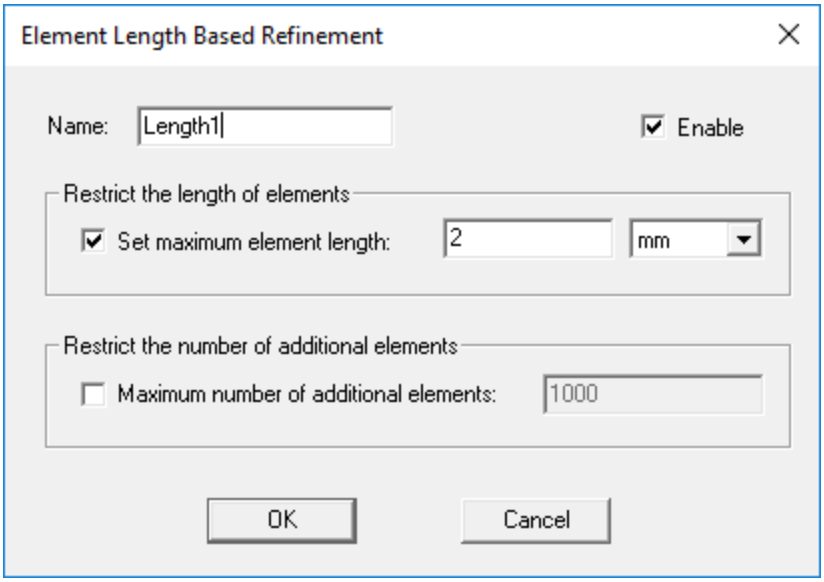
Length-based mesh refinement settings help control the length of the tetrahedral elements and the growth of the whole mesh. If you are not sure how big a refinement is needed, by way of mesh refinement you can set a ceiling on the mesh growth with the option **Restrict the Number of Additional Elements**.

To apply a length based mesh operation either to an object face or inside an object:

1. Select the object in the modeling workspace.
2. In the Project Manager, right-click **Mesh** > **Assign Mesh Operation** > **[On Selection / Inside Selection]** > **Length Based**.

   The **Element Length Based Refinement** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter a **Name** for the mesh operation, or leave the default.
4. Select the **Enable** check box to set the operation as active. You can later deselect this to deactivate it.
5. To restrict the length of tetrahedra edges, select the **Set maximum element length** check box and specify values and units using the drop-down menus.
6. To restrict the number of elements added during refinement, select the **Maximum number of additional elements** check box and specify a value.
7. Click **OK**.

When you set a mesh operation on a selection of objects (more than one) and set a maximum length, it applies to the whole mesh. Even if you selected just one object, when you refine its boundary, elements are added both inside and outside of that object.

When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, matching boundary issues performed at the finishing stage that might slightly alter the mesh size.

If you restrict the number of elements to say 10000, it will add 10000 elements. If you have multiple mesh operations each for 10000 elements, each mesh operation will add 10000 elements (assuming there was no element length targets).

If you set a restriction of say 2000 elements on an object which creates 1100 elements after lambda refinement without any mesh operation, the starting mesh will contain more than 3000 elements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Length-Based Mesh Refinement on Object Faces

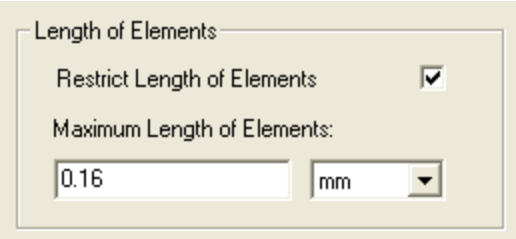
1. Select the faces you want to refine.

   Alternatively, select an object if you want to refine every face on the object.

2. Click **Mechanical** > **Mesh** > **Assign Mesh Operations** > **On Selection** > **Length-Based** or right-click **Mesh** in the Project Manager and select **Assign Mesh Operation** > **On Selection** > **Length-Based.**

   The *Element Length-Based Refinement* dialog box appears.

3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of tetrahedra edges touching the faces:



   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the tetrahedral edges touching the faces in the **Maximum Length of Elements** text box.

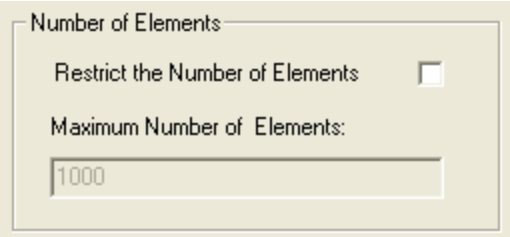
   The solver will refine the element edges touching the selected faces until their lengths are equal to or less than this value.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   For first order basis functions, a maximum length of $\frac{\sqrt{2}}{10}\lambda$ is recommended for radiation boundary surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To restrict the number of elements added during refinement of the faces:



   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.
   c. Click **OK**.

When the mesh is generated, the refinement criterion you specified is used. If the maximum number of elements is reached, some elements may exceed the requested maximum element length.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Assigning Length-Based Mesh Refinement Inside Objects

To instruct a solver to refine every face of an object and its interior:

1. Select the object you want to refine.
2. Click **Mechanical** > **Mesh> Assign Mesh Operation** > **Inside Selection** > **Length-Based**, or right-click **Mesh** in the Project Manager, and select **Assign Mesh Operation** > **Inside Selection** > **Length-Based...**.

   The *Element Length-Based Refinement* dialog box appears.

3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of the tetrahedral element edges inside the object:
   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the edges inside the object in the **Maximum Length of Elements** text box.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected object's faces.

   The solver will refine the element edges inside the object until they are equal to or less than this value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

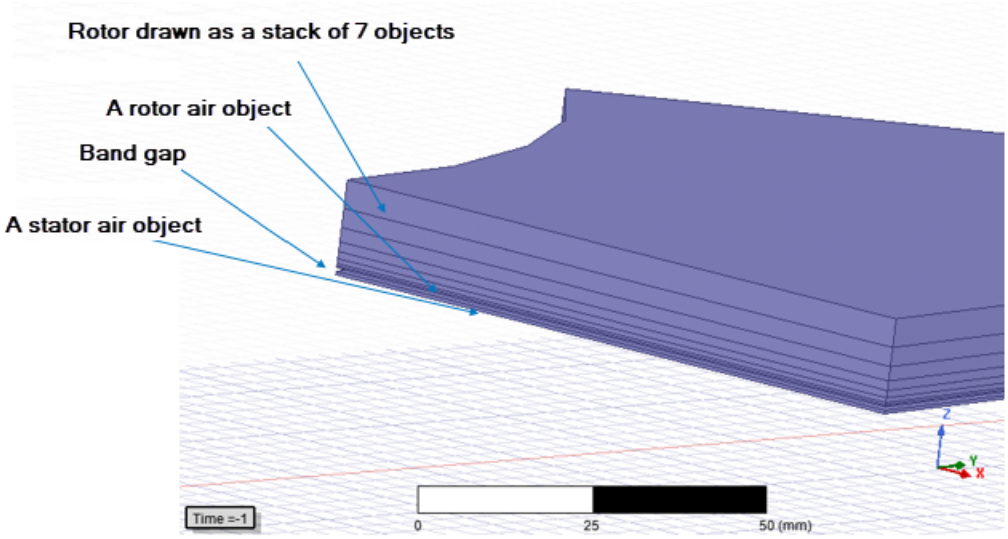
5. To restrict the number of elements added during the refinement inside the object:
    a. Select **Restrict the Number of Elements**.
    b. Enter the **Maximum Number of Elements** to be added.
    c. Click **OK**.

When the mesh is generated, the refinement criterion you specified will be used. If the maximum number of elements are reached, it may result in some elements exceeding the requested maximum element length.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Assigning Skin Depth-Based Mesh Refinement on Object Faces

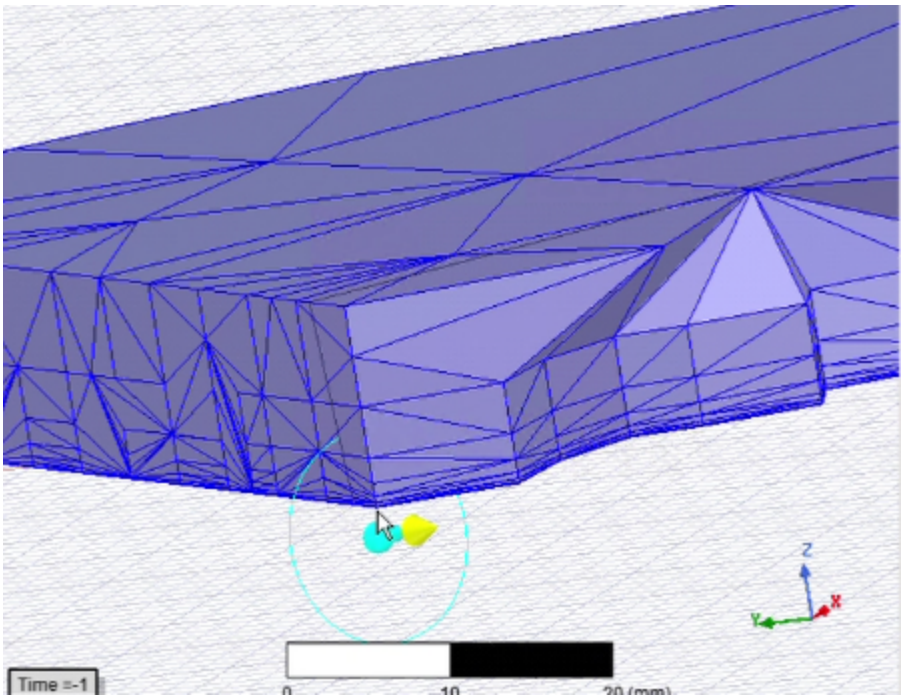
The Skin Depth-based mesh refinement lets you calculate or specify a skin-depth for mesh refinement. You can also specify the number of layers of elements for refinement where the skin depth is the total depth of all layers combined. These layers provide an easy, alternative approach than by creating physical models of each layer, pseudo-sheet bodies. Whereas creating and adjusting a complex, layered physical model is difficult, changing the skin depth parameters and number of layers is very easy. For example, consider a Maxwell rotor, drawn as a stack of seven objects:



To test various stacking models with pseudo bodies involves recreating the model with different thicknesses and remaking the entire initial mesh. The Tau mesher may not be available for all layer heights. You can achieve comparable results by using skin depth based refinement by reverting to the initial mesh, changing settings and applying mesh operations. The following figure shows a cut-plane image of the interior of a model that uses skin-depth based refinement

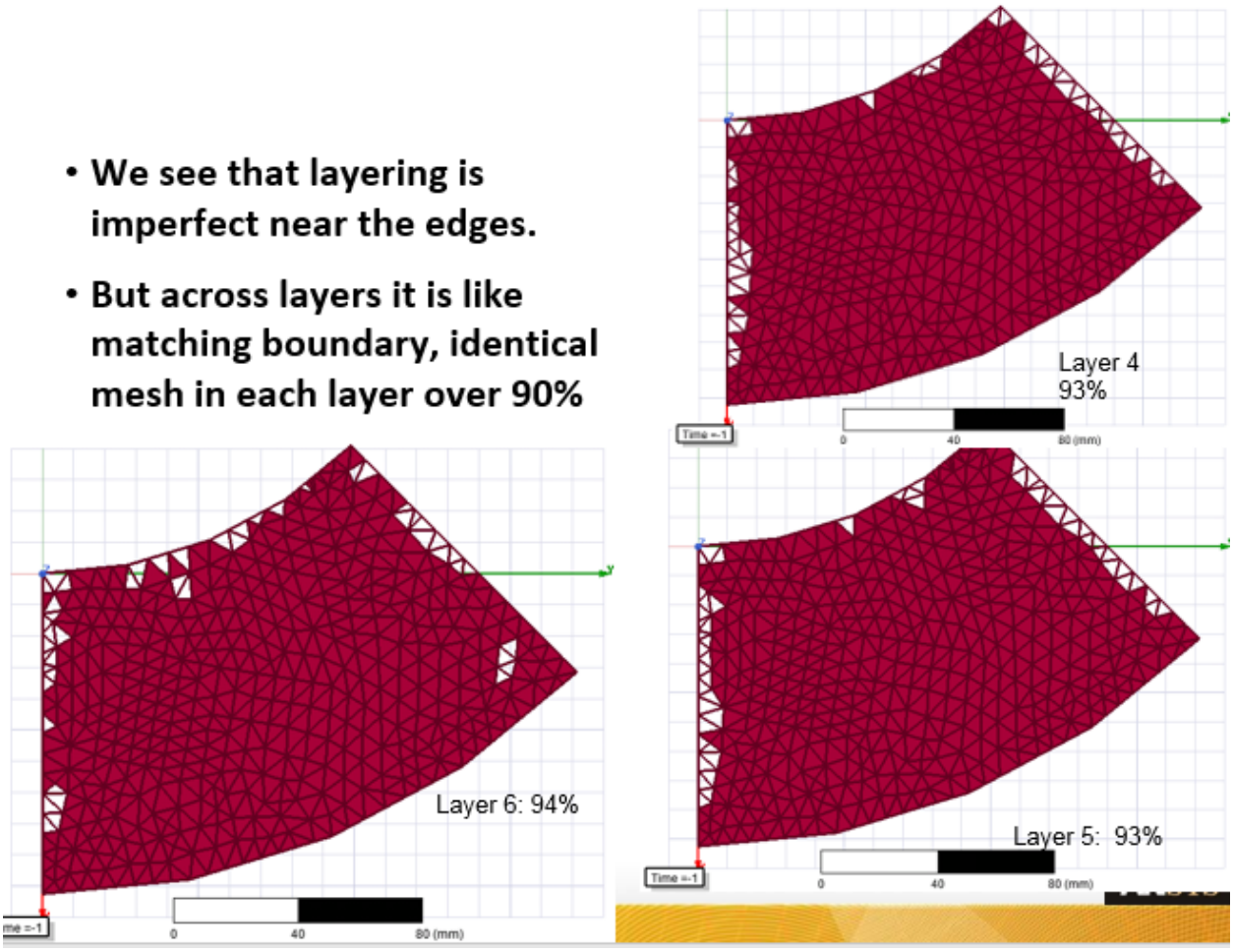
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and layers of elements. The Skin Depth is the total depth of all layers combined. Layered elements apply to the selected faces of solid bodies. The elements are stretched parallel to face, and are compressed in the normal direction.



While in some places the edges have imperfect layering, the interior is very good. You can also view the results layer by layer by using the Model Analysis dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layered skin depth mesh operation provides a tool toward finding a good solution. You can try different settings to improve results toward a good solution. For instance, for this example, these different settings were tried. For comparison, a model built with seven layers required 85K tets.

Tri length target mm	Total skin depth mm	Number of layers	Mesh Size element count	Perfect layering by tri count	Perfect layering by area
15.0	6.3	6	63K	43%	65%
10.0	6.3	6	68K	95%	94%
8.0	6.3	6	76K	96%	93%
6	6.3	6	97K	97%	94%

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The goal is not to have perfect layering, but to have a good solution that is much easier to achieve.

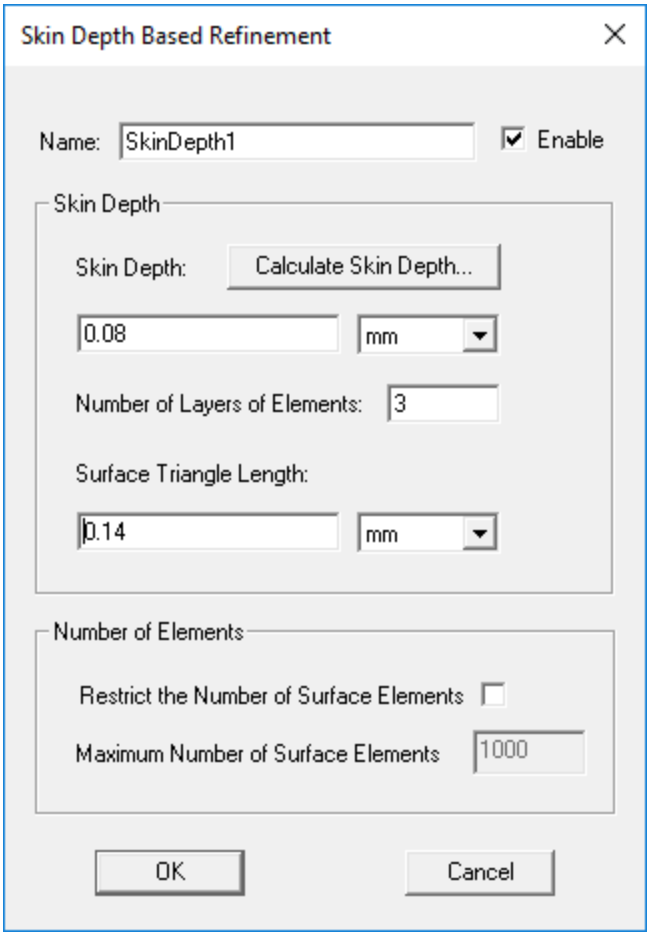
To use Skin Depth-Based Refinement:

1. Select the faces you want to be refined.

   > **Note:**
   >
   > It is possible to select a body and convert it to selecting all faces of the body. The user can use this method to select all faces and toggle a few faces out of selection. Selecting the whole body might select very large regions for refinement and increase the element count a lot.

2. Click **Mechanical** > **Assign Mesh Operation** > **On Selection** > **Skin Depth Based**.

   The *Skin Depth Based Refinement* dialog box appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh will be refined, click **Calculate Skin Depth**.

   The **Calculate Skin Depth** dialog box appears with values based on the selected object's material properties and the solution setup frequency.

   Accept or edit these values. When you click **OK** the solver calculates the skin depth and enters its value in the Skin Depth text box. You can accept the calculated values or provide your own.

4. In the **Number of Layers of Elements** text box, type the number of layers to add perpendicular to the object's surface. The skin depth is the total depth of all layers combined.

   The solver will add an equivalent number of mesh points to each layer. For example, if HFSS added 10 points to satisfy the **Surface Triangle Length**, it will add 10 points to each layer.

5. Optionally, provide the maximum edge length of the surface mesh in the **Surface Triangle Length** text box. The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   The solver will refine the surface triangle mesh (the faces of the tetrahedra touching the surface) until their edge lengths are less than or equal to the specified value.

6. By default, the Restrict Number of Surface Elements setting is unchecked. This allows the mesher to use symmetry more effectively. If you Restrict the number of surface elements, this may affect symmetry but can be used to protect against runaway refinement in specific cases. To restrict the number of elements added during refinement on the faces uncheck the box to enable the field for Maximum Number of Surface Elements. With the box unchecked, the field, unavailable, the number in the box provides an estimate of mesh growth.

When the mesh is generated, the refinement criteria you specified will be used. This operation will be approximately the same as having slabs of tetrahedra, but it is not guaranteed to prevent tetrahedra from crossing slab interfaces. Caution should be used with this mesh operation, as very thin layers may cause a reduction in mesh quality or unnecessarily cause the generation of a very large mesh. Further regions refined under this operation and its close neighbors do not participate in solution adaptive refinement. This is another reason to use this seeding operation with caution.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

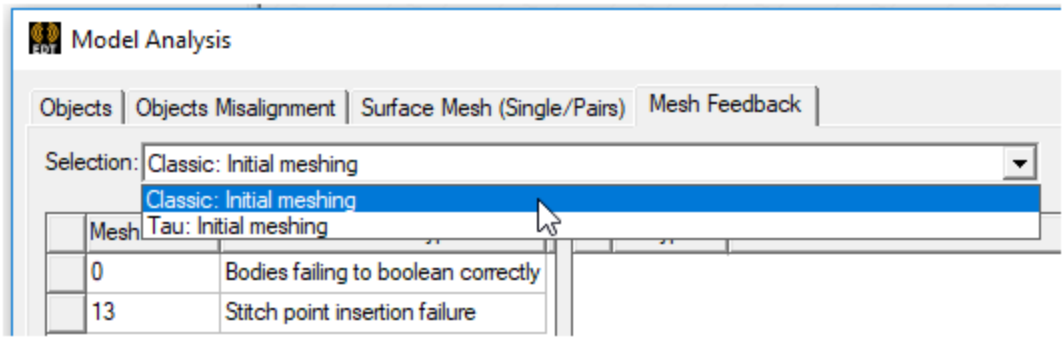
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visualization for Skin Depth Refinement Mesh Results

You can visualize the mesh to assess results either by using the clip-plane feature on a model with a mesh plot, or by using the Model Analysis dialog box as follows:

1. Select the object on which you have generated a meshplot, and click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback**.

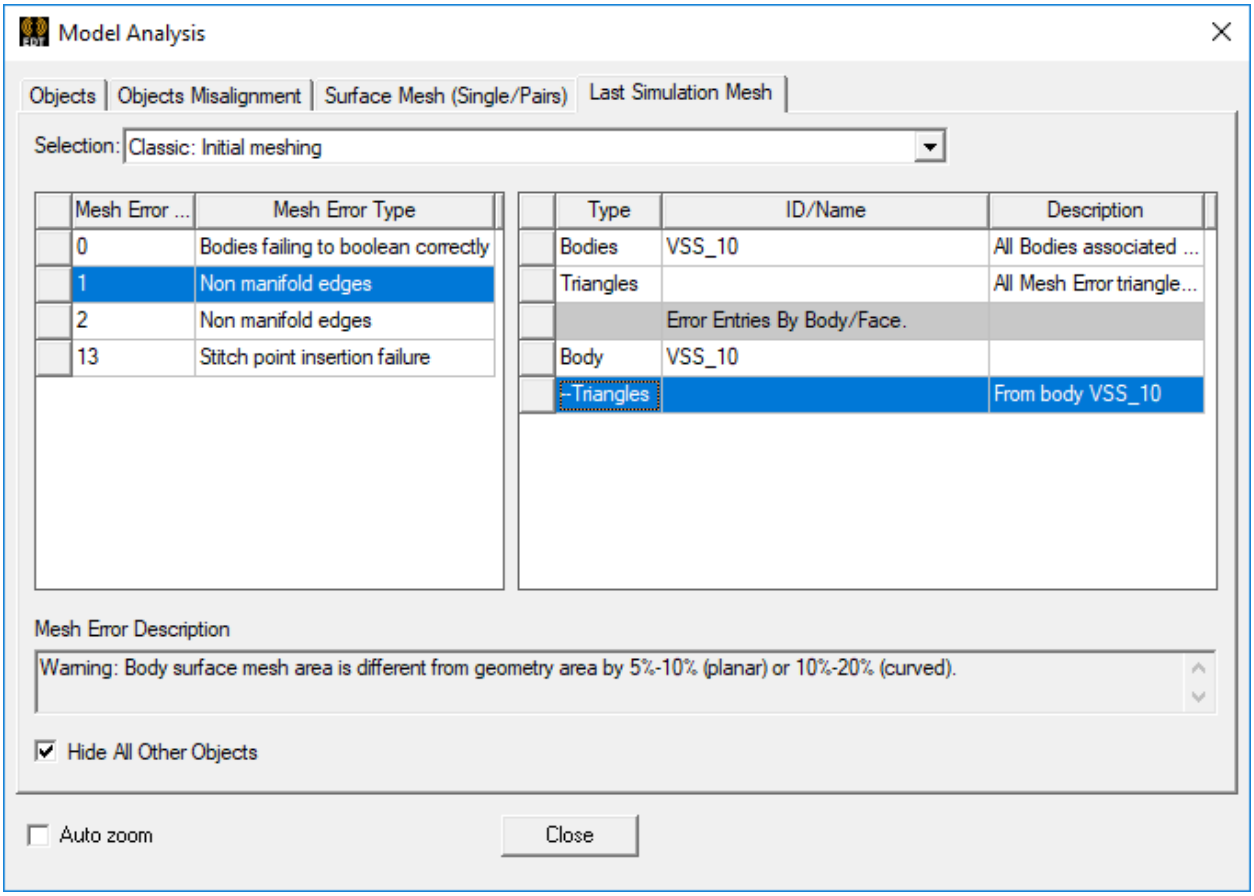
   The **Model Analysis** dialog box appears. If the project has solutions by different solvers, the **Selection** drop-down menu lets you select which mesh to view.



2. On the upper left table, under Mesh Error, select 0 through n , where 0 is the selected body, 1 is the layer closest to the face, 2 is the next, up to layer "n" as farthest.

   Selecting one of these rows causes a **Mesh Error Description** to display for the selected layer or body. The message gives the Total Expected count for triangles, as compared with the Success percentage by count and by area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. On the upper right table, select the body, face, or triangle desired, scrolling the list to find any object, face, triangle, or area of interest.

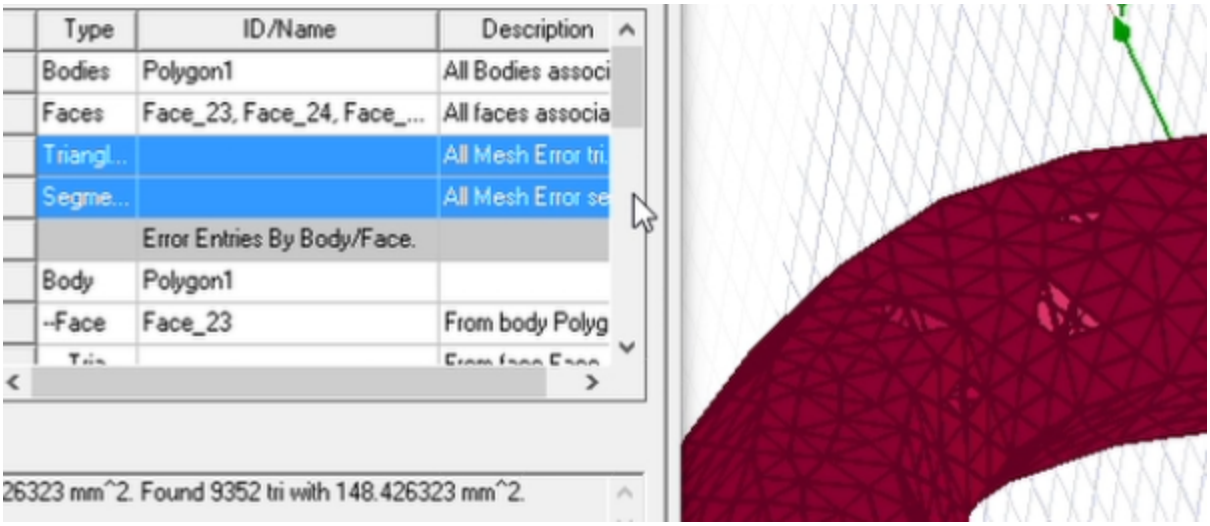
   Click Face, to see the selected face.

   Select Triangles to see the actual triangles created.

   Select Segment to show the triangles attempted.

   The following figure shows Triangle and Segment both selected (using Shift+click). Visually, the unfilled triangles represent places where the triangle is not on the desired layer. However the mesh is still very good and supports an accurate solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Check boxes allow you to **Hide All Other Objects** checked by default, and **Auto zoom to selection**, unchecked by default.

Modifying Surface Approximation Settings

Each solver applies surface approximation settings for selected objects when it generates the initial mesh. If you modify a solver's default settings after the initial mesh has been generated, they will not affect the mesh for that design variation. Surface approximation makes sense for curved surfaces, for which the mesh will not exactly reproduce the surface shape. It also can be used to restrict the aspect ratio of triangles on planar surfaces. For a given surface approximation slider setting, the approximation is more refined for IE and SBR+ solvers than for FEM solvers.

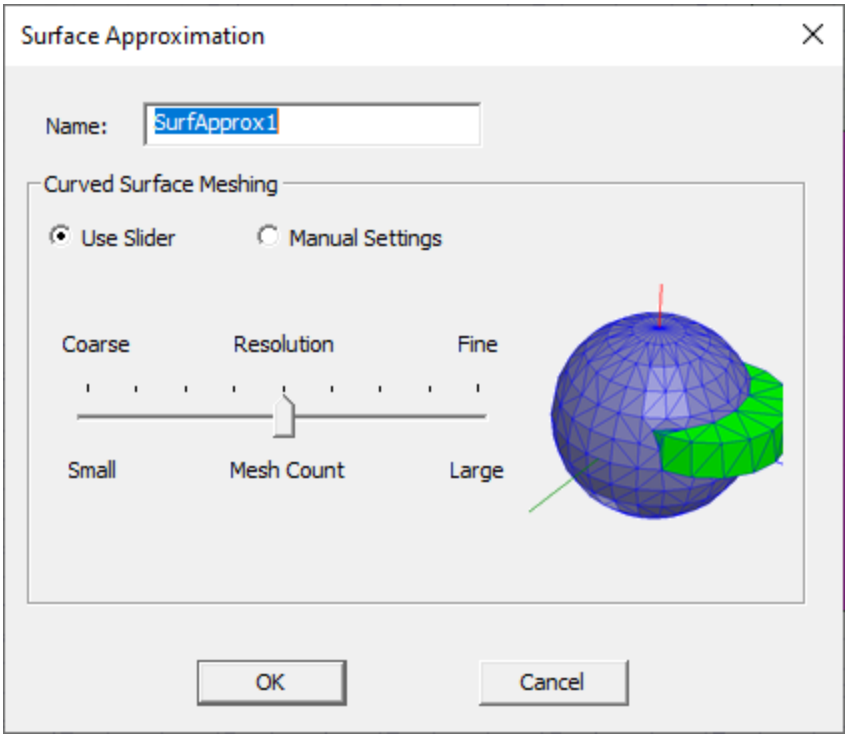
1. Select the faces for which you want to modify the surface approximation settings.

   Alternatively, select an object if you want to modify the surface approximation settings of every face on the object.

2. Click **Mechanical** > **Mesh** > **Assign Mesh Operation** > **Surface Approximation** or right click on **Mesh** in the Project Manager and select **Assign Mesh Operation** > **Surface Approximation**.

   The *Surface Approximation* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

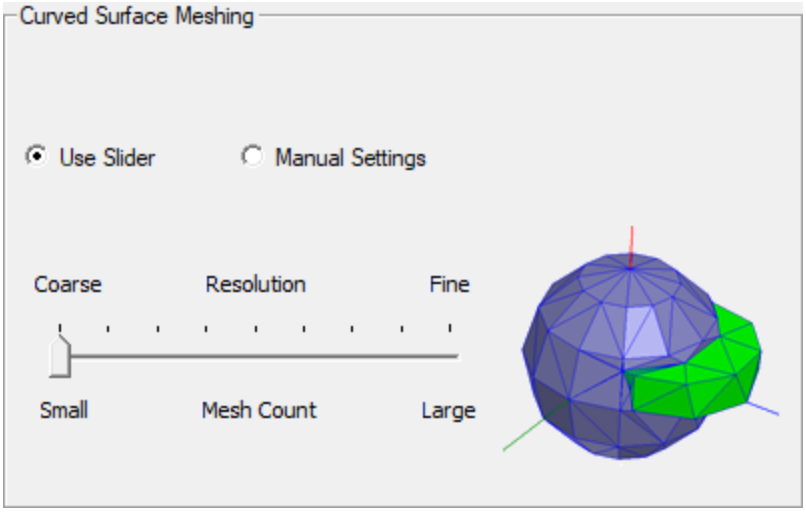


3. Type a name for the group of settings in the **Name** text box or accept the default name.
4. Under **Curved Surface Meshing**, you can select **Use Slider** or specify **Manual Settings**.

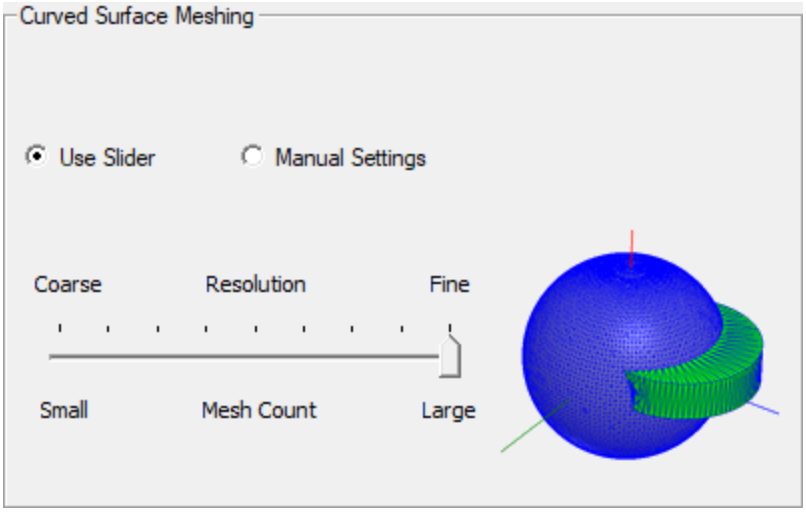
   The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*). Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

   Coarse Resolution, Small Mesh Count:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

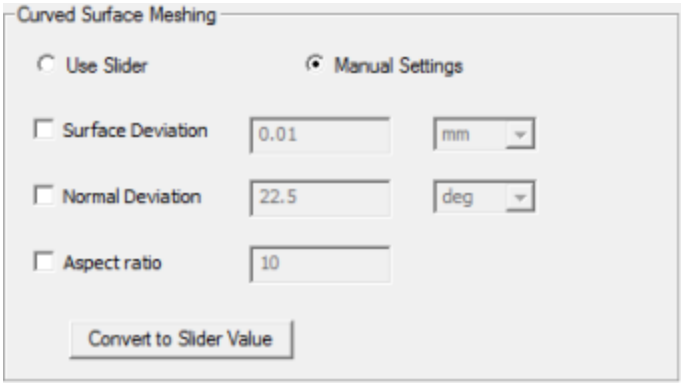


Fine Resolution, Large Mesh Count:



5. If you choose **Manual Settings**, the dialog box changes to show text fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> If you selected **Use dynamic surface resolution** in Initial Mesh Settings, use of **Manual Settings** is not permitted. You must use slider bar to specify one of three levels of surface representation: coarse (1-3), normal (4-6), and fine (7-9).

6. Select **Surface Deviation**. Use the drop-down menu to select a unit of measure and then enter the distance between the true surfaces of the selected faces and the meshed faces.
7. Select **Normal Deviation**. Use the drop-down menu to select a unit of measure and then enter the angular distance between the normal of the true surface and the corresponding mesh surface.
8. Select **Aspect Ratio** and then type a value in the text box. This value determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
9. Click **OK**.

   The settings are applied to the initial mesh generated on the selected surface. The group of settings is listed in the project tree under **Mesh**.

   You can also specify Initial Mesh Settings to apply to all objects. However, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Selecting **Use dynamic surface resolution** in the Initial Mesh Settings specifies the best-practice mesh operations over the geometric models. The default mesh operations or user-defined mesh operations may or may not be replaced by optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get smoother curvature representation while small curve faces, such as fillets and small curve objects, get relatively coarse triangulations to reduce overall mesh size.

This dynamic surface resolution mesh operation supports 3D volume mesh and surface mesh in all products.

Important:

The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

Applying Curvilinear Elements

For **HFSS**, **Maxwell**, and **Mechanical** designs, Curvilinear elements provide an accurate representation of the CAD model geometry by improving the conformity of element nodes to curved CAD surfaces. You can apply curvilinear elements as a Mesh Operation on selected curved model faces see procedure below or as a global option in the Initial Mesh Settings dialog box.

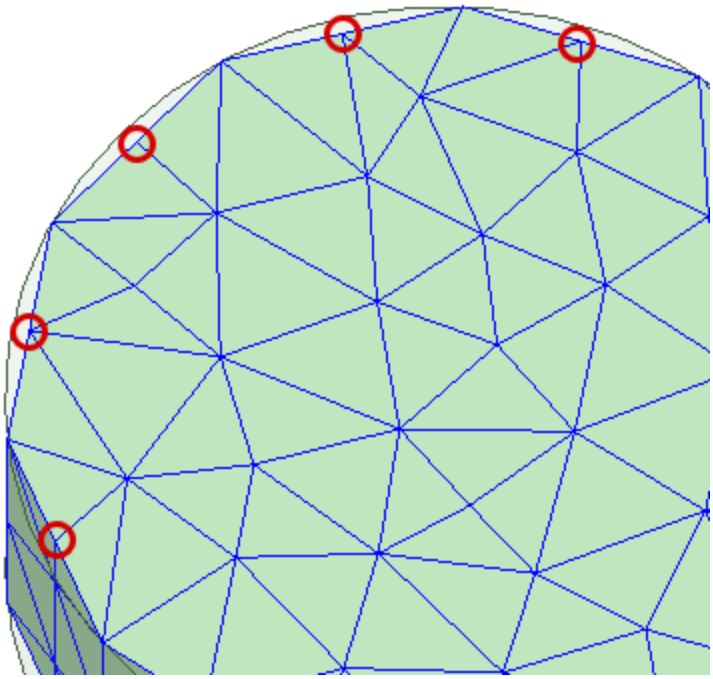
For **Mechanical** designs:

When curvilinear elements are applied globally or to specific CAD faces, the meshing process attempts to pull corner nodes of elements added during mesh refinement to the curved CAD faces. The following figures explain the effects with and without curvilinear elements applied:

- With the **Apply Curvilinear Elements** option *not* selected:

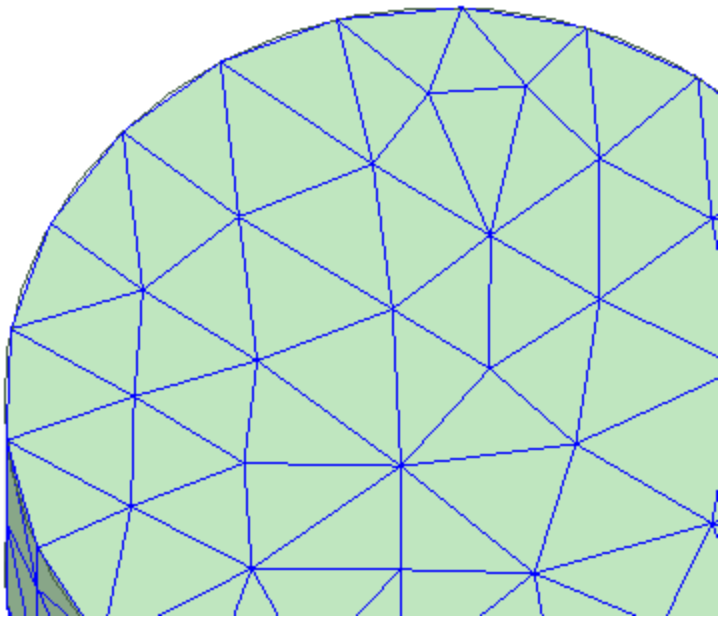
  Corner nodes of an initial coarse mesh are located on curved CAD surfaces. However, as the mesh is refined, the corner nodes of any additional elements remain along the plane of the initial mesh's element faces. Therefore, the number of facets along a curved face (in the circumferential direction) does not increase, and the facets remain coarse. This effect is shown in the first mesh plot below. Note the red-circled nodes. These nodes were added as a result of mesh refinement, and they were not pulled to the curved CAD face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- With the **Apply Curvilinear Elements** option *selected*:

  The meshing process will attempt to pull all corner nodes of additional elements added during mesh refinement to the curved CAD faces. Therefore, the facets along curved faces are finer, and the conformity of the mesh to the curved CAD faces is improved. The effect is as shown in the following, second mesh plot:

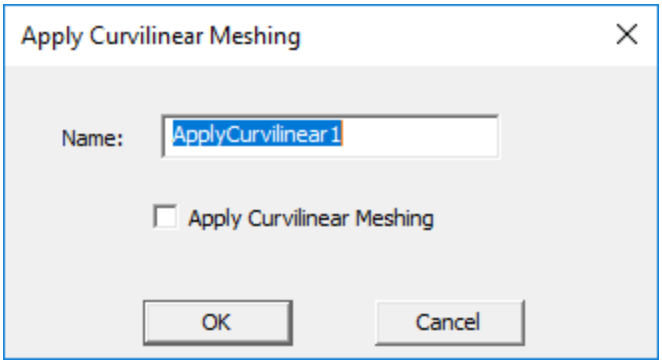


How to Apply Curvilinear Elements to Specific Curved Faces:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object or face to which you want to apply curvilinear elements.
2. Select **Mesh** in the Project Manager, right-click for the short cut menu and select **Assign Mesh Operation** > **Apply Curvilinear Meshing**.

   The **Apply Curvilinear Meshing** dialog box appears.



3. Either accept the default name or provide one of your own.
4. Select **Apply Curvilinear Meshing**.
5. Click **OK**.

The defined Mesh Operation appears in the Project Manager.

Assigning Cylindrical Gap Treatment

1. Select the object you want refined.
2. Click **Mechanical** > **Mesh** > **Assign Mesh Operation** > **Cylindrical Gap Treatment**

   A **Cylindrical Gap** mesh operation is applied to the selected object, and a **Cylindrical Gap** icon appears in the Project Manager's Mesh Operations folder.

   The Cylindrical Gap operation only works for the TAU (tolerant) mesher. It is a specific mesh operation assigned to the “Band” object for a rotational transient simulation.

   While you can select any object and assign the Cylindrical Gap operation to it in any Maxwell or Mechanical–Thermal simulation, the Cylindrical Gap operation is effective only for rotational simulations, for which cases the TAU (tolerant) mesher will be chosen as the default mesher; or for which you have manually selected the TAU (tolerant) mesher.

   The TAU (tolerant) mesher analyzes the “Band” object, calculates the surface normal angle for the band rotational surface, and automatically assigns the desired mesh operation; or adjusts the existing mesh operations to the adjacent true surfaces at both static and moving sides to ensure a smooth and uniform mesh for the “Band” mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Cylindrical Gap mesh operation is assigned to a regular object, TAU analysis finds the object is not suitable for transient mesh, in which case the regular TAU mesh will be applied. A Warning message is then displayed in the message window.

> **Note:**
>
> - There can be only one cylindrical gap operation per design.
> - The operation cannot be assigned to multiple objects.

3. If you wish to change the name of the mesh operation, you can double-click the **Cylindrical Gap** icon to open the **Cylindrical Gap Mesh Operation** dialog box in which you can type a name for the mesh operation in the **Name** box.
4. Optionally, for 3D transient simulations of rotational models only, if you wish to apply a clone mesh operation, check **Clone Mesh**. This option is unchecked by default.
    - When **Clone Mesh** is selected, the **Band Mapping Angle** setting is enabled. The valid range of angles is 0.1 through 3.0 degrees, inclusive. The default band mapping angle is 3 degrees. (For more information on this setting, refer to *Band Mapping Angle*.)

      > **Note:**
      >
      > If the model is not suitable to make a clone mesh or the clone mesh process fails, the program automatically falls back to the regular TAU 3D mesher. Because the clone mesh is generated in the initial mesh, the initial mesh is deleted and regenerated once this option is changed and you click **OK**.

    - When **Clone Mesh** is selected, the **Number of Layers** setting is enabled. You can input the number of mesh layers on the **Moving Side** and **Static Side**, respectively. The valid ranges for these settings are integers ≥ 1. The default values are 1. (For more information on this setting, refer to *Layers in Band Mesh*.)

3D Clone Mesh Generation

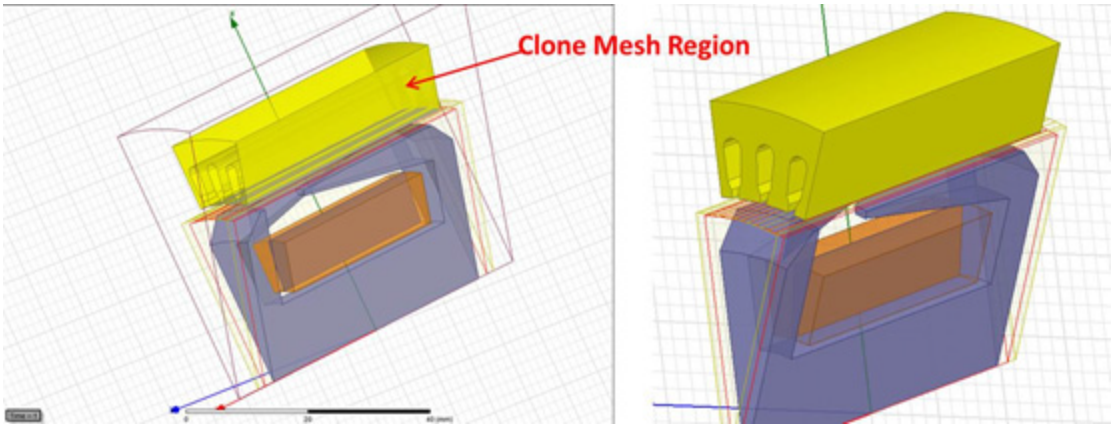
The 3D clone mesh is generated only when the following three conditions are met:

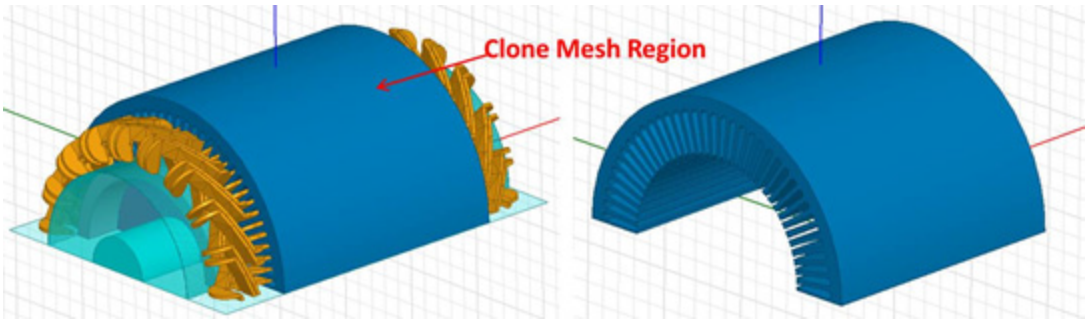
- The Clone Mesh check box is checked.
- The CylindricalGap mesh operation has been assigned.
- The model has a non-skewed stator with more than two teeth.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

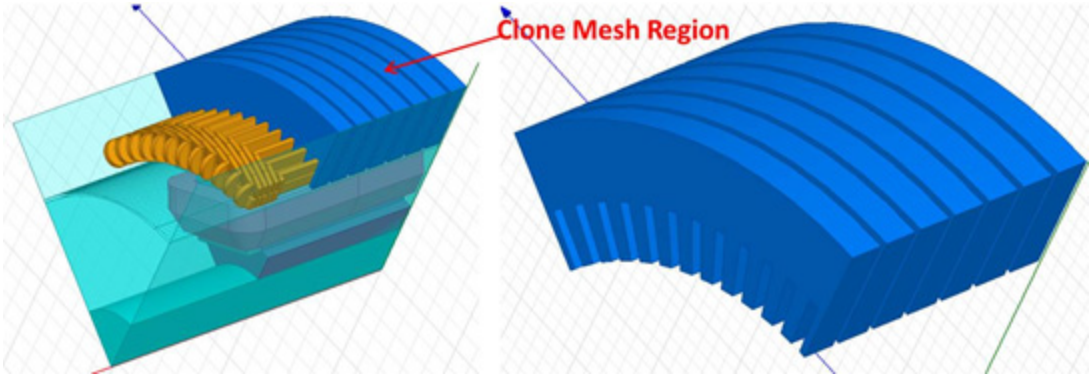
> **Note:**
>
> When Clone Mesh is used, the only additional mesh operation that can be used is surface approximation. If any other mesh operation is used, the clone mesh operation will be ignored.

The solid bodies adjacent to the band are detected to identify the clone object. If the object has more than two teeth that are close to the band, the teeth will be verified to see they have the identical geometric features. If the teeth are skewed, TAU will fall back to the regular mesh. Otherwise the region from the radius of teeth to the outer edge of the object will be further classified and verified to see if the object has the multiple 3D identical sub-regions.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

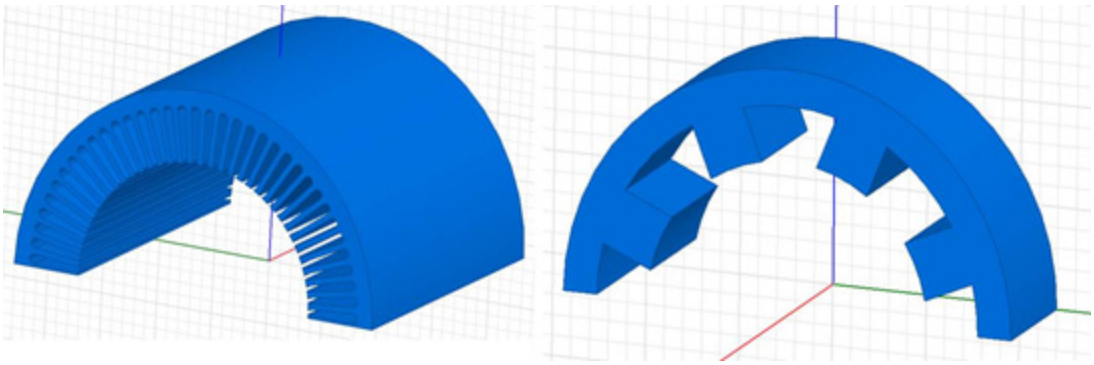


If a 3D clone region has been identified, TAU will classify the 3D rotational model into a static region and a moving region by the band object radius. The Static region includes the clone region and skewed coils regions at one side or two sides. Clone mesh method will then be used to generate the mesh in the clone region, while the TAU regular mesh method will be used to generate the mesh at other skewed regions.

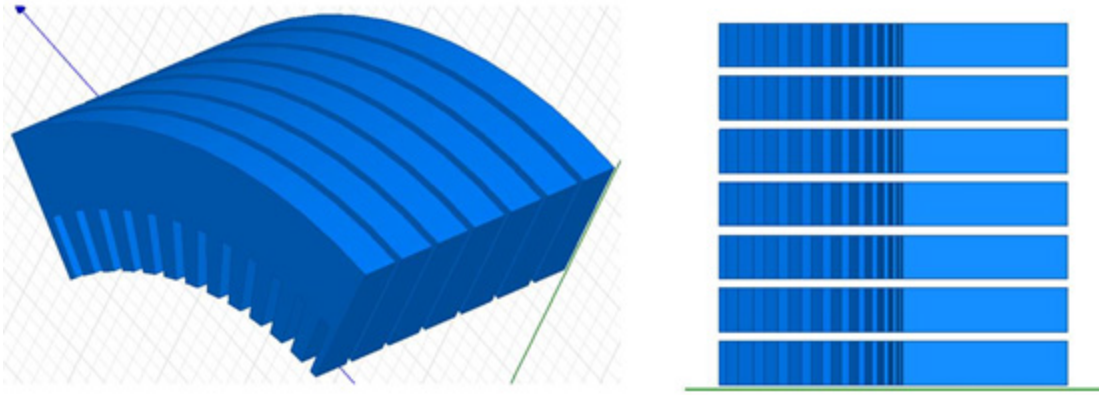
The moving region includes skewed coils regions in which the TAU regular mesh method is used to generate the mesh, and also the non-skewed sweep regions in which the TAU sweep mesh method is used to make the mesh.

3D Clone Mesh

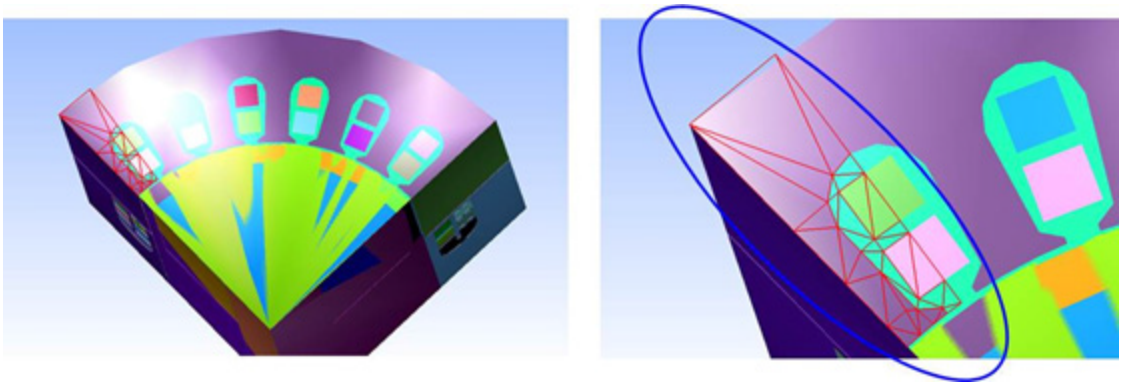
A 3D clone region may have only one simple object, or several sections of clone objects with the gap between as shown in the following images.

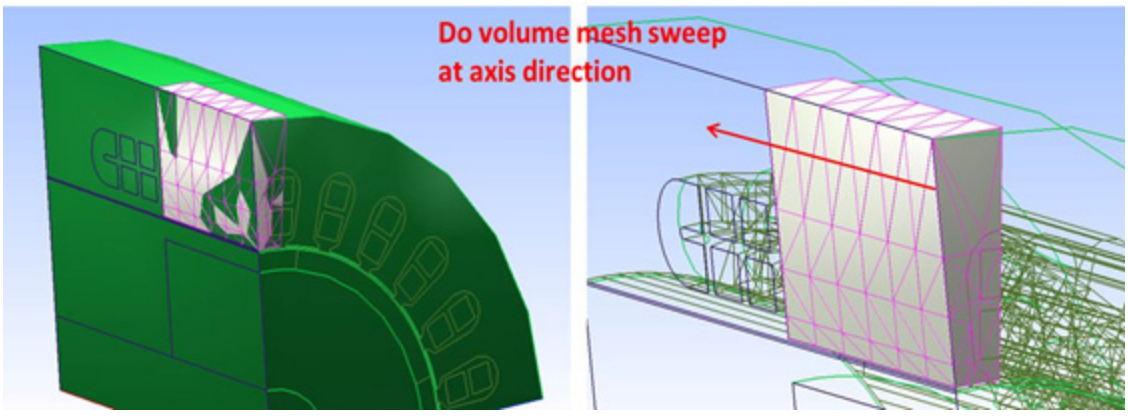


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To ensure the identical volume mesh in each sub-region, the volume mesh is generated in the Master region, then by using a copy-and-paste method to duplicate the mesh at all child regions. In the 3D master region, if the region is also symmetric, then a symmetric mesh is generated. “Sweeping mesh” technology is used to make the volume mesh in the master region. In the half of the master region, a surface mesh is generated at the axis side. The volume mesh is generated by sweeping the surface mesh in the axis direction to fulfill the clone master region.

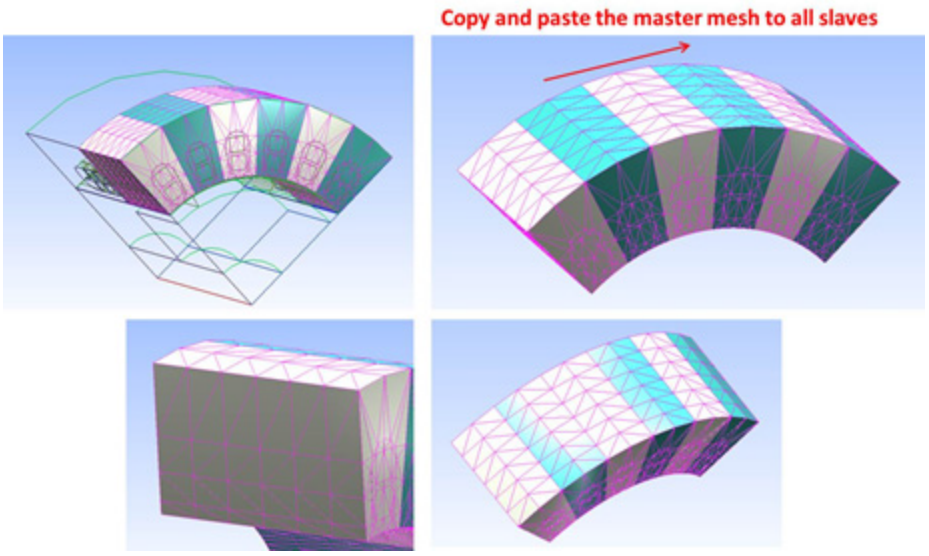




Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The volume mesh will be mirrored to the other half of the master region to produce the symmetric mesh in the master region.

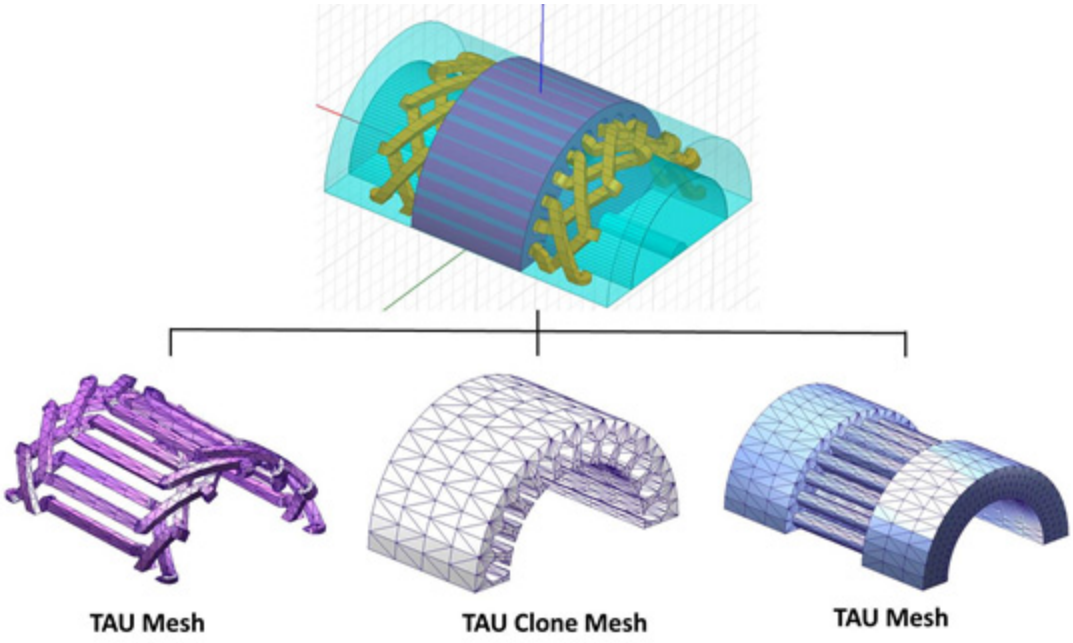
The volume mesh in the master region is then duplicated and translated to the child regions with copy and paste technology in TAU to complete the 3D clone region mesh.



3D TAU Mesh

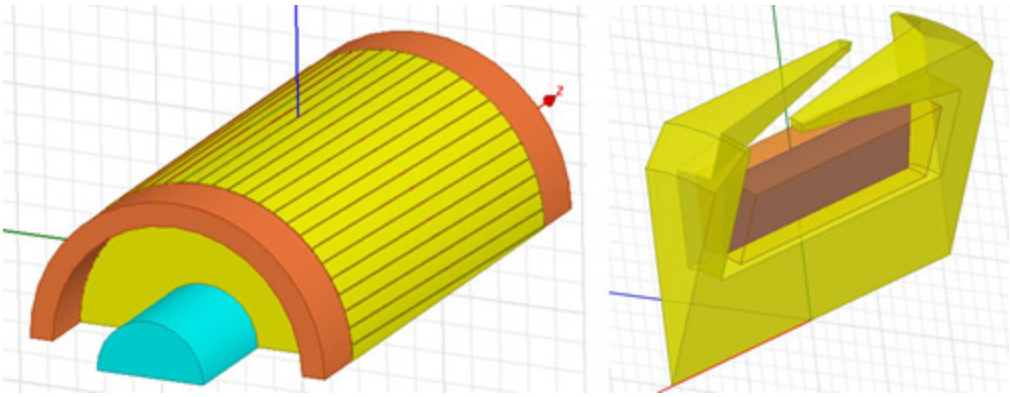
Beyond the 3D clone region, normally the regions have skewed coils. Therefore, the TAU regular mesher will be used to make the mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



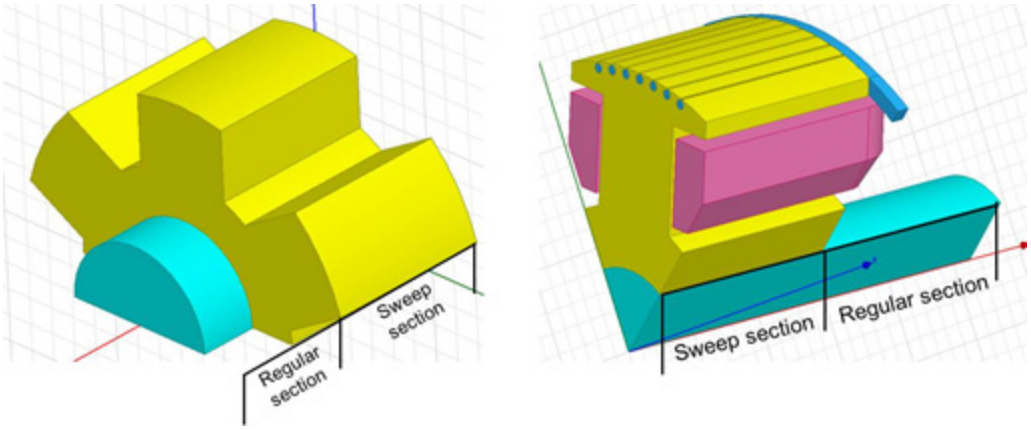
3D Sweep Mesh

The rotational model for a moving region may have different structures. It may have all skewed coils and rotors. In this case, the TAU regular mesh will be used to make the mesh.

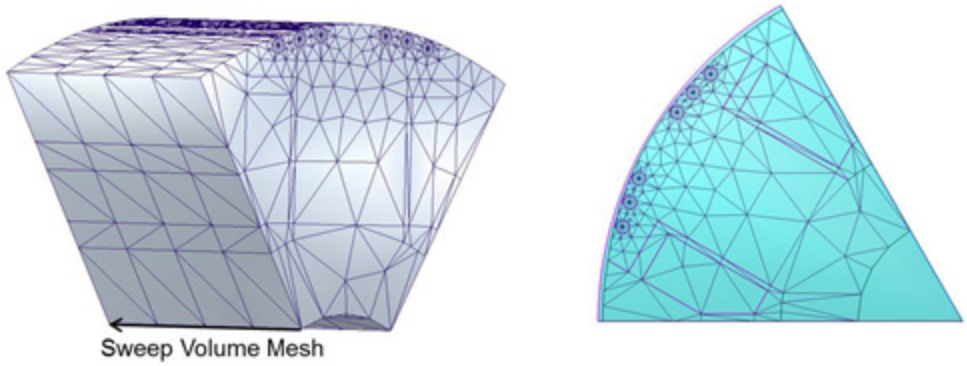


At the axis direction, the moving region can be classified into several specific regions. If the region has skewed objects, it is called a “regular section,” and the TAU regular mesh is applied. If the region can be geometrically sweeping at the axis direction, the region is called a “sweep region,” and the TAU sweep mesh method is applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

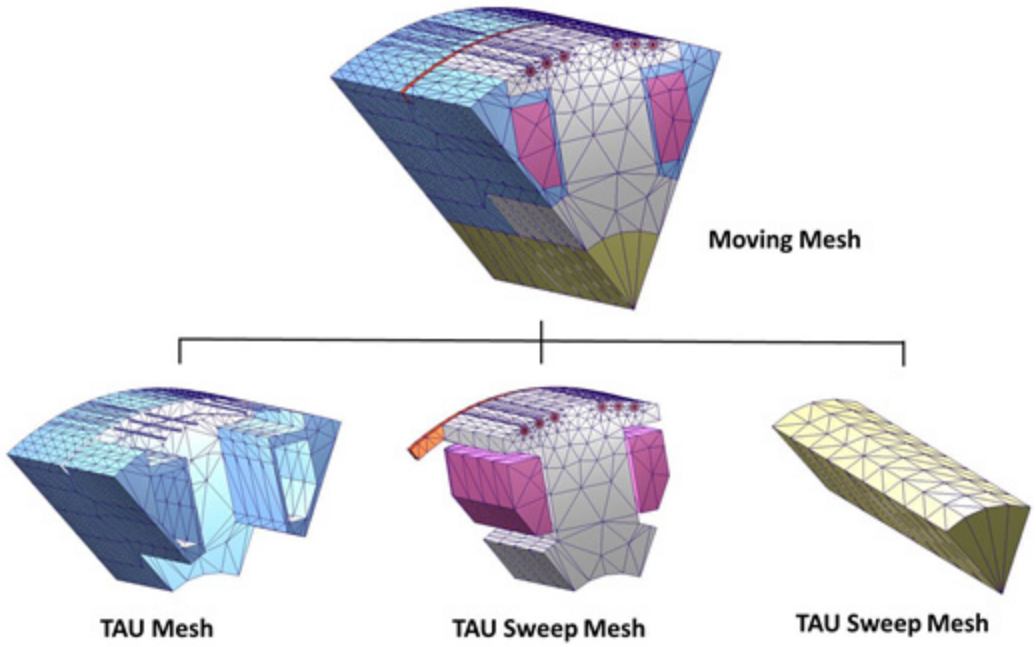


For the “sweep section,” a surface mesh is generated at the side, then the volume mesh is made by sweeping the surface mesh in the axis direction. This ensures a uniform mesh in the rotational direction, while maintaining the long aspect ratio in the axis direction to keep the mesh size small.



The volume mesh for the rest of the regions is made by the TAU regular mesher.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Band Mapping Angle

Because mesh mapping errors can occur at the interface between the band of motion and the stationary objects when mesh segmentation is performed, the band mapping angle allows these errors to be corrected by making the rotating angle per time step equal to the angle of each mesh segment in the direction of rotation. Thus, the band mesh will match the mesh on the stationary region for each time step. Reducing the mesh mapping error also reduces the noise for the torque calculation and improves simulation accuracy.

The **Band Mapping Angle** setting on the Cylindrical Gap Mesh Operation dialog box, enabled when **Clone Mesh** is selected, supports 3D transient designs for which:

- The band of motion is a true surface.
- The model is not skewed.
- The model is suitable for clone mesh – that is, the model can be segmented into identical sections in the direction of rotation.

The recommended band mapping angle (the angle the rotor rotates in one time step) typically equals the rotational speed multiplied by the time step. For example, if the rotational speed is 1.5 degrees per second and the time step is 2 seconds, then the band mapping angle should be 1.5 x 2 = 3 deg.

The starting position of the simulation should also be considered. In the above example, if the starting position is 0 deg or 3 deg, then 3 deg would be a good band mapping angle. If the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

starting position is 1.5 deg, the mesh will not be mapped at the starting position, in which case 1.5 deg is the recommended setting.

Another approach for determining the band mapping angle is to start with the model. For example, assume that the stator has 90 slots in the full model, with the solution domain angle equaling 1/6 of the full 360 degree model or 15 slots for 60 mechanical degrees. If you would like to solve with each slot having 8 mesh segments, there would be 120 total mesh segments (15 slots x 8 segments per slot). In this case, the band mapping angle would be 60 (deg) /120 = 0.5 deg. To ensure rotation through one mesh segment per time step, assuming the rotational speed is 1200 rpm, the time step should be set as:

$$Timestep = \frac{1}{(speed_{rpm}/60seconds) \times TotalSegmentNumber \times (360^\circ / DomainAngle^\circ)}$$

$$Timestep = \frac{1}{(1200/60) \times 120 \times (360/60)} = \frac{1}{14400} seconds$$

> **Note:**
>
> For band mapping angles less than 0.5 degrees, clone meshing may improve with the **Allow 3D Small Band Mapping Angle** beta feature. This feature is accessed from **Tools** > **Options** > **General Options** > **Beta Options** button, which opens the following window:

Clone Mesh Density

For Maxwell 3D rotational transient designs and Mechanical–Thermal designs of rotating machines, this meshing operation allows you to specify the clone mesh density while keeping the identical mesh in identical sub-regions. Different from Length-Based Mesh Refinement, this Clone Mesh Density mesh operation will only be applied to clone mesh. Only when clone mesh is enabled under cylindrical mesh operation, mesh will respect this mesh operation. Mesh density can be controlled while identical mesh is maintained.

To control the clone mesh density:

1. Select the object.
2. Click **Mechanical** > **Mesh Operations** > **Assign** > **Inside Selection** > **Clone Mesh Density**.

   The **Clone Mesh Density** dialog box appears.
3. Type a name for the mesh operation in the **Name** box, or accept the default name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. To specify the radial direction density, enable the radial direction option. Type the maximum length of the element size required inside the object(s) in the **Maximum element length** box, and select the units. Maxwell refines the element edges inside the object(s) until they are equal to or less than this value. The default value is set to 10% of the maximum dimension of the bounding box of the model.
5. To specify the axial direction density, enable the axial direction option. Enter the **Number of layers** to be added. Maxwell refines the mesh in axial direction inside the object(s) until the number of mesh layers in the axial direction are equal to or more than this value.
6. Click **OK**. You need to specify mesh density in at least one direction to enable this mesh operation.

When the clone mesh is generated, the refinement criteria you specified are used. You can specify different maximum length of the element size on different objects. Note that identical density setting should be applied to identical objects to ensure the specified density to be applied on those objects.

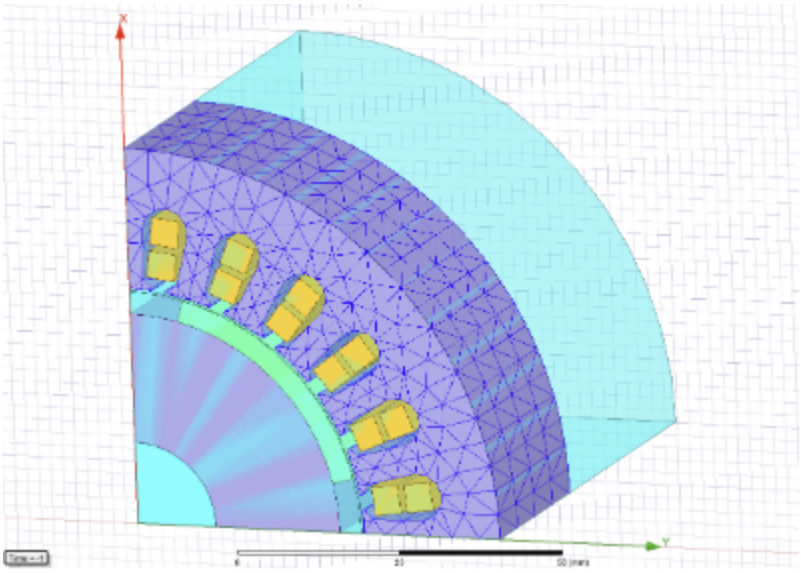
You can also specify different axial direction number of layers on different objects. Note that the number of layers in axial direction are maintained separately inside and outside of the band. If objects share a same region in axial direction on the same side of the band, the mesh refinement will refines until the layer thickness equal or smaller than the smallest average layer thickness in this region.

The Clone Mesh Density can not be applied to band object. To control the element size on band, use the **Band Mapping Angle** setting.

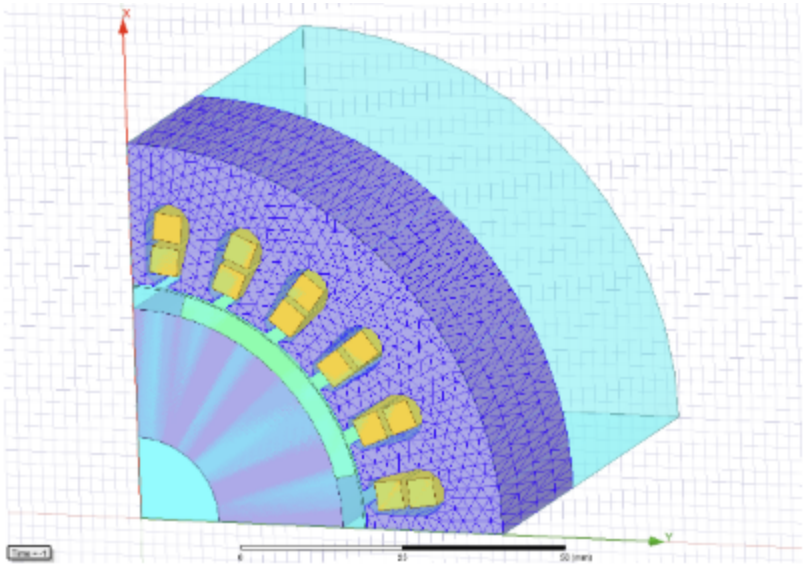
Clone Mesh Density can only be respected in clone mesh region. Mesh density in regions that are skewed or not clonable may not be controlled as precise as it is in clone region.

Original clone mesh without Clone Mesh Density mesh operation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

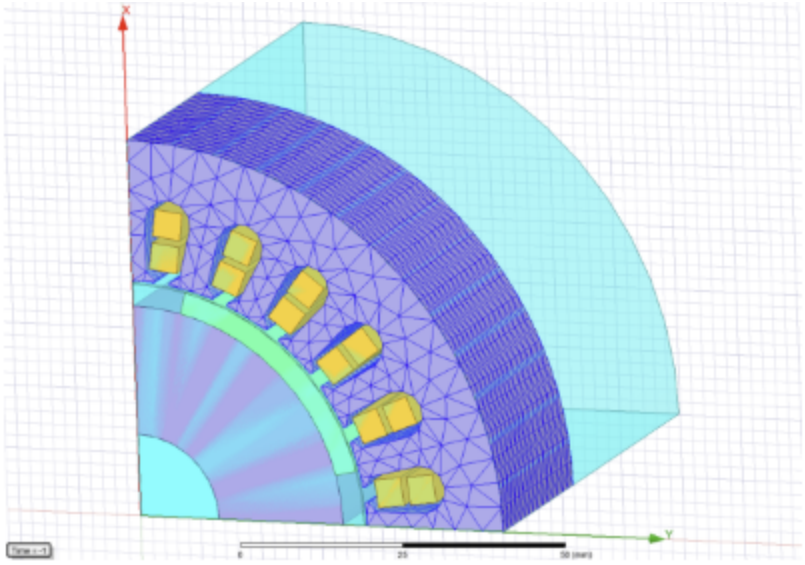


Clone mesh with Clone Mesh Density mesh operation on Stator. Radial direction mesh is refined according to maximum element length specification:



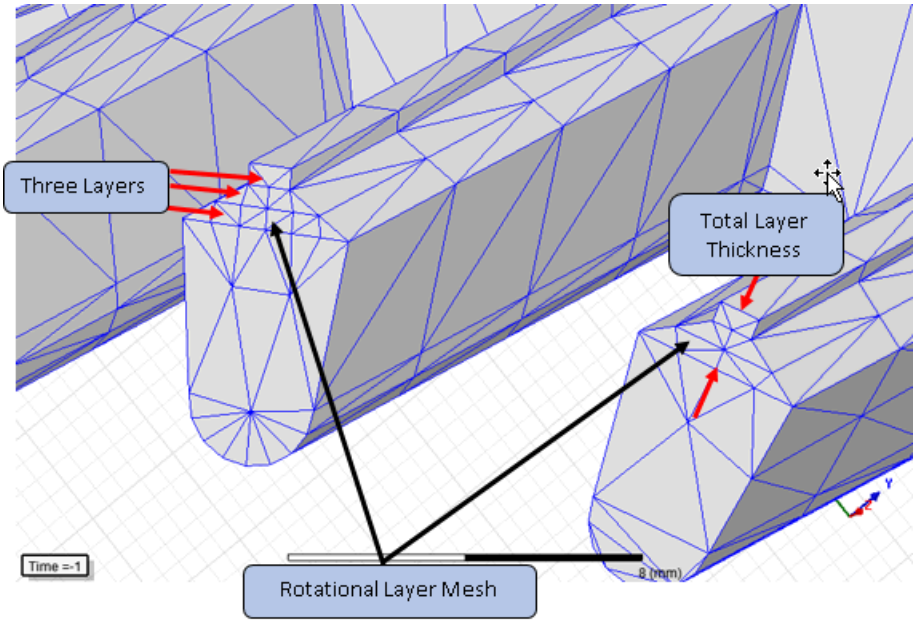
Clone mesh with Clone Mesh Density mesh operation on Stator . Axial direction mesh is refined according to number of layers specification:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rotational Layer Mesh Generation

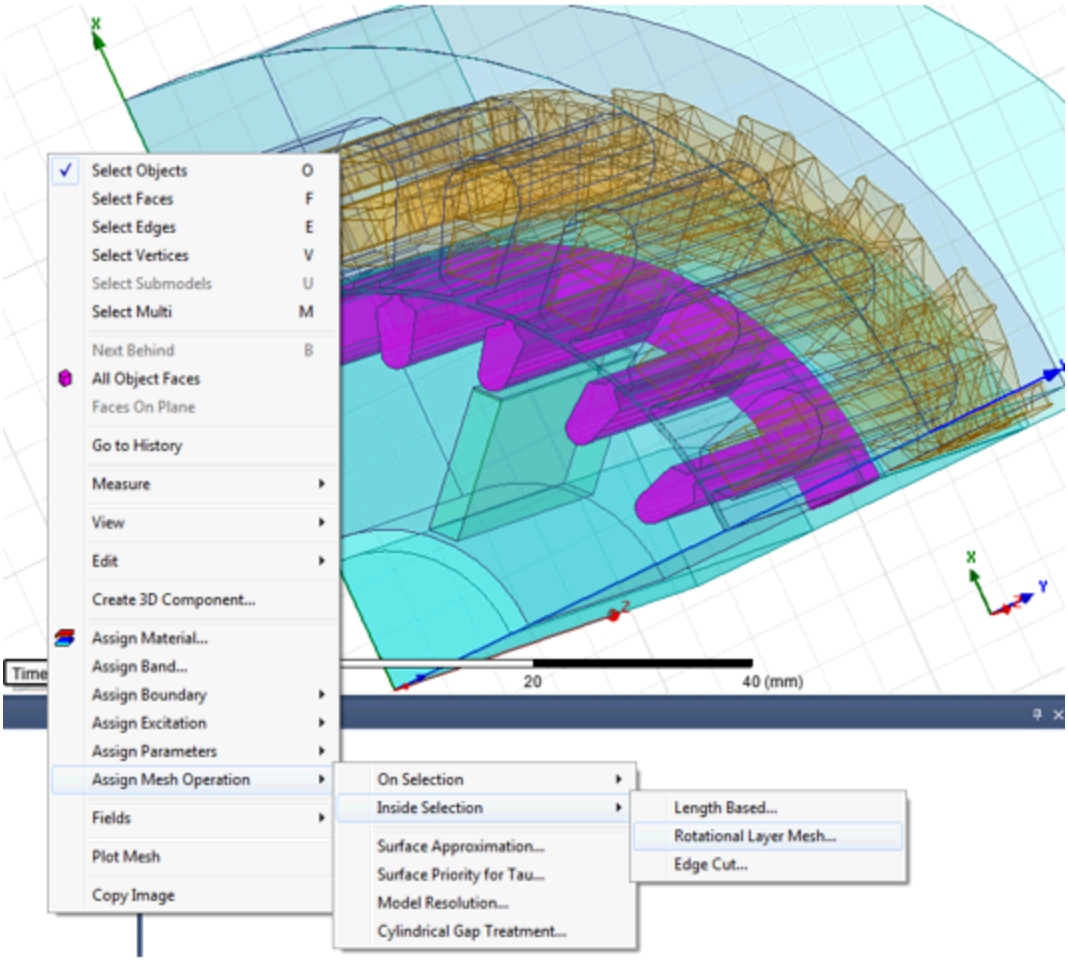
Rotational Layer Mesh creates up to three layers in the radial direction starting from the faces most adjacent to the “Band”. You can define the **Number of Layers** (up to three) and **Total Layer Thickness**.



To apply Rotational Layer Mesh:

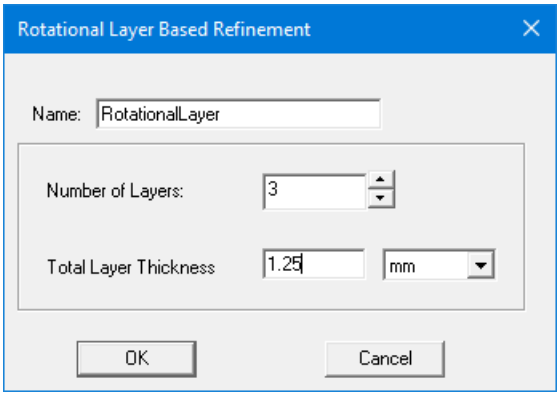
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the desired bodies near the band.
2. Right-click to open the context menu, and select **Assign Mesh Operation** > **Inside Selection** > **Rotational Layer Mesh**.



3. In the **Rotational Layer Based Refinement** dialog box, you can rename the mesh operation, specify the number of layers (maximum of 3), and the total thickness of all the layers combined.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK**.

Specifying Surface Priority for TAU Mesher

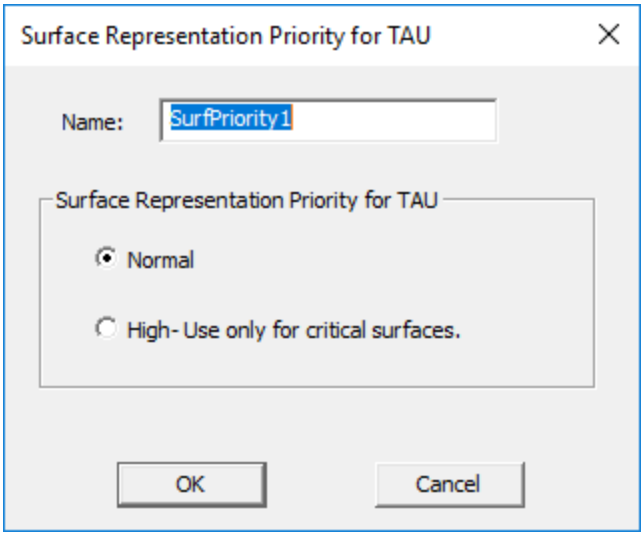
You can choose to specify the surface representation priority for the TAU mesher. For most designs, you can let the solver automatically choose which of two meshing approaches to take. The solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics. In most cases, the solver uses TAU mesh, rather than the classic mesh. In general, it looks for specific features (for example, stacks of large planar parallel facets with small gaps) and situations where the initial TAU mesh is 4 times larger than the Classic. Experienced users may choose to designate a high surface representation priority for the critical surfaces with the TAU mesh is used.

> **Important:**
>
> Use this mesh operation with caution. Use this option only if the model has tiny elements or critical boundary conditions since the solver performs several additional strict mesh verification steps on the selected objects.

1. Select the surface of interest.
2. If desired, specify a Name for the Mesh operation.
3. Right-click the **Mesh** icon in the Project tree to view the shortcut menu and select **Assign Mesh Operation** > **Surface Priority for Tau**.

   The **Surface Representation Priority for Tau** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Select **High** for the critical surface.
5. Click **OK**.

   The Mesh operation appears in the Project Manager.

Specifying the Model Resolution

The model resolution parameter is used by the meshmaker to distinguish large features from small features in the model. This setting controls how large a feature must be to be resolved by the meshmaker. For example, if you set the model resolution length to 20 mm, any model features smaller than 20 mm are not represented in the mesh. Neither the model nor the model files are changed. The resolution only controls how the mesh for the model is represented.

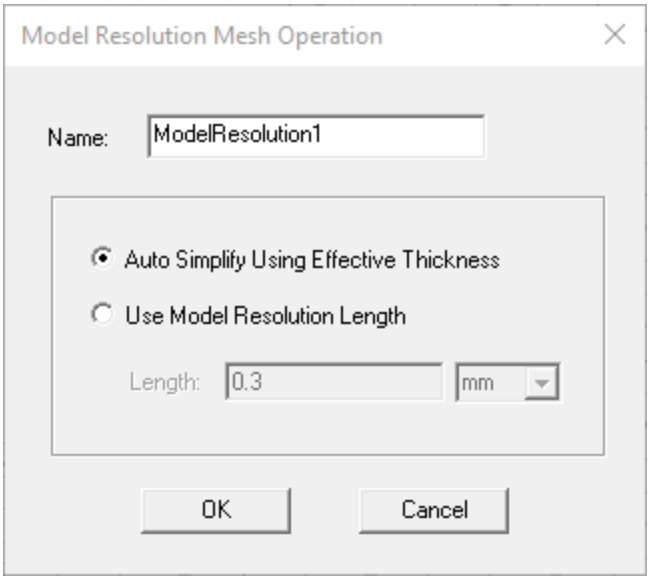
While removing small details, if the meshmaker finds that the representation of the model is not accurate enough, it returns an error condition. The meshmaker then starts with the most accurate representation and prunes away the details smaller than the model resolution length. It returns an error if the specified model resolution length forces the final representation to deviate too greatly from the model. You can set Model Resolution on one or more objects to remove unnecessary details from the mesh representation. This can be used to reduce the mesh complexity of the selected objects.

> **Important:**
>
> A cautionary note is needed concerning the use of model resolution. It can sometimes make model faces fail to be represented in the mesh, which can effectively remove some of your boundary conditions. This can only happen if faces are closer together than the model resolution distance you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object or objects on which to specify a Model Resolution operation.
2. Click on **Mechanical** > **Mesh** > **Assign Mesh Operation** > **Model Resolution** or right click on the **Mesh** icon in the **Project** tree and select **Assign Mesh Operation** > **Model Resolution**.

   The **Model Resolution Mesh Operation** dialog box displays.



Alternatively, you can display the same dialog box if you:

a. Right-click either **Mesh** in the Project Tree, or right-click in the **3D Modeler** window to display the respective shortcut menu.
b. Click on **Assign > Model Resolution** in the **Project Tree** menu or click on **Assign Mesh Operation > Model Resolution** on the shortcut menu.

The **Model Resolution Mesh Operation** dialog box contains text fields for the mesh operation *Name* and radio buttons with choices for the following:

- **Auto Simplify Using Effective Thickness**

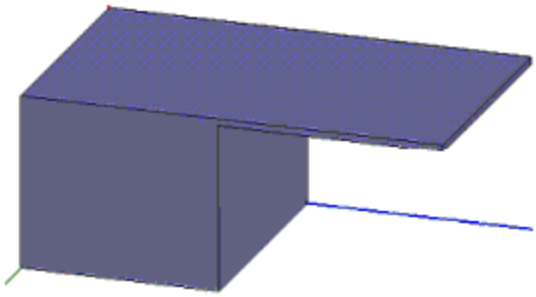
  The mesher calculates the resolution length based on each object's effective thickness. One mesh operation can be assigned to many objects, and each will be simplified based on its own dimensions. Use the **Auto Simplify** selection:

- To remove many details while retaining an object's overall shape and size.
- For objects of generally uniform thickness.
- To assign one mesh operation to many objects.
- **Use Model Resolution length**

This enables fields for you to specify the resolution value and units. Use this selection for:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Tighter control of mesh accuracy.
- Objects of non-uniform thickness. For example, the thin section of the object shown below might be lost with **Auto Simplify**:



3. After defining the operation, click **OK**.

   This adds the named Model Resolution operation under the **Mesh** icon in the Project Manager.

> **Note:**
>
> Setting Model Resolution will invalidate any existing solutions.
>
> When two objects in contact have different model resolution lengths, the smaller length will apply for the common regions.

Specifying a Source for the Initial Mesh

You may choose to specify a source for the initial mesh from either the current design or another design. The source mesh should represent a geometrically equivalent model.

> **Note:**
>
> For more information, see Mesh Linking.

To specify a source for the initial mesh:

1. Under the **Advanced** tab of the **Solution Setup** dialog box, click the check box for **Import Mesh**.

   This displays the *Setup Link* dialog box. By default, the **Source Project** check boxes for **Use This Project** and the **Source Design** check box for **Use This Design** are selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When you setup a link to an external source, the **Lamda refinement** option is deselected under the **Options** tab to avoid over-refinement of the linked mesh.

If you uncheck **Use This Project**, you enable fields for selecting a different project and radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.

2. To specify a Source Project file click the ellipsis [...] button to open a file browser window.

   When you selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

   Use the radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.

   When you select a Project File, the Design field and the Solution field are filled in with default values, and the drop down menus contain any available Projects and solutions.

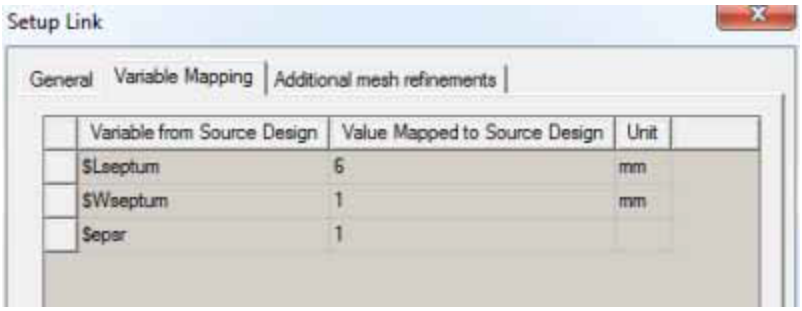
   > **Note:**
   >
   > The solution in the source design must provide data for the target design's adaptive frequency as well as its sweeps. That is, the adaptive frequency for the target design must be included in the sweep in the source design.
   >
   > If necessary, you can open the source design and add an appropriate frequency point to an existing sweep.

3. For the Source Design, if you leave **Use This Project** checked, you can uncheck **Use This Design** to enable the a drop-down menu for to select from other available designs.
4. For the **Source Solution** field, you can use the drop-down menu to select from other possible solutions.
5. Use the check box specify whether to **Simulate the source design as needed** to produce the mesh.
6. Use the check box to specify whether to **preserve the source design solution**. Note that in the **Extractor mode**, the source project will be saved upon exit. **Extractor mode** means that the software is opened during the link solely for the purpose of solving.
7. The second tab, **Variable Mapping**, lets you view any variables contained in the Project you select.

   When there are variables in the source design, you can choose to "map" these variables to constant values, expressions or variables in the target designs. Variable mapping

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

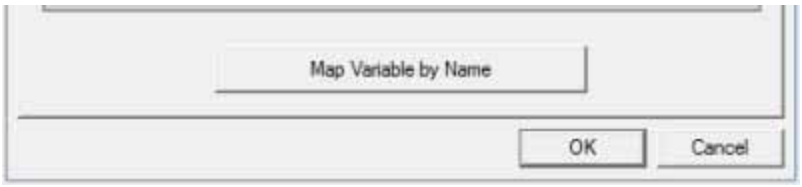
becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.



When a variable in the target design has the same name of a source design variable but the 2 variables are not mapped, the **Parameter** tab will become the active tab with the following message box popped.

You can exit the **Setup Link** dialog directly by selecting "Accept Setup and Exit Dialog" or reexamine the parameter mapping by selecting "Return to Setup Dialog".

In the **Setup Link** dialog box, for linked designs with variables of the same name, you can click **Map Variables by Name** to automatically map same named variables. A variable that does not have a same named counterpart, and retains its value in the source design.



Click **OK** to proceed without the variable mapping.

8. You can select the **Additional mesh refinements** tab to specify the following:
   - For Mesh Operations, you can select radio buttons to either **Apply mesh operation in target design on the imported mesh**, or **Ignore mesh operations in target design** (default).
   - For *Port Adapt*, you can check **Perform port adapt in target solve setup**.
9. Click **OK** to accept the setup and close the **Setup** dialog box.

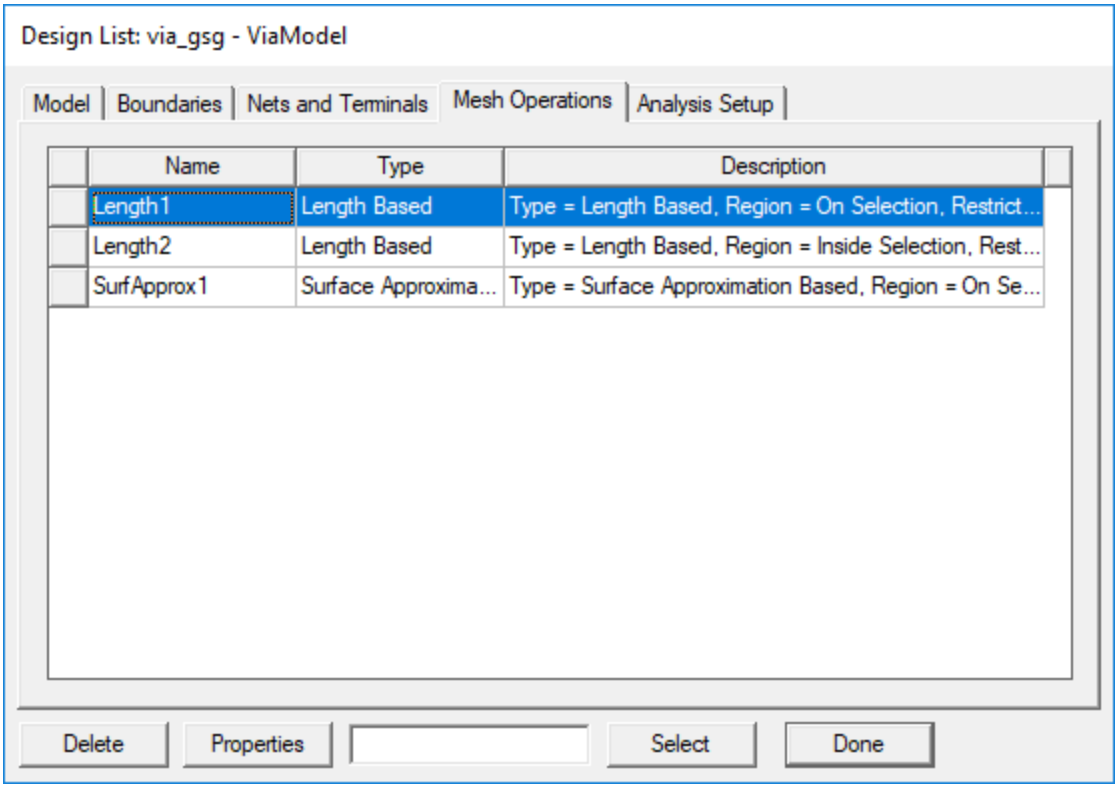
Viewing List of Mesh Operations

To list all mesh operations for the project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Tree, right-click **Mesh** and select **List**.

   The **Design List** window appears, on the **Mesh Operations** tab.



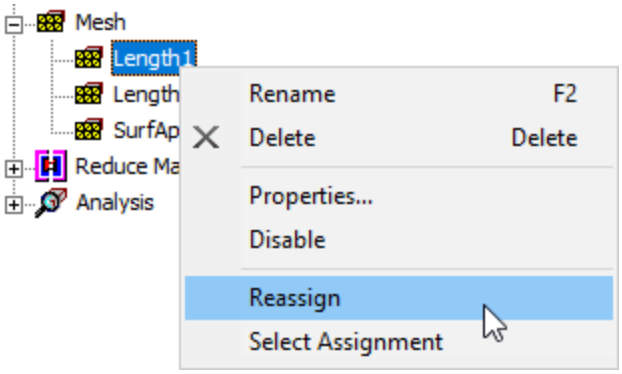
2. Click **Done** to close the window.

Reassigning a Mesh Operation

To reassign a previously assigned mesh setup:

1. Select the object(s) to which you want to reassign the mesh operation.
2. In the Project Tree, under **Mesh**, right-click the mesh refinement you had previously defined and select **Reassign**.

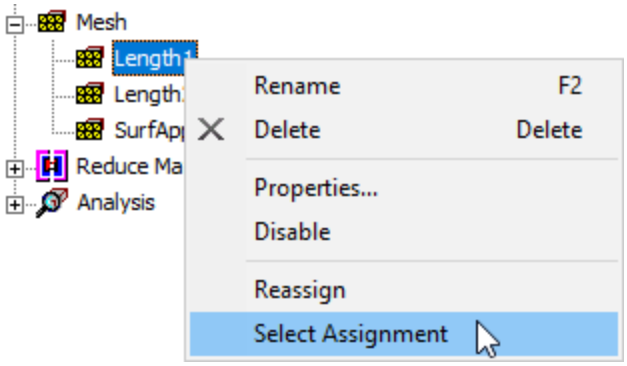
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The mesh operation is reassigned to the currently selected objects.

To view the objects to which a mesh operation is assigned:

1. In the Project Tree, select the mesh operation.
2. Right-click the previously completed mesh refinement and select **Select Assignment**.



The objects to which the operation is assigned are selected in the modeling workspace.

Reverting to the Initial Mesh

The initial mesh is the mesh that is generated the first time a design variation is solved. It includes surface approximation settings, but does not include lambda refinement or defined mesh operations.

If you have modified the design setup, and do not want to use the existing current mesh, revert to the initial mesh prior to solving.

- On the **Mechanical** > **Analysis Setup**> **Revert to Initial Mesh**.

Reverting to the initial mesh is useful when you want to evaluate how a different solution frequency affects the mesh generated during an adaptive analysis. You lose all solution data for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

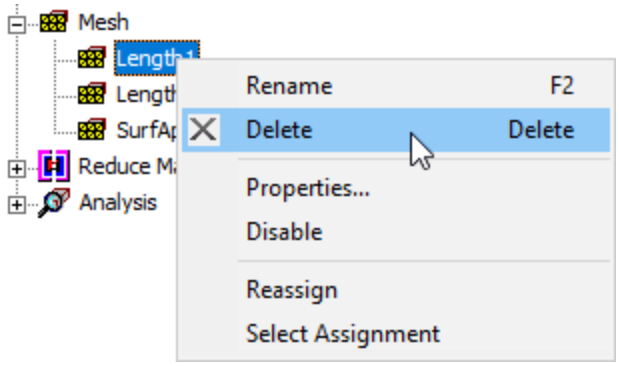
a solve setup and all of its sweeps when you revert to the initial mesh for that setup. You can do this for all solve setups at once by selecting the command through Analysis in the menu system or project tree, or for a specific solve setup via its right mouse click menu in the project tree.

Deleting Previously Assigned Mesh Refinements

You can delete an individual mesh refinement, or delete all previously assigned mesh refinements.

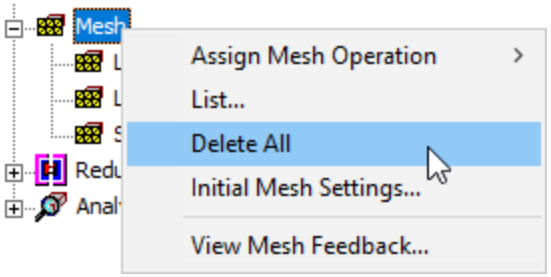
To delete a single refinement:

- In the Project Manager, right-click the mesh refinement and click **Delete**, or press the **Delete** key.



To delete all previously assigned mesh refinement data:

- Click **Mechanical** > **Mesh** > **Delete All**.
- In the Project Manager, right-click **Mesh** and select **Delete All**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

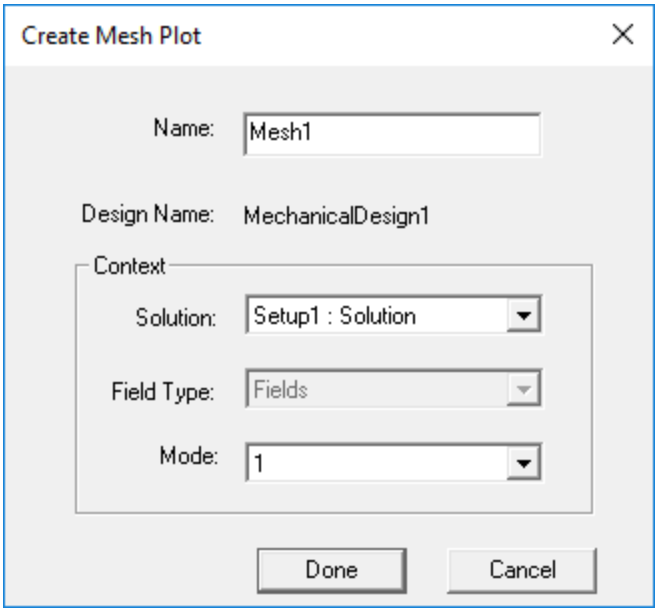
> **Note:**
>
> This will not alter any existing meshes that have already made use of one or more mesh refinement operations. However, deleted mesh operations will not be used again after reverting to the initial mesh.

Plotting the Mesh

Before or after the solution is complete, you can plot the finite element mesh on surfaces or within 3D objects. For an HFSS or Maxwell design that includes Layout Components, you can also select nets and layers from the layout components to plot.

1. Select one or more surfaces or objects to create the mesh plot on or within. If the design is HFSS and includes Layout Components, you can select the surfaces either before or after selecting **Plot Mesh**.
2. Select **Plot Mesh** in one of two ways:
   - Click **Mechanical** > **Fields**> **Plot Mesh**.
   - In the Project Manager, right-click **Field Overlays** and select **Plot Mesh...**

   The *Create Mesh Plot* dialog box appears.

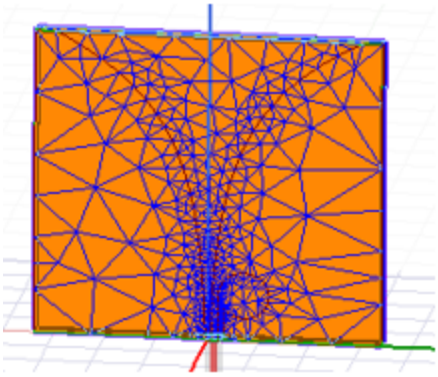


3. Select the **Solution** to use from the drop-down menu.
4. If **Field Type** is enabled, you can select from the drop-down menu for a mesh plot.

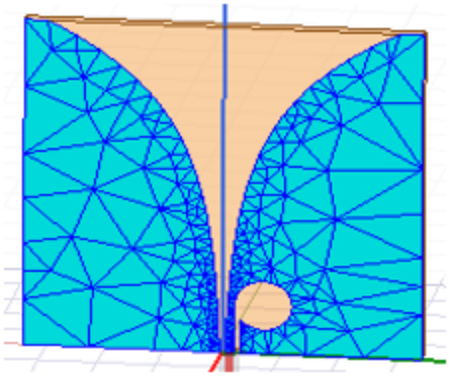
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Done**.

   The mesh appears on the surface or object you selected. For example:

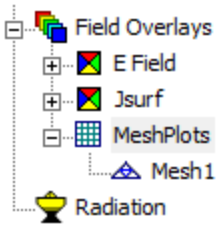


Mesh Plot of All Objects



Mesh Plot of Selected Surfaces

   An icon for the mesh also appears in the Project Manager under **Field Overlays** > **Mesh Plots**.



If a solution is ongoing, you can select the **Mesh Plots** icon in the in the Project Tree, right-click to display the shortcut menu, and select **Update Plots**. This updates the mesh plot to include the latest data available. After the last adaptive pass, the mesh plot is automatically updated.

If a mesh includes seeding, these effects appear at the start of the adaptive passes. Any mesh adaptation at the start of a subsequent pass is not plotted until after that pass is completed. This delay ensures that mesh plots and actual solutions remain consistent with each other.

You can modify an existing plot by selecting the plot and changing the properties.

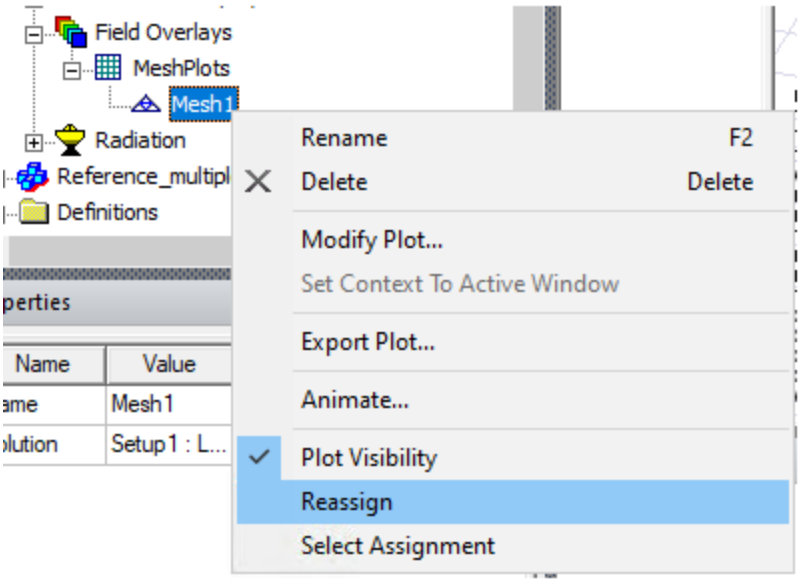
If you modify the geometry of an object, the solution is invalidated and the mesh plot is removed.

If you have plotted the mesh of a Solve-Inside object, you can view the internal mesh by using either the clip-plane feature or the Model Analysis dialog box.

After making selections click **Done**. The mesh plot will be generated on the objects belonging to the selected layer/net combinations.

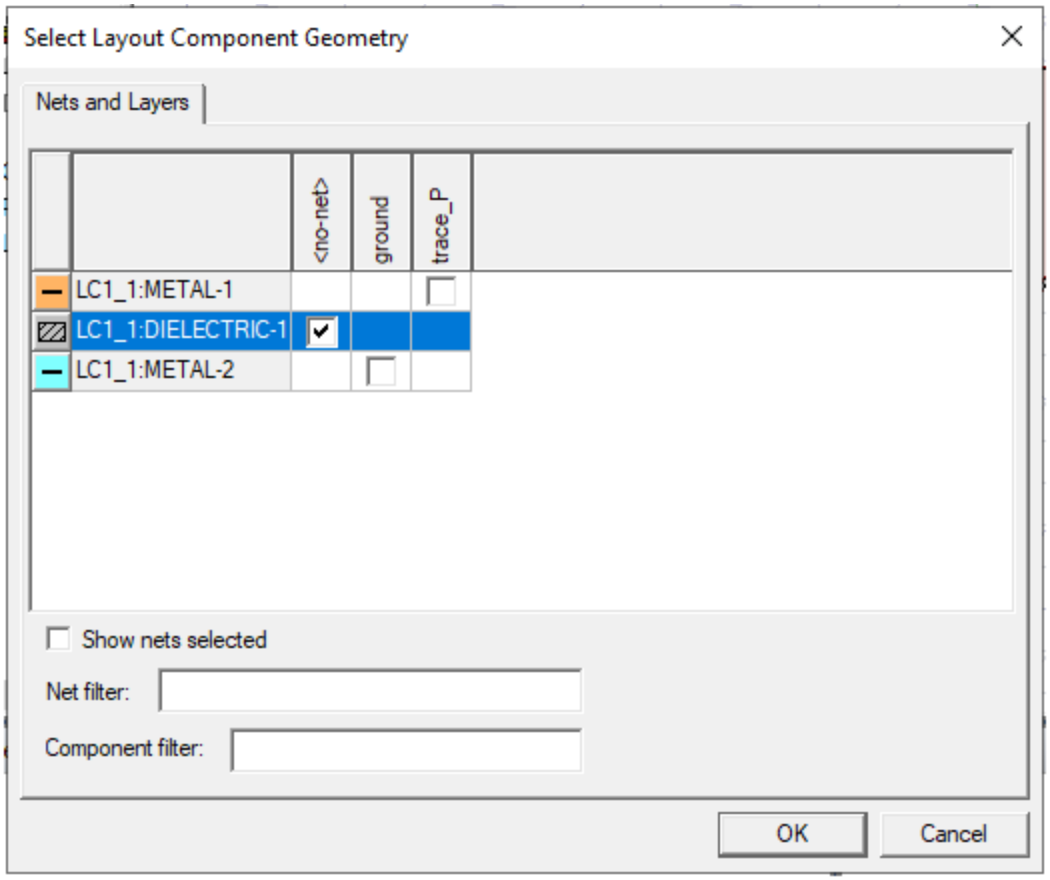
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to reassign a mesh or field plot, either select 3D geometries from layout components in the modeler window before selecting **Reassign** in the right-click menu, or select **Reassign**.



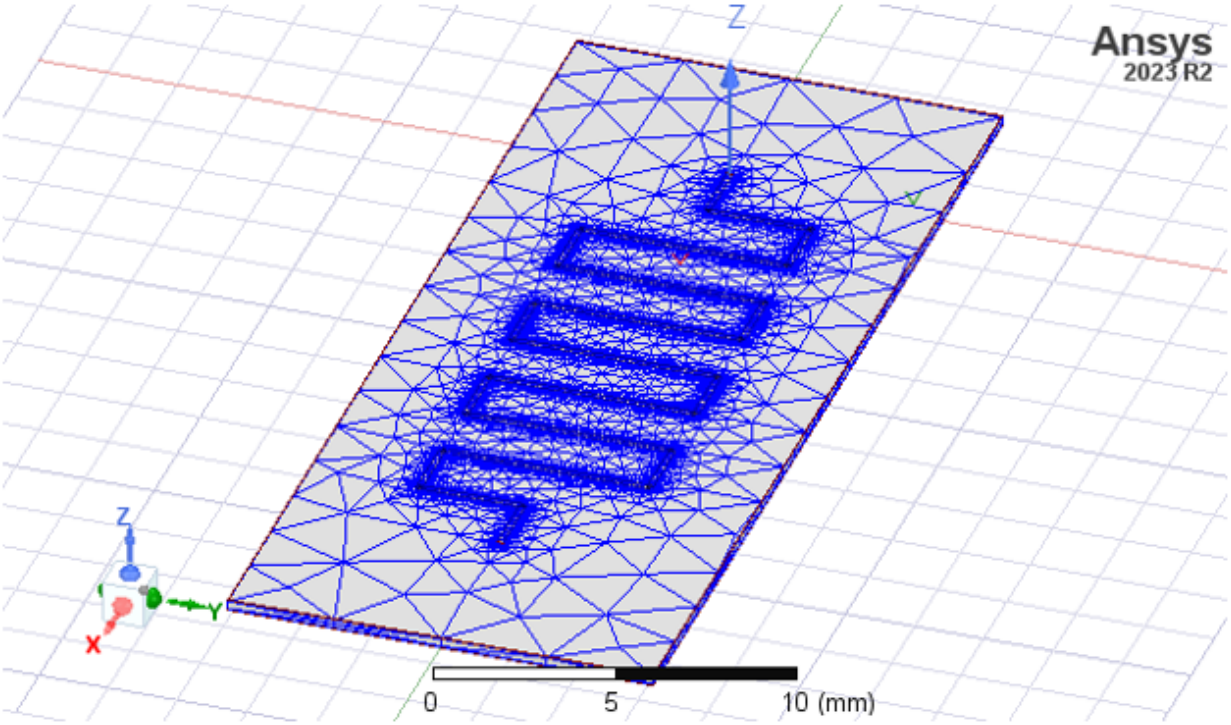
For HFSS designs with layout components, the **Select Layout Component Geometry** dialog appears. This has the same functionality and menus as the this **Nets and Layers** section of the **Create Mesh Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



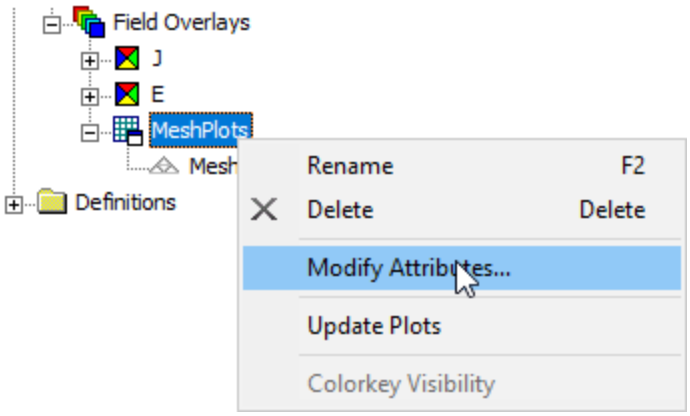
Change the selection in the dialog and click **OK**. The mesh plot will be re-assigned to the newly selected objects, in addition to any 3D geometries previously selected from the modeler window. If you click cancel, the plot is only re-assigned on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message. Change the selection in the dialog and click **OK** to display the reassigned mesh plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



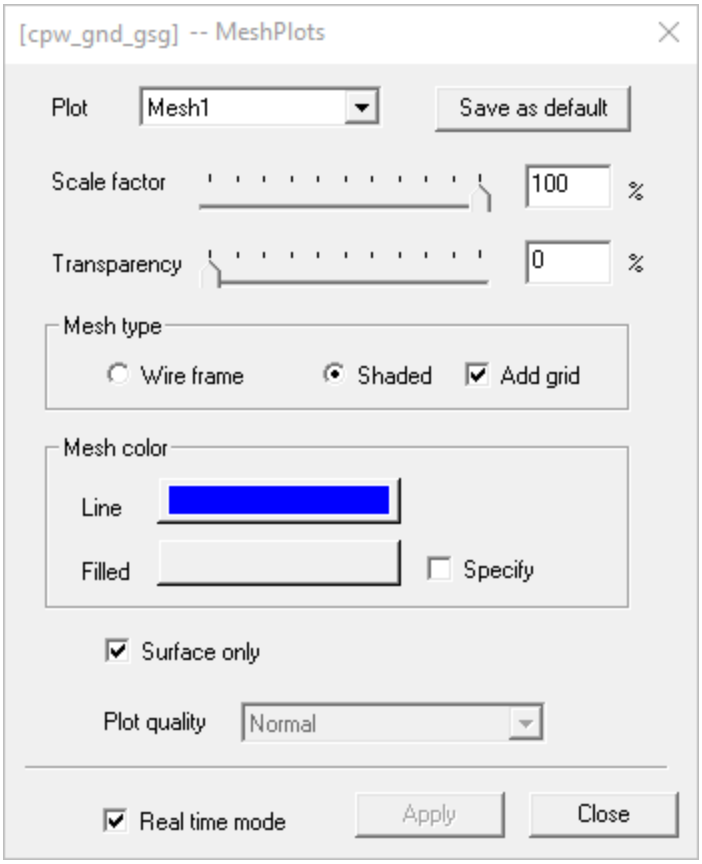
Setting Mesh Plot Attributes

1. In the Project Tree, expand **Field Overlays**.
2. Right-click **Mesh Plots** and select **Modify Attributes**.



The *MeshPlots* window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can also open the MeshPlots window from **Mechanical** > **Fields** > **Modify Plot Attributes**. You will be prompted to select the folder containing the mesh you want to modify.

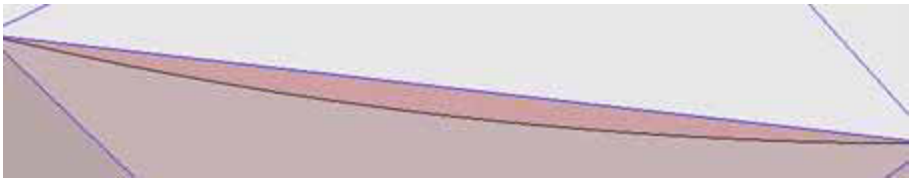
For Mesh plots, the following attributes can be modified:

Plot	A drop-down menu of available plots.
Scale Factor	The size at which the tets are displayed. Scaling may let you analyze particular situations better. For example, a scale factor of 80% draws the tetrahedra at 80% of their original size. Use the **Scale factor** slider to increase or decrease the size of the tetrahedra.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparency	The degree of transparency for the tets. This is useful for viewing objects or plots behind the current plot. Use the **Transparency** slider to increase or decrease plot transparency.
Mesh Type	Whether to display the tets as wire frame or shaded, and whether to Add Grid.
Mesh Color for Line and Fill	The color for the tet edge lines and fill. Clicking the button for each displays a color selection dialog box. You must select **Specify** to enable fill color.
Surface Only	Whether to plot the surface only, or all tets inside selected objects.
Plot Quality	The resolution quality of the plot. Normal by default. Use the drop-down menu to select Coarse, Normal, Fine, or Very Fine. The higher the resolution, the more memory used.
Real Time	Whether to show changes to a mesh immediately in the view window. If this option is cleared, click **Apply** when you want to see the changes.

For **Plot Quality**, higher quality selections permit visualization of curvilinear mesh elements. With the **Coarse** setting, facets do not snap to the midpoint edge:



The **Normal** setting displays snapping to the midpoint edge:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Fine** setting displays more of the curvilinear tets:

The **Very fine** setting gives the most accurate display, but uses the most memory:

3. Click **Save as default** button if you want the tab's settings to apply to mesh plots created after this point.
4. Click **Close**.

Generating Mesh without Solving

If you want to refine the mesh on a face or volume but do not want to generate a solution, define mesh operations. Then do one of the following:

- Select [**Mechanical**] > **Analysis Setup** > **Generate Mesh**.
- Right-click the **Analysis** or **Setup** icon in the **Project** window and select **Generate Mesh**.

The same solve machine rules that apply to solving any other setup also apply here. The mesh operation will be sent to the default solve machine. If you specified "Prompt for analysis machine when launching analysis" under **Tools** > **Options** > **General Options** > **Analysis Options**, the **Server Setup** dialog box appears and you can specify a solve machine.

- If a current mesh has been generated, the solver will refine it using defined mesh operations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a current mesh has not been generated, the solver will apply the mesh operations to the initial mesh.
- If an initial mesh has not been generated, the solver will generate it and apply the mesh operations to the initial mesh.
- If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.

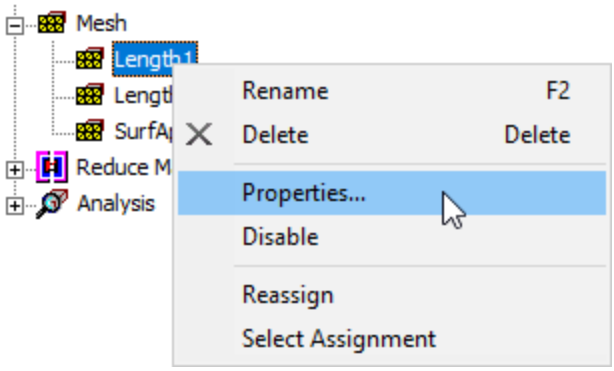
> **Tip:**
>
> Define a new mesh operation rather than modify an existing mesh operation. The solver will not re-apply a modified mesh operation.

Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them or by reverting to the initial mesh.

Viewing Mesh Properties

To view mesh properties:

- In the Project Tree under **Mesh**, right-click a previously completed mesh refinement and select **Properties**.



A dialog box opens with properties specific to that mesh (length-based, surface approximation, etc.). You can edit the mesh settings as appropriate.

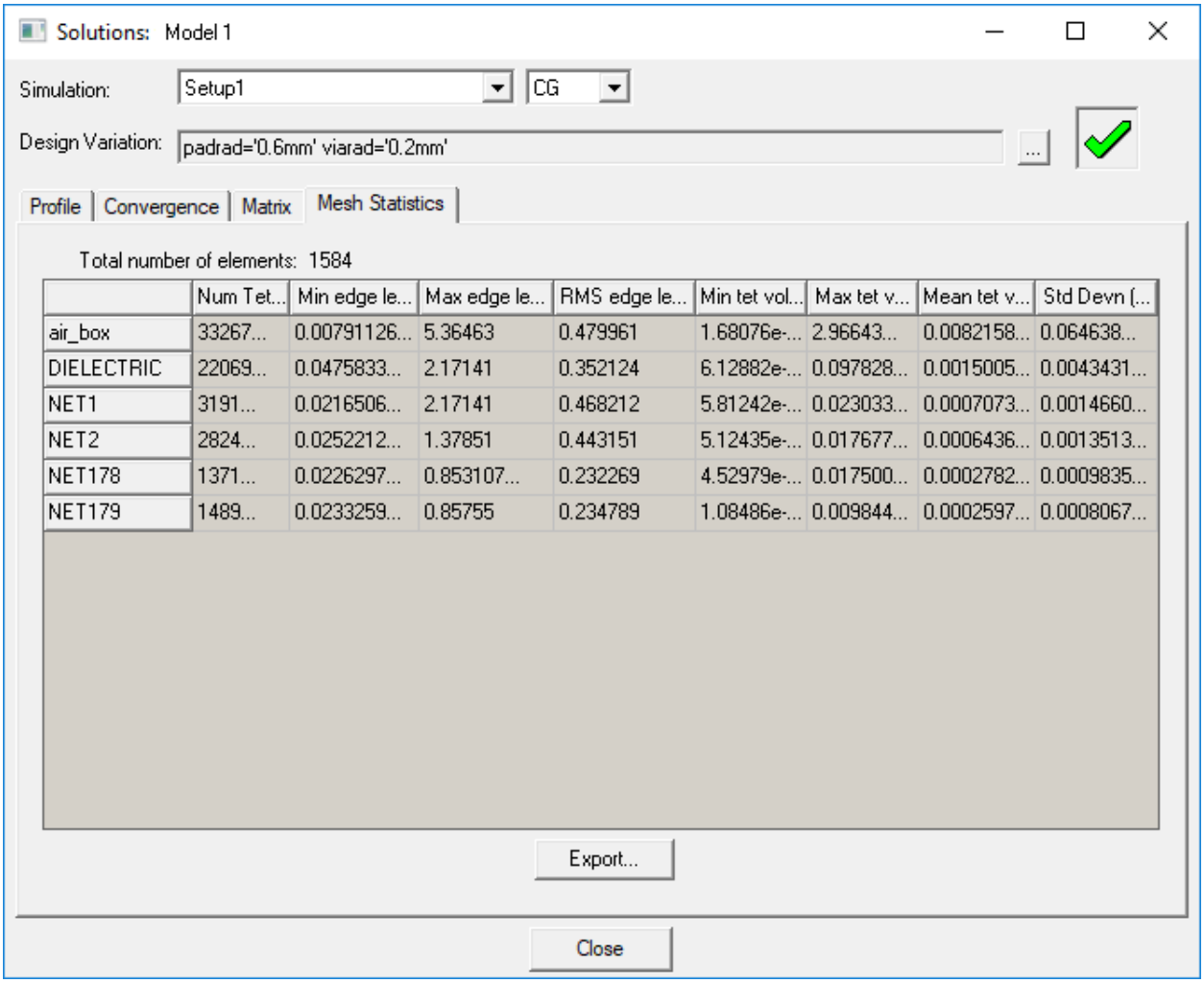
Viewing Mesh Statistics

To view an adaptive solution's mesh information, either during or after the solution process:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Tree, right-click the solution setup of interest, and then click **Mesh Statistics**. Alternately, click the **Solution Data** icon on **Results** tab of the ribbon.

   The **Solutions** window appears with the **Mesh Statistics** tab displayed.



The table lists design elements and for each includes: Num Tets, Min edge length, Max edge length, RMS edge length, min tet vol., max tet vol., mean tet vol. and standard deviation (min elem area, max elem area, mean elem area and standard deviation for 2D designs).

If mesh repairs have been performed, two additional columns appear in the table: **Recovered %** and **Repaired %**. These columns indicate the fraction of an object that was successfully recovered, and the fraction that needed some repair.

To sort mesh statistics by column from low to high values or vice-versa, click a column header. This displays a shadowed triangle pointing down to indicate a list ordered from highest to lowest;

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and a triangle pointing up to indicate a list ordered from lowest to highest. Clicking again inverts the order.

Click on a blank cell above the object list to invert the order of objects. In this case, the cell does not display a directional triangle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20 - Toolkits

In the Ansys Electronics Desktop, you can add toolkits, which are custom functions that perform a specific task. Toolkits can help automate and streamline the process of building your Ansys Mechanical design. Toolkits can aid in:

- Geometry creation
- Modeling (boundaries, excitations, solution setup)
- Productivity
- Post processing & reporting

Use IronPython for scripting the user interface and commands. Custom toolkits can be accessed from the **Mechanical** > **Toolkit** menu.

Currently, no toolkits are included for Mechanical designs, so none will appear when you first access the **Toolkit** menu. However, you can create toolkit folders, create your own custom scripts, and include them in the **Toolkit** menu. The folder names must be defined exactly as described and pictured in the following instructions:

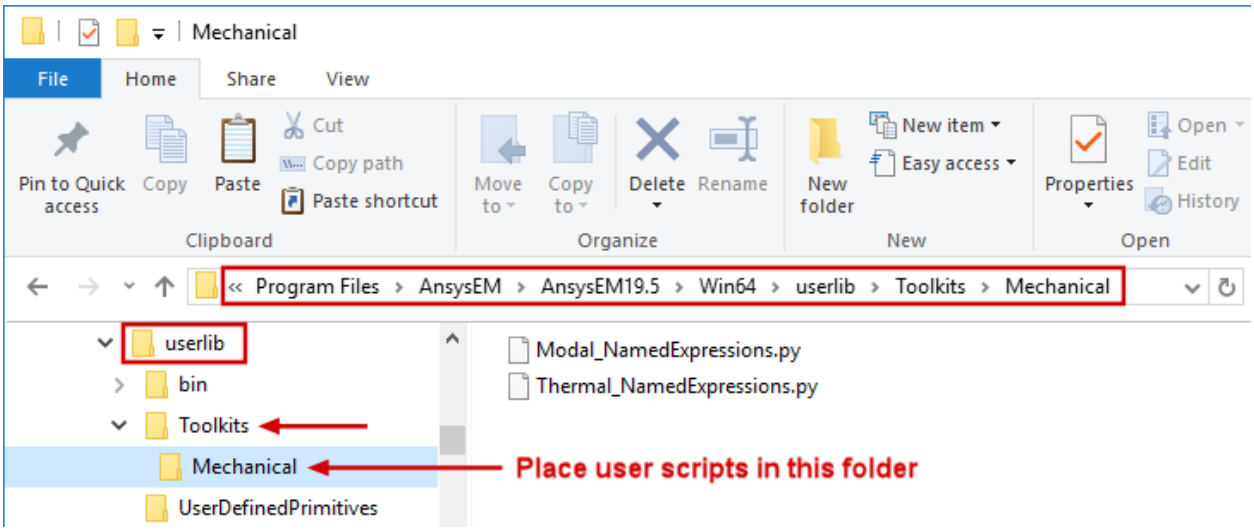
1. Create a **Toolkits** folder in the **"<*installation_dir*>\v251\AnsysEM\userlib"** directory.

   The default <*installation_dir*> path is "C:\Program Files\ANSYS Inc" on Windows and /opt/ansys_inc/ on Linux.

   > **Note:**
   >
   > You must have administrative privileges to create folders and files within the *Program Files* branch of the system drive.

2. Create a **Mechanical** subfolder in the *Toolkits* folder.
3. Save one or more script files to the **Mechanical** subfolder at the bottom of the path just created.

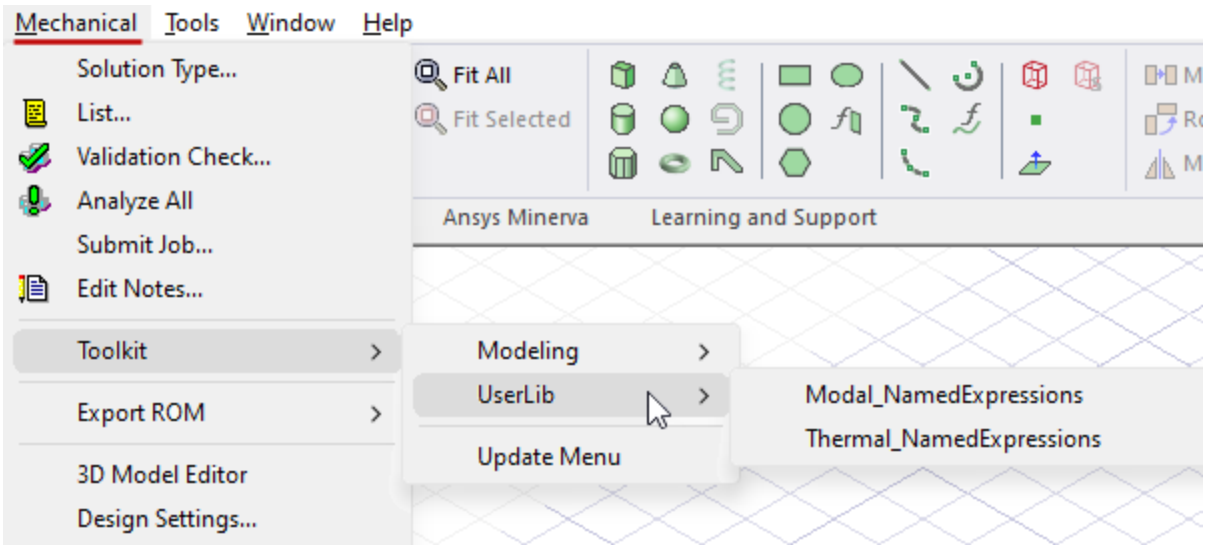
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Do not attempt to record scripts directly to a folder in the *Program Files* branch of your system drive. Record the script to a working folder, edit it as needed, and then copy the *.py file into the **...\userlib\Toolkits\Mechanical** folder.

4. Click **Mechanical** > **Toolkit** > **Update Menu** to refresh the menu contents.
5. Go to the **Mechanical** > **Toolkit** > **UserLib** menu to verify that your scripts are available for selection:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When you open or insert a Mechanical design, the *Mechanical* > *Toolkit* > *UserLib* menu is updated automatically. However, if you add, remove, or rename any scripts, you can use the **Mechanical** > **Toolkit** > **Update** command to immediately refresh the menu contents.

Modal Toolkit Example

Modal Example:

The following IronPython script creates named expressions to augment the predefined modal field results. Specifically, the normalized displacement and the X, Y, and Z components of displacement are added as available field results to any design in which you have executed this toolkit.

Normalized_Displacement is equal to the calculated displacement magnitude divided by the maximum displacement magnitude that occurs anywhere in the model. Therefore, the greatest value this result can have is 1. However, under either of the following two conditions, the maximum result on your plot, table, or overlay may be less than 1:

- The *Normalized_Displacement* is plotted or tabulated along a polyline that does not pass through the point of maximum displacement
- The objects or faces selected for a *Normalized_Displacement* overlay do not contain the point of maximum displacement

> **Note:**
>
> - Before attempting to copy this toolkit example to your computer, create the ...*Toolkits**Mechanical* folder following the instructions in the Toolkits topic.
> - You must have administrative privileges to create folders in, or copy files to, the *Program Files* branch of your computer's system drive.

Copy and paste the following script into a plain text file and save it to a working folder using the file name, **Modal_NamedExpressions.py**. Then, copy the file into the ...**\Toolkits\Mechanical** folder you created previously. After updating the *Toolkit* menu (**Mechanical** > **Toolkit** > **Update**), the Iron Python script will appear in the *Mechanical* > *Toolkit* > *UserLib* menu.

Iron Python Script – *Modal_NamedExpressions.py*:

```
# ------------------------------------------------ #
# Ansys Electronics Desktop Version 2025.1         #
# ------------------------------------------------ #
# This script adds the following Modal Field       #
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# results as Named Expressions:                          #
#   * Normalized_Displacement                            #
#   * X_Displacement Component                           #
#   * Y_Displacement Component                           #
#   * Z_Displacement Component                           #
# ---------------------------------------------------- #
# NOTE: To determine the Normalized Displacement, the  #
# maximum displacement magnitude is calculated with    #
# ALL OBJECTS considered. Therefore, when basing       #
# your report or overlay on fewer than all objects,    #
# the maximum value shown may be less than 1.          #
# ---------------------------------------------------- #
import ScriptEnv
ScriptEnv.Initialize("Ansoft.ElectronicsDesktop")
oDesktop.RestoreWindow()
oProject = oDesktop.GetActiveProject()
oDesign = oProject.GetActiveDesign()
oModule = oDesign.GetModule("FieldsReporter")
oModule.CopyNamedExprToStack("Mag_Displacement")
oModule.CopyNamedExprToStack("Mag_Displacement")
oModule.EnterVol("AllObjects")
oModule.CalcOp("Maximum")
oModule.CalcOp("/")
oModule.AddNamedExpression("Normalized_Mag_Displacement",
"Fields")
oModule.EnterQty("Displacement")
oModule.CalcOp("ScalarX")
oModule.AddNamedExpression("X_Displacement", "Fields")
oModule.EnterQty("Displacement")
oModule.CalcOp("ScalarY")
oModule.AddNamedExpression("Y_Displacement", "Fields")
oModule.EnterQty("Displacement")
oModule.CalcOp("ScalarZ")
oModule.AddNamedExpression("Z_Displacement", "Fields")
```

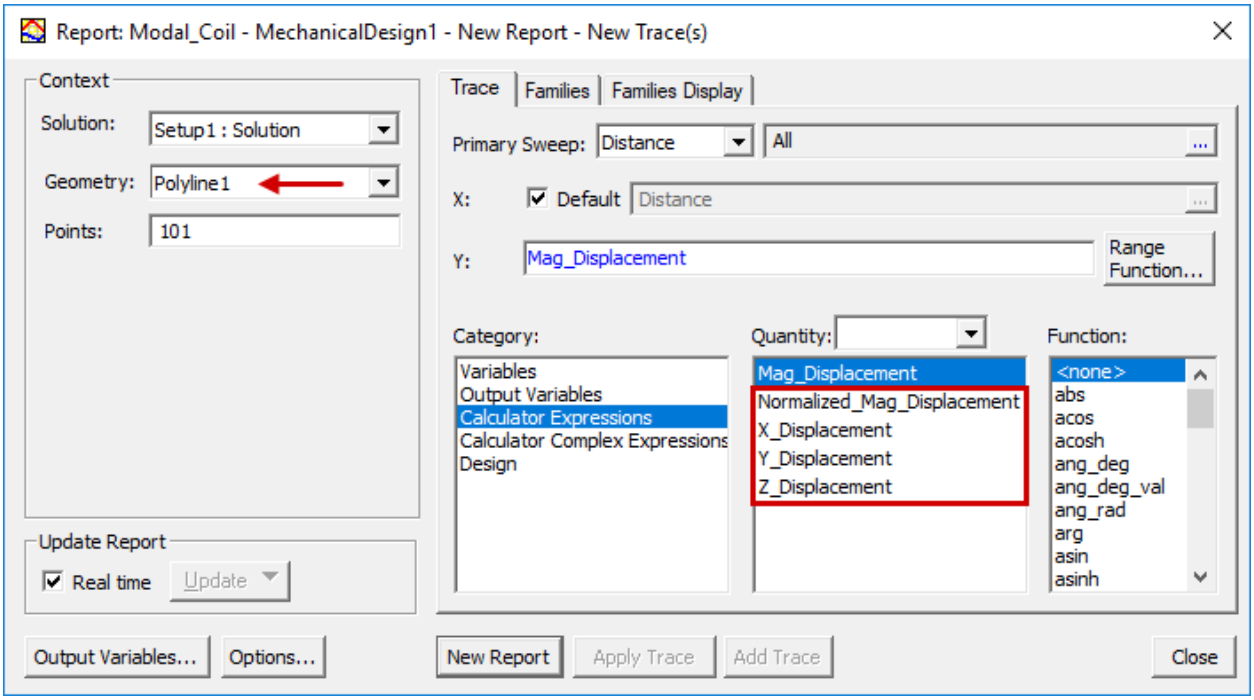
Instructions:

You must run this toolkit once (**Mechanical** > **Toolkit** > **UserLib** > **Modal_NamedExpressions**) for each new Mechanical – Modal design you create in which you wish to see the additional field results. After running this toolkit, the additional *Calculator Expressions* shown below (in the red box) will appear in the *Reports* dialog box for all *Fields Reports*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

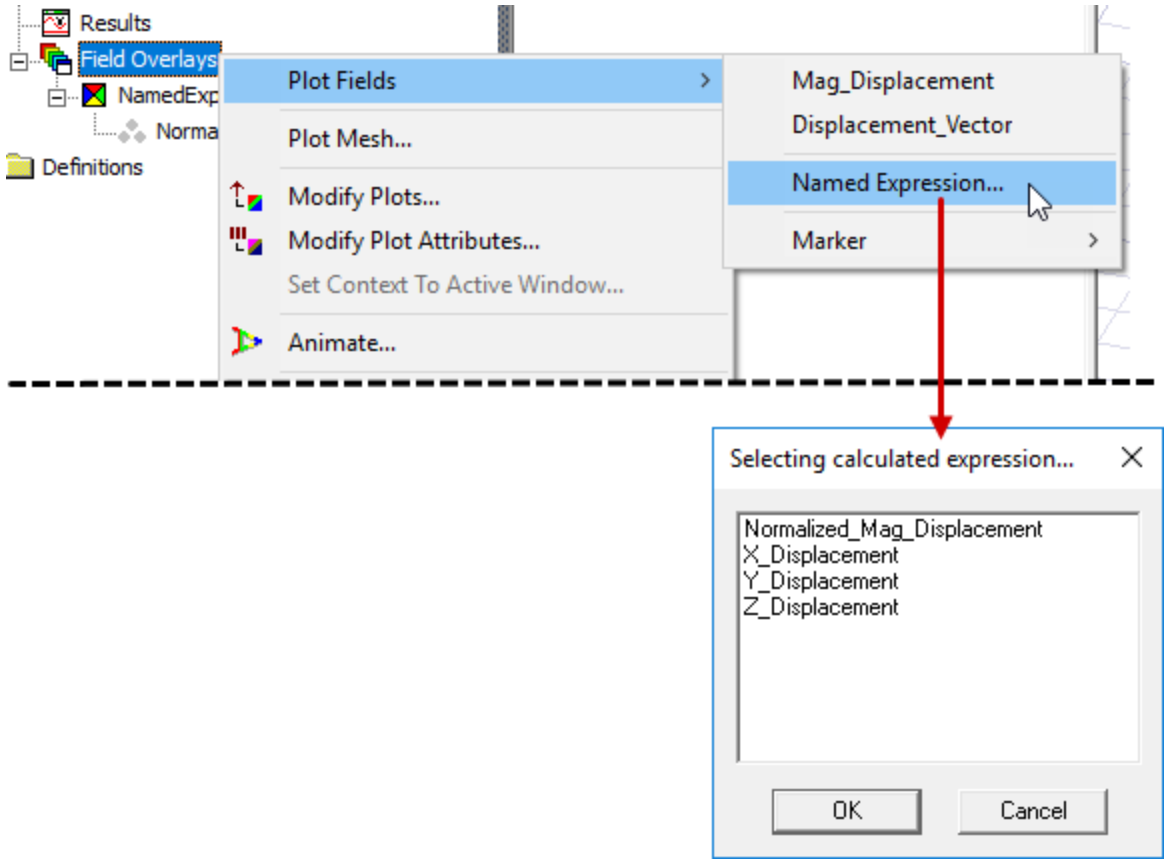
Note:

As is true for the predefined *Mag_Displacement* result, you must draw a polyline and select it as the *Geometry* along which these additional *Calculator Expression* field results will be reported.



These *Named Expressions* will also be available from the *Selecting calculated expression* dialog box (for field overlays):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Thermal Toolkit Example

The following Iron Python script creates named expressions to augment the predefined thermal field results. Specifically, the global maximum, mean, and minimum temperatures are added as available field results to any design in which you have executed this toolkit.

The scope of these temperature results is the full set of object volumes. For a steady-state solution, there is a single value reported for each named expression, three total. For a transient solution, the three temperature results can be reported at the first and last time step plus any additional time steps for which you optionally chose to save field results.

Note:

- Before attempting to copy this toolkit example to your computer, create the ...*Toolkits\Mechanical* folder following the instructions in the Toolkits topic.
- You must have administrative privileges to create folders in, or copy files to, the *Program Files* branch of your computer's system drive.
- Indentation and spacing are critical for Iron Python scripts. Do not alter the indentation or spacing, substitute tabs for spaces, or remove blank lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy and paste the following script into a plain text file and save it to a working folder using the file name, **Thermal_HeatRateOfFace.py**. Then, copy the file into the ...**\Toolkits\Mechanical** folder you created previously. After updating the *Toolkit* menu (**Mechanical** > **Toolkit** > **Update**), the Iron Python script will appear in the *Mechanical* > *Toolkit* > *UserLib* menu.

Iron Python Script – *Thermal_NamedExpressions.py*:

```
# ------------------------------------------------- #
# Ansys Electronics Desktop Version 2025.1          #
# ------------------------------------------------- #
# This script adds the following three Thermal fields   #
# results as Named Expressions:                     #
#    *  Global_Max_Temp                             #
#    *  Global_Mean_Temp                            #
#    *  Global_Min_Temp                             #
# ------------------------------------------------- #
# NOTE: The scope of these expressions is all object    #
#       volumes. Therefore, for any steady-state,       #
#       solution, each expression produces as single    #
#       resultant value. Transient solutions produce a  #
#       max, mean, and min temperature results for each  #
#       time step that is saved.                        #
# ------------------------------------------------- #
#
# Initialize:
#
import ScriptEnv
ScriptEnv.Initialize("Ansoft.ElectronicsDesktop")
oDesktop.RestoreWindow()
oProject = oDesktop.GetActiveProject()
oDesign = oProject.GetActiveDesign()
oModule = oDesign.GetModule("FieldsReporter")
oModule.CalcStack("clear")
#
# Add Global Maximum Temperature expression:
#
oModule.EnterQty("Temp")
oModule.EnterVol("AllObjects")
oModule.CalcOp("Maximum")
oModule.AddNamedExpression("Global_Max_Temp", "Fields",
    [
        "Fundamental_Quantity:=", ["Temp"],
        "EnterVolume:="         , ["AllObjects"],
        "Operation:="           , ["VolumeValue"],
        "Operation:="           , ["Maximum"]
    ])
#
```

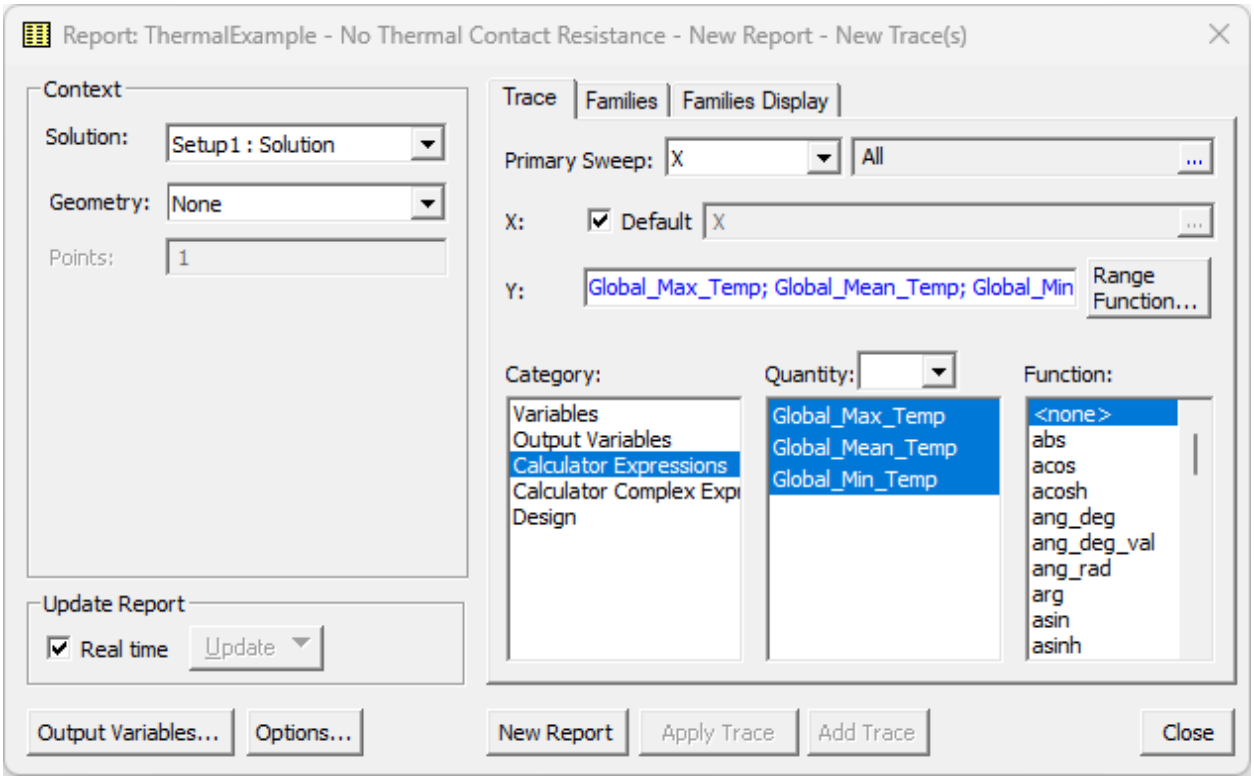

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Add Global Mean Temperature expression:
#
oModule.EnterQty("Temp")
oModule.EnterVol("AllObjects")
oModule.CalcOp("Mean")
oModule.AddNamedExpression("Global_Mean_Temp", "Fields",
    [
        "Fundamental_Quantity:=", ["Temp"],
        "EnterVolume:="         , ["AllObjects"],
        "Operation:="           , ["VolumeValue"],
        "Operation:="           , ["Mean"]
    ])
#
# Add Global Minimum Temperature expression:
#
oModule.EnterQty("Temp")
oModule.EnterVol("AllObjects")
oModule.CalcOp("Minimum")
oModule.AddNamedExpression("Global_Min_Temp", "Fields",
    [
        "Fundamental_Quantity:=", ["Temp"],
        "EnterVolume:="         , ["AllObjects"],
        "Operation:="           , ["VolumeValue"],
        "Operation:="           , ["Minimum"]
    ])
```

Instructions:

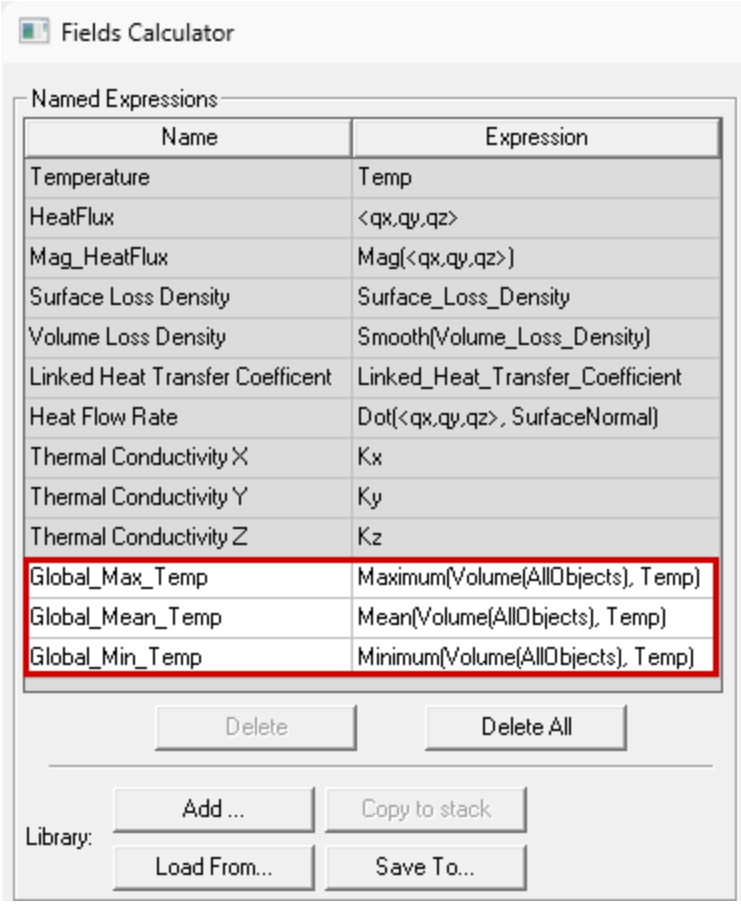
You must run this toolkit once (**Mechanical** > **Toolkit** > **UserLib** > **Thermal_ NamedExpressions**) for each new Mechanical – Thermal design you create in which you wish to see the additional field results. Thereafter, the named expressions appear in the *Report* dialog box for field plots or tables::

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Additionally, they will be available from the *Named Expressions* panel in the *Fields Calculator* window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While the three new named expressions will also be available for field overlays, a color contour based on a single global temperature value would not be useful. For steady-state thermal solutions, a data table is the best way to present the three values. For transient thermal solutions, the values can be presented as either a table or plot (curve traces) of the three values versus time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21 - Ansys Workbench Integration Overview

Ansys Workbench combines the strength of its core product solvers with the project management tools necessary to manage project workflow. In Ansys Workbench, analyses are built as *systems*, which can then be combined into a *project*. The project is driven by a schematic workflow that manages the connections between the systems.

From the schematic, you can interact with applications (called workspaces) that are native to Ansys Workbench and that display within the Ansys Workbench interface. Native workspaces include: Project Schematic, Engineering Data, and Design Exploration (Parameters and Design Points).

You can also launch applications that are data-integrated with Ansys Workbench, meaning the application's interface remains separate, but the data from the application communicates with the native Ansys Workbench data. Thus, data can be passed back and forth between any Ansys Electromagnetics product on a Workbench Project Schematic and any supported Ansys or Ansys Electromagnetics desktop product. Depending on the application, data integration can include basic actions such as saving projects, as well as more complex actions such as the coupling of Ansys Electromagnetics product variables to Workbench Design Exploration parameters.

Data-integrated applications include the following Ansys Electromagnetics products:

Circuit, HFSS, Icepak, Maxwell/RMxprt, Mechanical, Q3D Extractor, and Twin Builder.

> **Note:**
>
> For detailed information on working with Ansys Workbench, please refer to the Workbench documentation.

Integrating Ansys EM Suite with Ansys Workbench

You can integrate the Ansys Electromagnetics Suite with the Ansys Workbench application after both have been installed on your system. For successful integration, both preinstalled applications must be of the same version. A single operation completes the integration process for all supported design types.

The same procedure is used to integrate the applications or to decouple them (if they had previously been integrated). The procedure is as follows:

1. From the Windows Start Menu, select **All Programs** > **Ansys EM Suite 20*yy* R*n*** > **Modify Integration with Ansys 20*yy* R*n***.

   A *Modify Integration with Ansys 20yy Rn* command window appears.

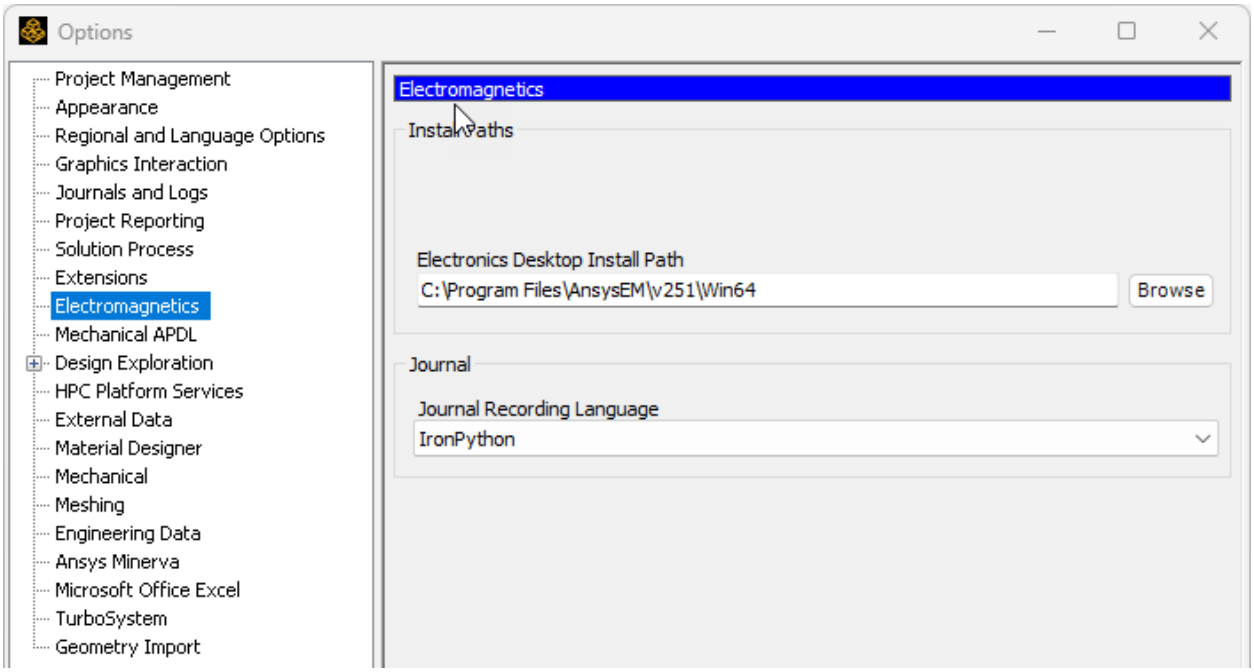
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If both applications are present on the system, their installation paths will be reported, and you will be asked if you wish to integrate them.
- Alternatively, if integration has already been preformed, you will be asked if you wish to decouple them.

2. Type **yes** to perform the prompted action or **no** to abort the operation without making any changes. Then press **Enter**
   - If you enter *"yes,"* a confirmation line appears in the command window indicating that integration (or decoupling) has been completed, and the window remains open.

     Press **Enter** again to close the command window.
   - If you enter *"no,"* the operation aborts and the command window closes immediately.

> **Note:**
>
> You can confirm that the Ansys 20*yy* R*n* application is “aware” of the integrated application via the *Options* dialog box in Workbench. The path to the integrated Electronics Desktop application is listed in the *Electromagnetics* group of settings.



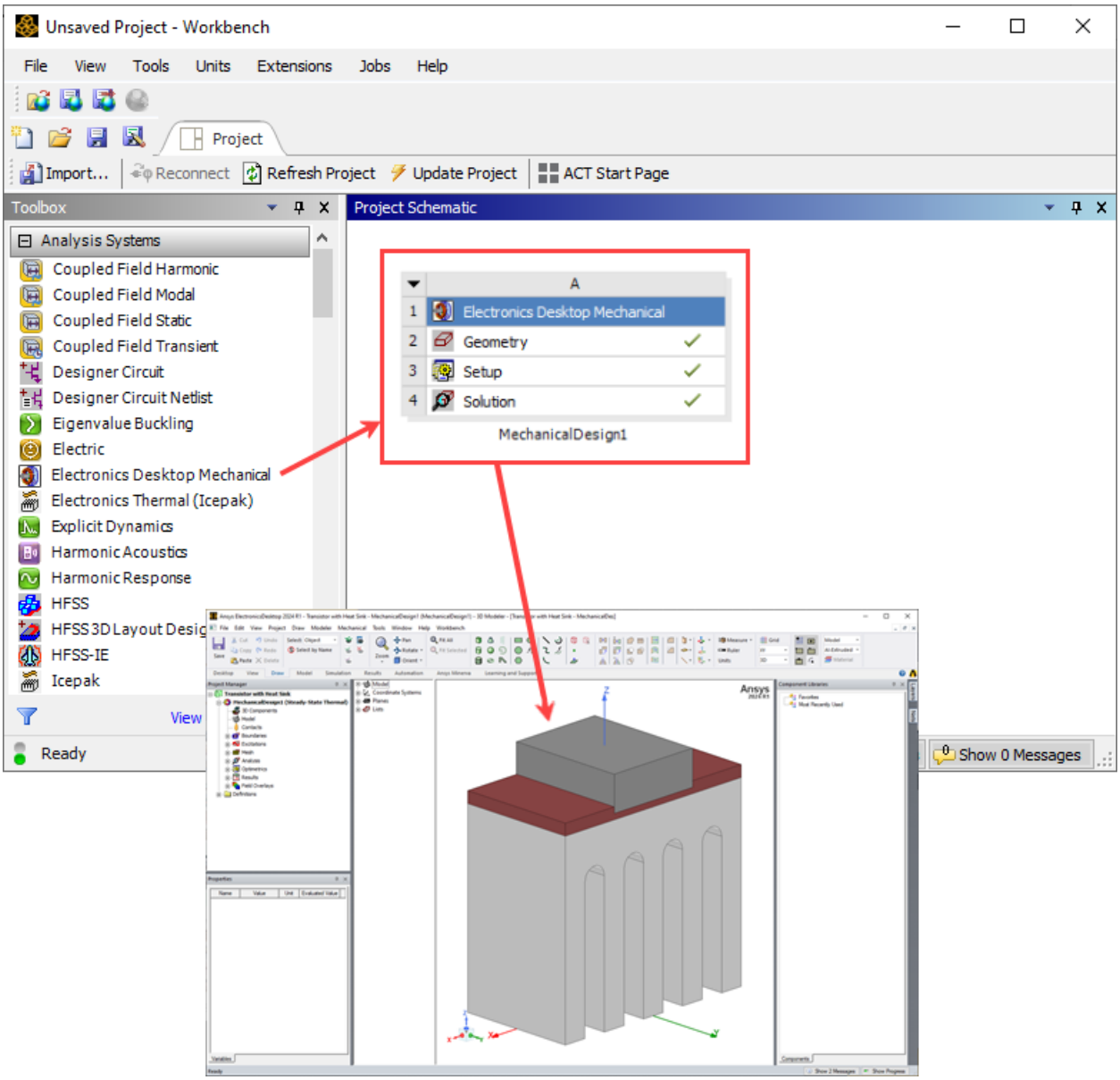
In this group of options, you can also specify Workbench's default **Journal Recording Language** for use when interoperating with the Ansys Electromagnetics products. **Iron Python** is the default language, and the alternative choice is **Visual Basic** (VB). Changes applied here

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

affect any new Workbench or Ansys Electronics Desktop session launched after the change. Any open instance will continue recording in the language that was in effect when the program was launched.

Workbench Data Integration Overview

Ansys EM data-integrated applications can reside on a Workbench **Project Schematic**, as shown.

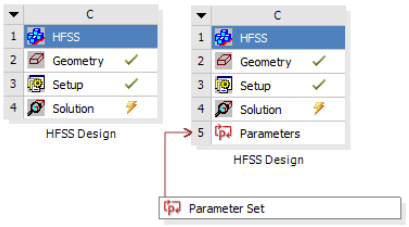


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Objects placed on a Workbench Project Schematic (such as instances of Ansys Electromagnetics projects) are referred to as *systems*. The number of cells in the system depend on the desigh type, as follows:

- **RMxprt, Circuit, HFSS 3D Layout,** and **Twin Builder:** Appear on the Workbench Project Schematic as systems with three cells: **<*System_Type_Name*>, Setup,** and **Solution**.
- **HFSS, Maxwell, Q3D Extractor, Icepak,** and **Mechanical:** Include an additional **Geometry** cell along with the **<*System_Type_Name*>, Setup,** and **Solution** cells, for a total of four cells.

If you invoke Ansys **DesignXplorer** to use variables for refining a design, a **Parameters** cell is added, with a link to the associated Workbench **Parameter Set**. Refer to the Ansys Workbench help for details on working with systems, cells, and parameter sets.



All supported Ansys Electromagnetics Suite products integrate with Workbench commands, services, and DesignXplorer in a similar manner. Some capabilities include:

- Adding New Analysis Systems
- Importing Existing Ansys EM Projects into Workbench
- Editing Ansys EM Models in Workbench
- Ansys EM to Workbench Geometry Transfer
- Ansys EM to Workbench Material Data Transfer
- Analyzing Ansys EM Models in Workbench
- Performing Parameter Studies in Workbench
- Scripting in Workbench

In addition to these major features, Workbench can archive, save, back up, duplicate, and delete Ansys Electromagnetics projects used in a Workbench project. Progress information and messages from integrated Ansys EM projects are also displayed in Workbench.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Look in the *Ansys Workbench Help* to find detailed information about performing the above-listed operations.

For details specific to creating or importing **Electronics Desktop Mechanical** systems in the Workbench application, see Electronics Desktop Mechanical Systems in Workbench.

Electronics Desktop Mechanical Systems in Workbench

The following details are specific to Ansys Electronics Desktop (AEDT) **Mechanical** systems within a Workbench schematic. These details apply Whether the system is newly created or is an existing AEDT Mechanical design that has been imported into the Workbench schematic:

- All Mechanical solution types are supported within a Workbench schematic (Modal, Steady-State Thermal, Transient Thermal, and Structural).
- Only *Steady-State Thermal* and *Transient Thermal* solution types are currently supported for linking *Electronics Desktop Mechanical* systems to other systems within the Workbench schematic.

  Additionally, within the Workbench schematic, you can only link the Engineering Data, Geometry, and Solution (temperature results) from a thermal solution in *Electronic Desktop Mechanical* to a *Static Structural* or *Transient Structural* analysis system.

  Any linking from one AEDT solution to another, aside from those described in the preceding paragraph, must be set up within the Electronics Desktop application, *not* the Workbench schematic. For example, mesh linking, importing convection film coefficients from Icepak, importing EM Losses from supported source solutions, or importing layout components for mesh mapping must be set up entierely within the AEDT application.

- Each AEDT project referenced within a Workbench schematic can only contain a single design.

  Instead of attempting to link to a multi-design AEDT project, create multiple *Electronics Desktop Mechanical* systems within the Workbench schematic, each referencing a single-design AEDT model.

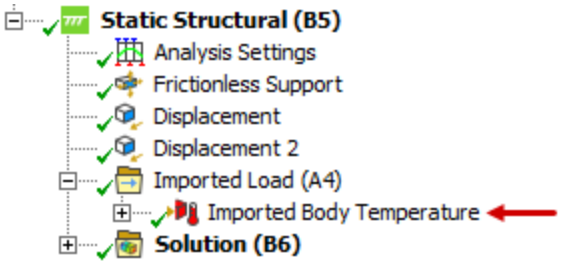
  Use the same practice whether linking to another Ansys Electronics Mechanical design or to HFSS, Icepak, Maxwell 2D, Maxwell 3D, or Q3D designs.

- When you create a new *Electronics Desktop Mechanical* system by dragging it from the Workbench Toolbox, the following will occur when you right-click the **Geometry** cell and choose **Edit**:

  The Ansys Electronics Desktop application will launch and a new project will be created having a blank Mechanical design already added to it.

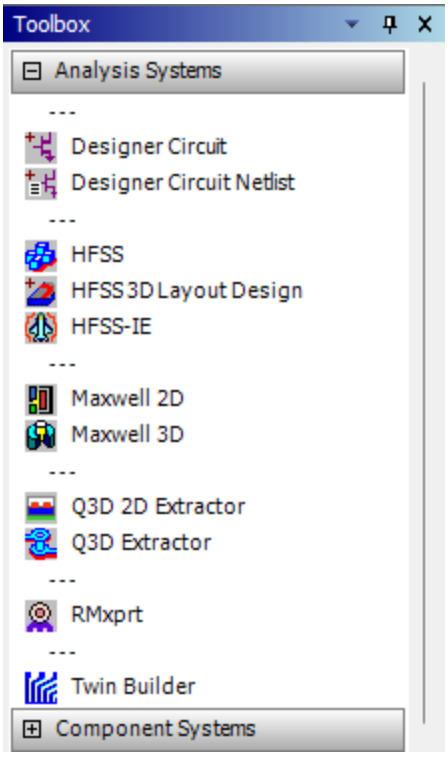
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you import an existing AEDT project, a copy of the design is created in a subfolder of the Workbench project location, preserving the original source design.
- After setting up and solving the Electronics Desktop Mechanical system and linking its solution to a Structural system setup, an **Imported Body Temperature** load will be automatically created and listed in the *Structural* solution branch of the Ansys Mechanical application. Edit the Structural setup to see this imported load.



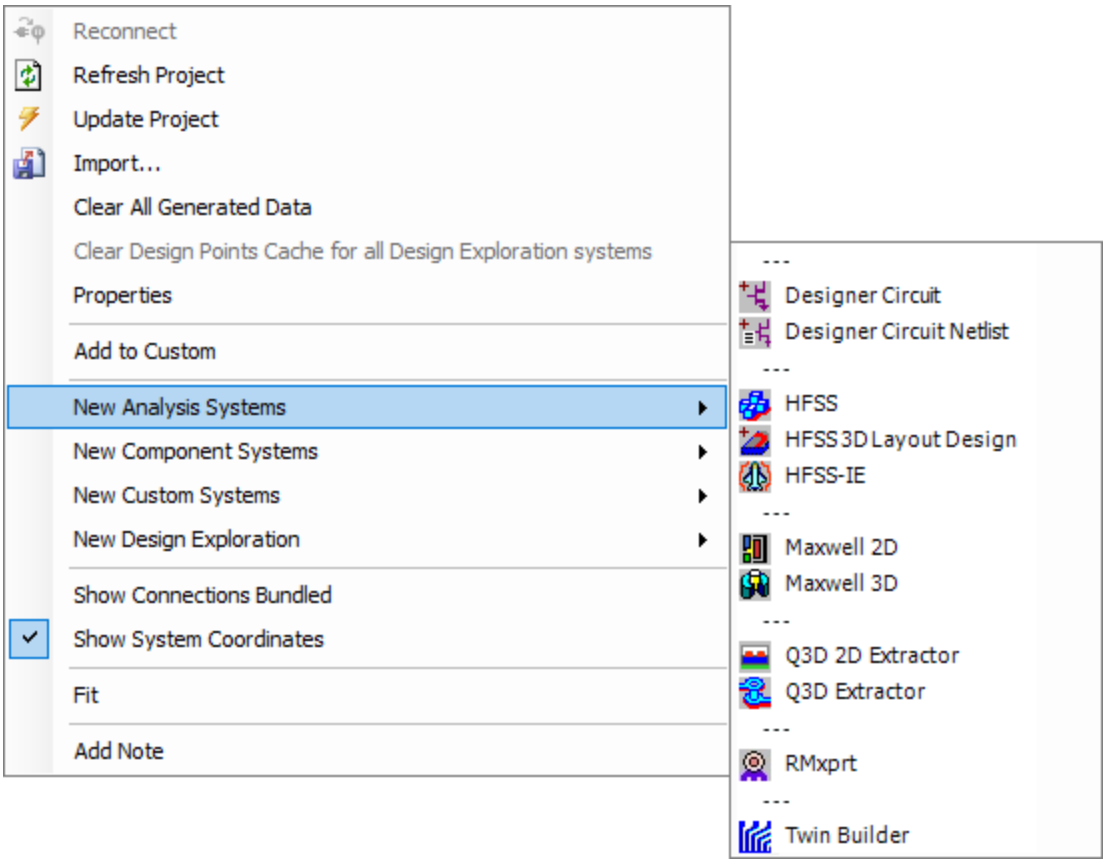
Adding New Analysis Systems

A new Ansys Electromagnetics Analysis System can be added to a Workbench Project Schematic by dragging and dropping it from the Toolbox:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternately, you can add a system by right-clicking in the Workbench Project Schematic area and selecting **New Analysis Systems** > **[System Type]**.

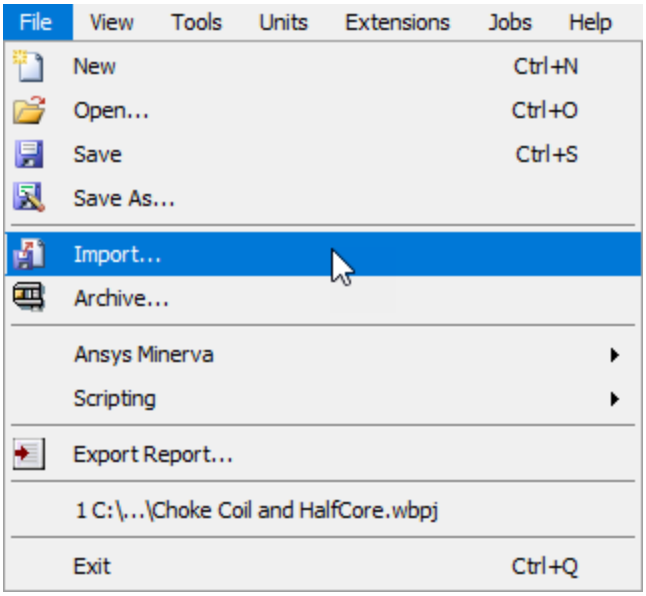


Importing Existing Ansys EM Projects into Workbench

You can import existing Ansys Electromagnetics Suite projects into a Workbench project schematic using **File** > **Import**. When you do, a copy of the Ansys Electromagnetics project is placed in the Workbench project folder. The original Ansys Electromagnetics project remains intact.

> **Note:**
>
> Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

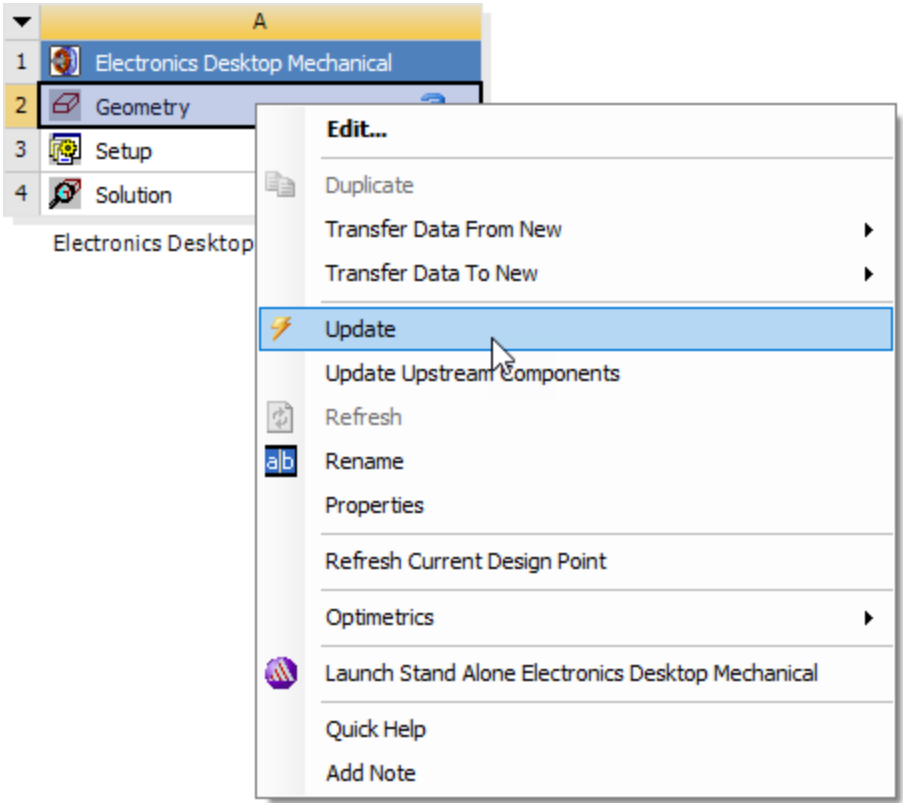


Editing Ansys EM Models in Workbench

You can edit various properties and parameters of the Ansys Electromagnetics project (geometry, setup, solution, etc.) either by right-clicking the project in Workbench and selecting **Edit**, or by double-clicking the project.

Doing so launches the Ansys Electromagnetics desktop application and loads the project so that you can set up your project in a familiar desktop environment. Changes made to the Ansys Electromagnetics project are saved to the project instance in the Workbench project folder.

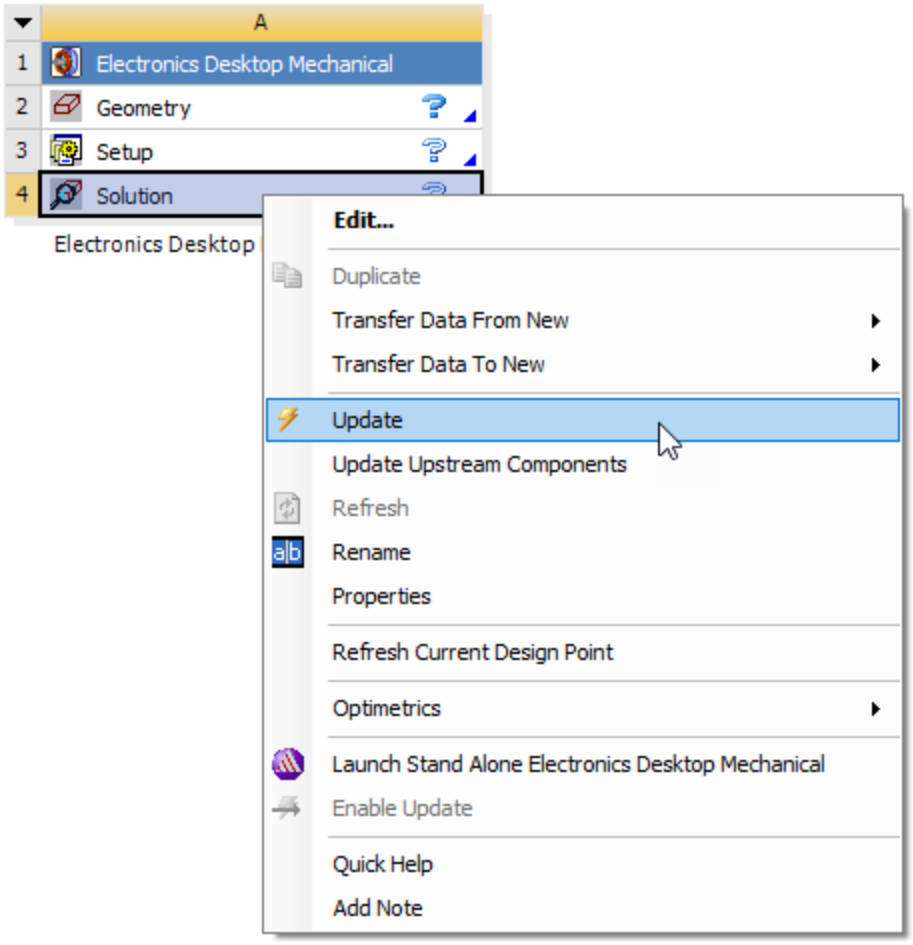
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



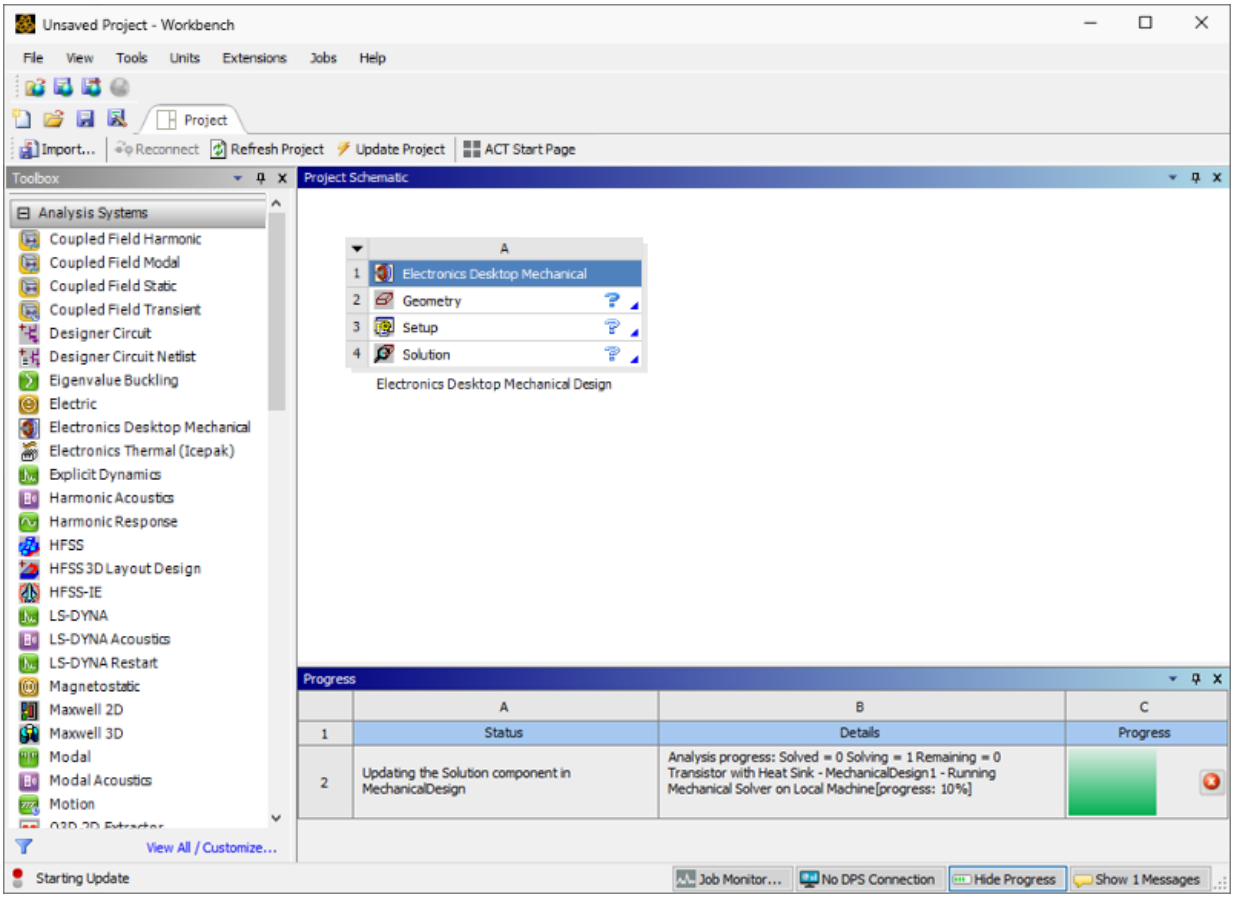
Analyzing Ansys EM Models in Workbench

You can use Workbench's **Update** command to run analyses in the integrated Ansys Electromagnetics project. Progress information is also shown in Workbench.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



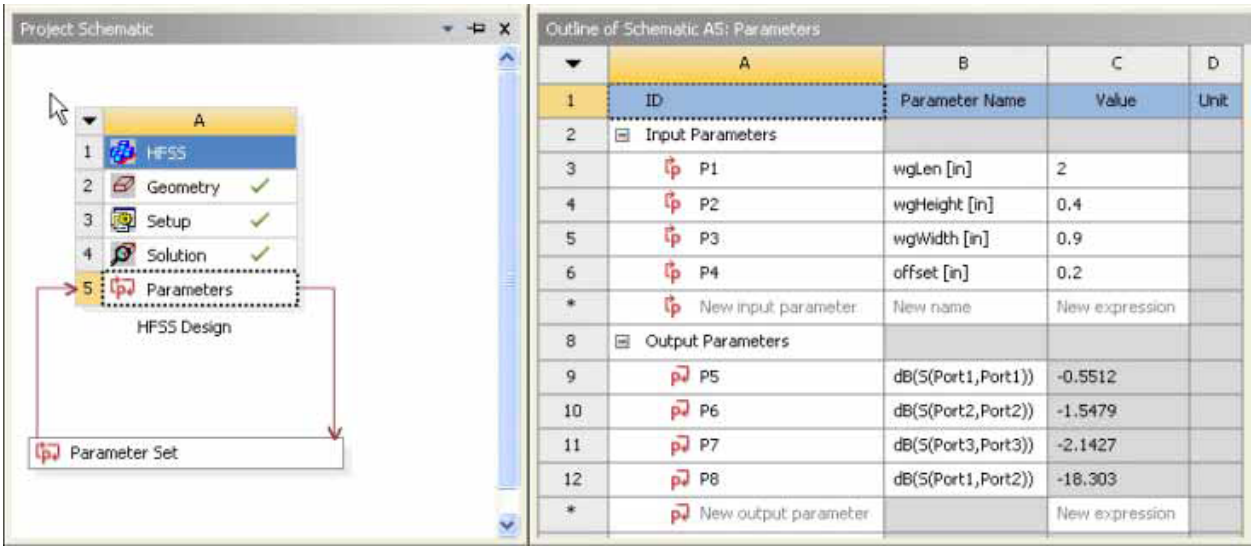
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Performing Parameter Studies in Workbench

Workbench **Parameter Sets** allow you to change parameter values and units of measure, or add new parameters. Parameter data is passed back to the Ansys Electromagnetics application for updated analyses. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

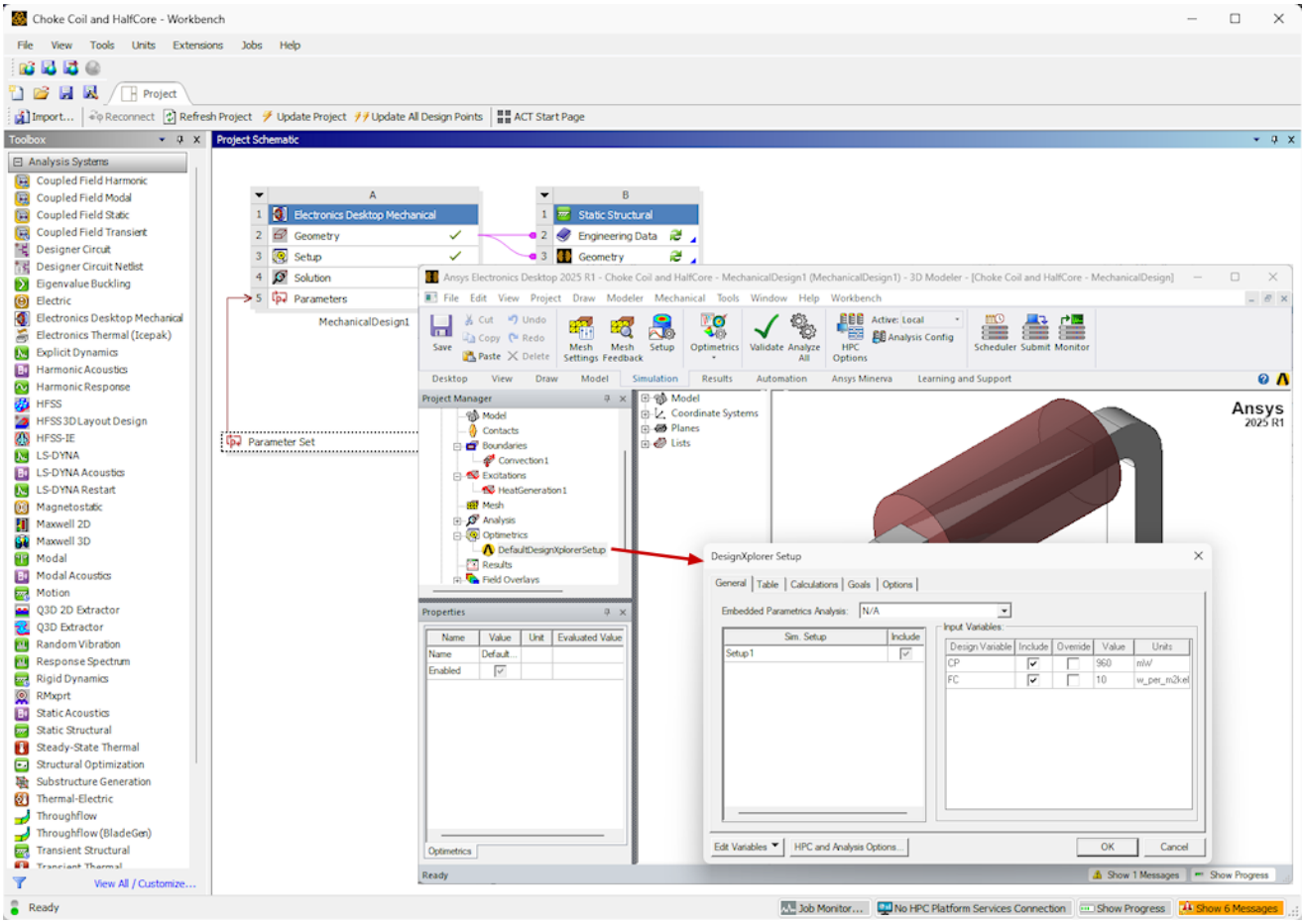


Note:

Because Workbench and Ansys Electromagnetics Desktop have different unit systems, units defined in Desktop are Dimensionless in the Workbench Parameters table and thus cannot be changed. Refer to the *Ansys Design Modeler User's Guide* > *Project Schematic Operations* > *Parameters in Project Schematic* > *Parameter Units* topic for additional information.

Parameters from the Ansys Electromagnetics project are exposed to Workbench through the **DesignXplorer** setup. The Ansys Electromagnetics system's cell status on the Workbench project is updated as changes are made in the Ansys Electromagnetics application. The following example is a Mechanical, Steady-State Thermal design in which two variables have been defined. One variable (CP) is the coil power (960 mW heat generation) and the other (FC) is the film coefficient of the assigned convection boundary:

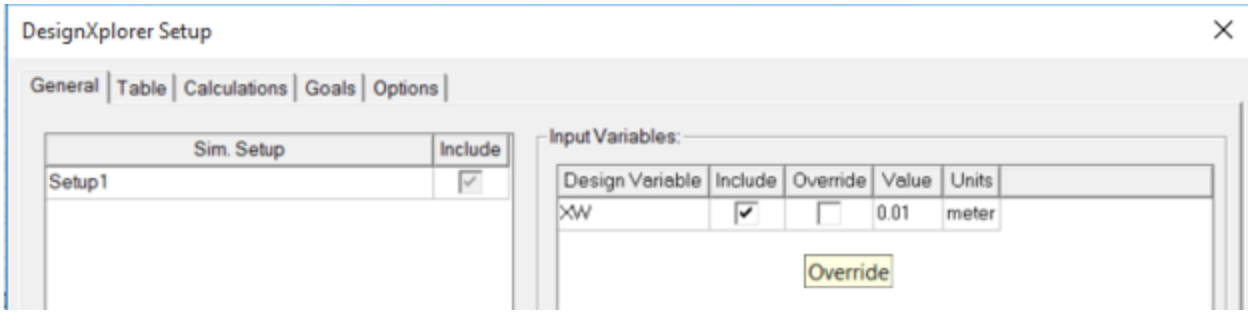
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



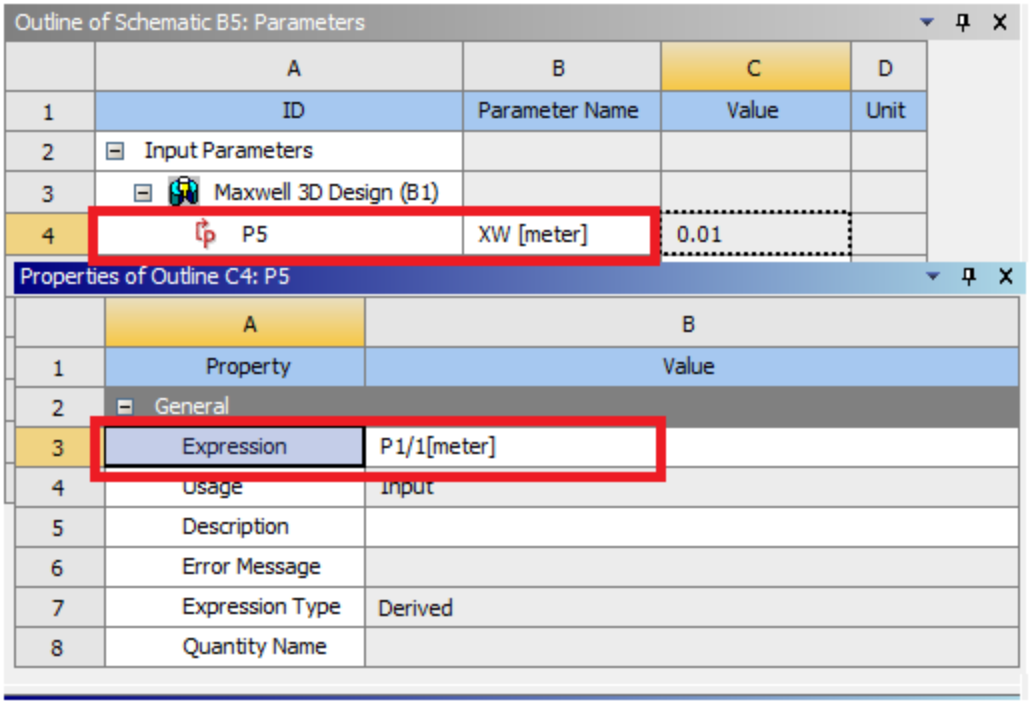
The workflow for using Electronics Desktop systems with the Workbench to take advantage of Distributed Analysis is as follows:

1. In the Ansys Electronics Desktop, specify a variable an as input parameter. The following figure shows a DesignXplorer Setup that includes a geometry variable called XW This variable is mapped to a variable in the corresponding Workbench Design, in this example, P5.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

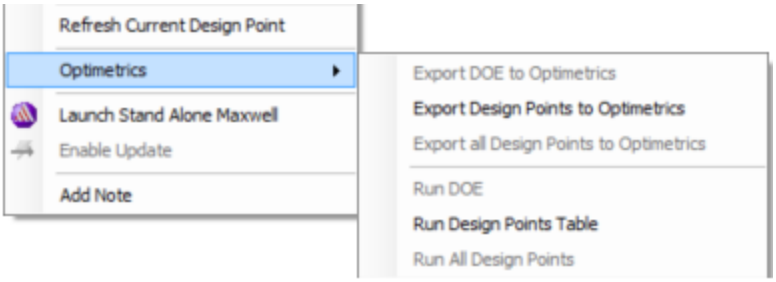


2. In the Workbench Design, change the P5 expression to a new value (in this example figure, the P5 parameter is assigned an expression of P1/1[meter] to tie it to the geometry system's variable.



In Workbench, the **Export Design Points to Optimetrics** and **Run Design Points Table** commands are enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

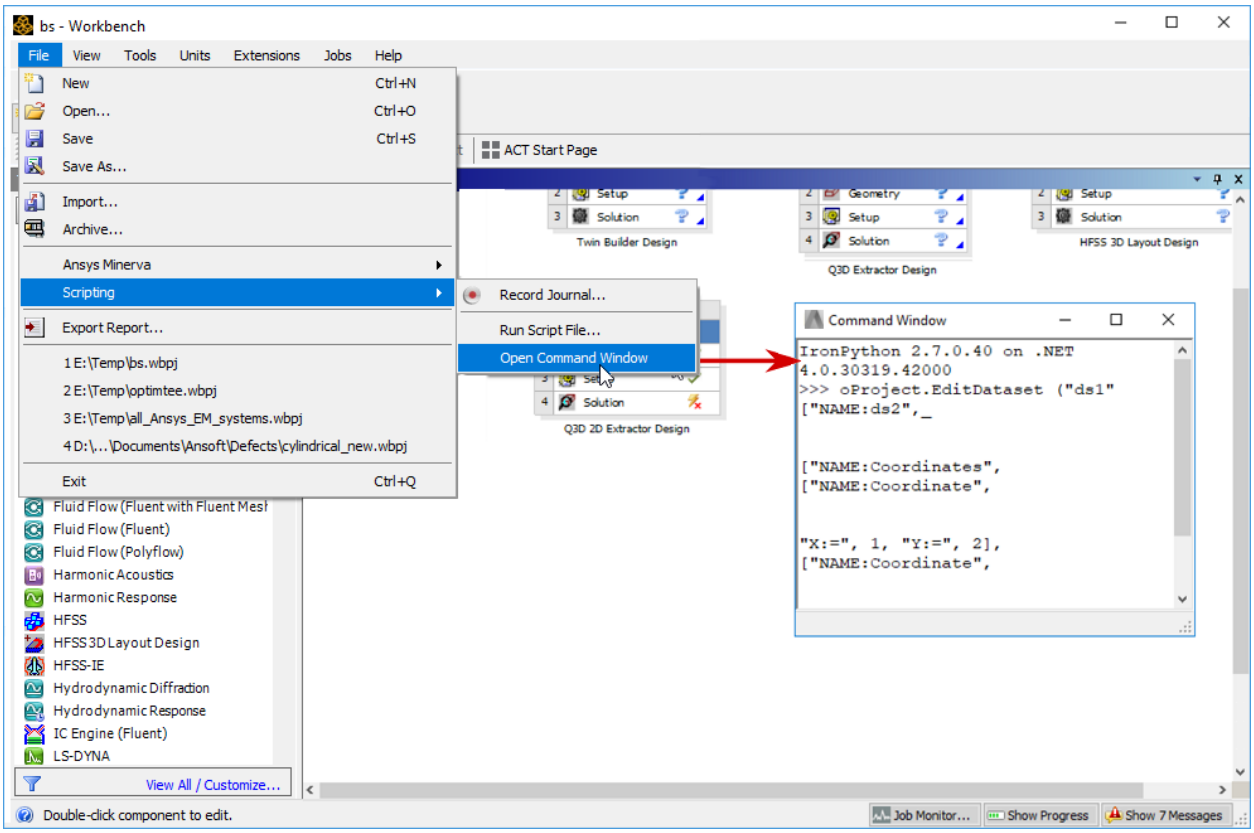


3. In Electronics Desktop, you can click on the default DesignXplorer Setup, and choose **Generate Variation Data**.

If you have configured your Analysis Configuration for multiple machines, the Project is ready for Distributed Analysis. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Scripting in Workbench

Scripts that include Ansys Electromagnetics projects can be recorded and run in Workbench.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

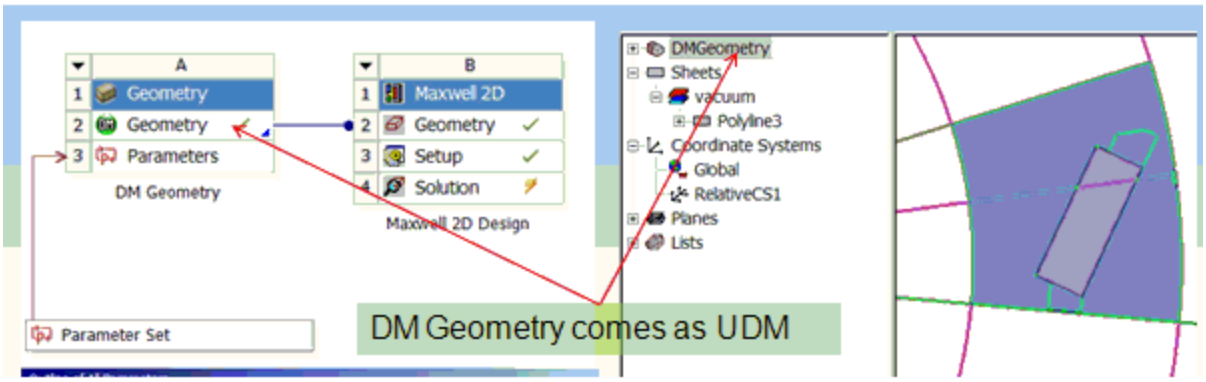
When a Workbench journal is run from the command line, Electronics Desktop allows the Ansys EDT portion to also run in batch mode (that is, when using RunWB2 -b).

Example: From a Windows Command prompt, enter the following command to replay the journal named "myReplayJournal.wbjn":

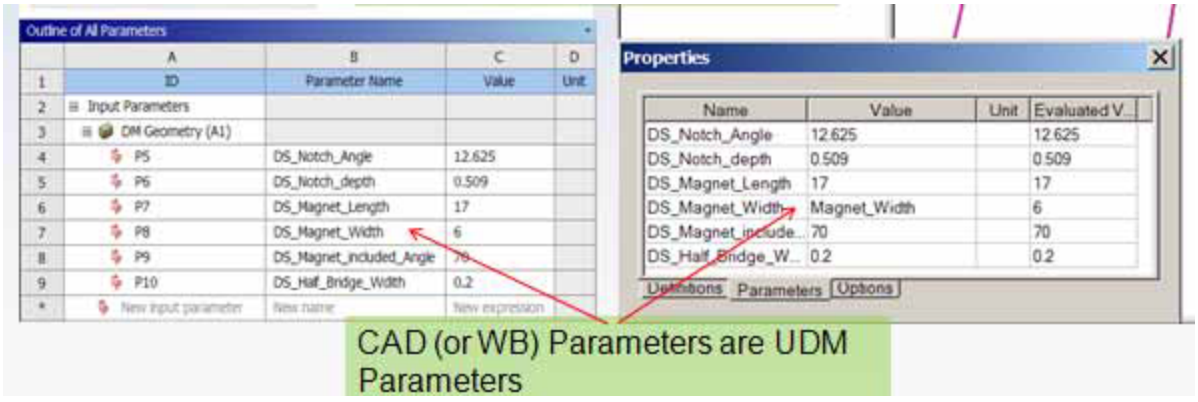
```
RunWB2 -b -r myReplayJournal.wbjn
```

Ansys EM CAD Integration through Workbench

Ansys Electromagnetics CAD integration is a Workbench feature available for Ansys Electromagnetics 3D Products - HFSS, Maxwell, Q3D, Icepak, and Mechanical, as the Ansys Framework. The feature is available only through Workbench, and is not available via standalone Ansys Electromagnetics products.



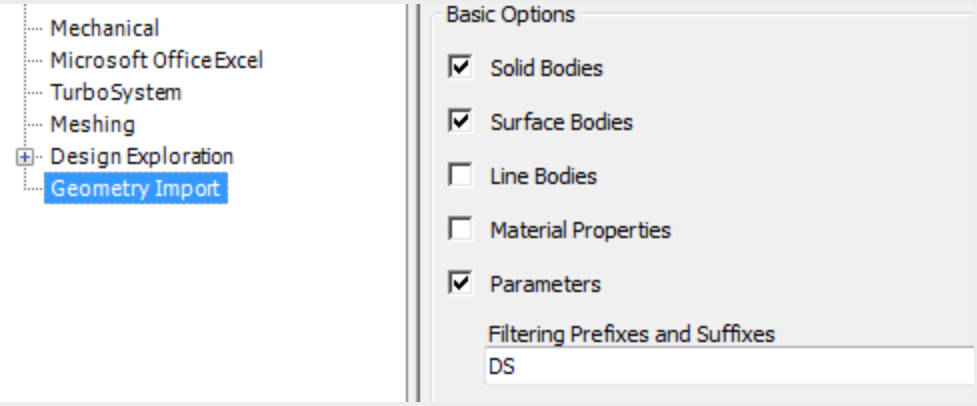
The UDM format makes it possible to exchange parameters. Ansys online help contains further description of the UDM feature and function.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The parameters shown in the previous example all have a DS prefix. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Ansys Electromagnetics CAD integration makes it possible to consume geometry from multiple upstream sources (any CAD or Ansys Electromagnetics product).

This feature supports direct interfaces with major CAD systems, including

- Creo Parametric
- UG NX
- CATIA V5
- SOLIDWORKS
- Autodesk Inventor
- Ansys Design Modeler (DM)
- Discovery UDM model

Note:

The CAD software must be installed on the user machine (not required for solve nodes).

The following sections contain additional information:

CAD Integration Functionality

CAD Integration and Geometry Sharing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bi-Directional CAD Integration

CAD Integration Model Edits

Multiple Geometry Links for CAD Integration

CAD Integration Functionality

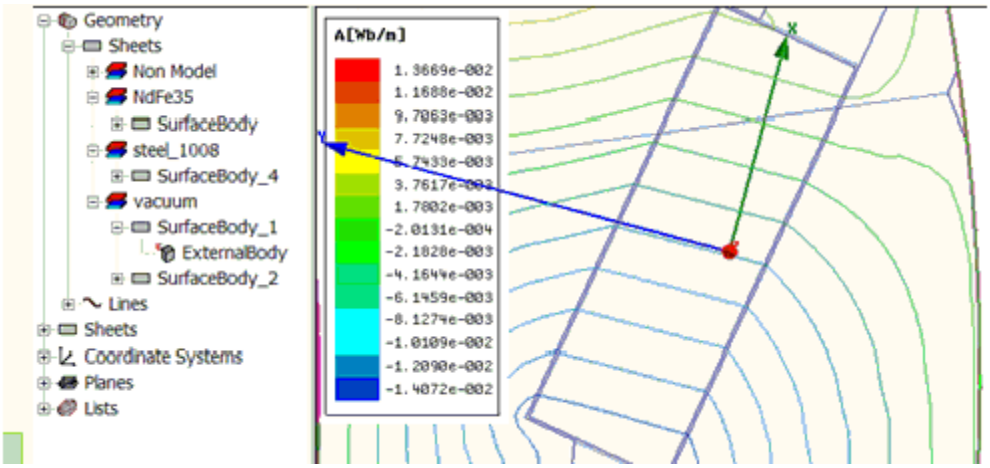
Healing with CAD Integration

Important Geometry Options for CAD Integration

CAD Integration Functionality

CAD Integration includes the following functionality:

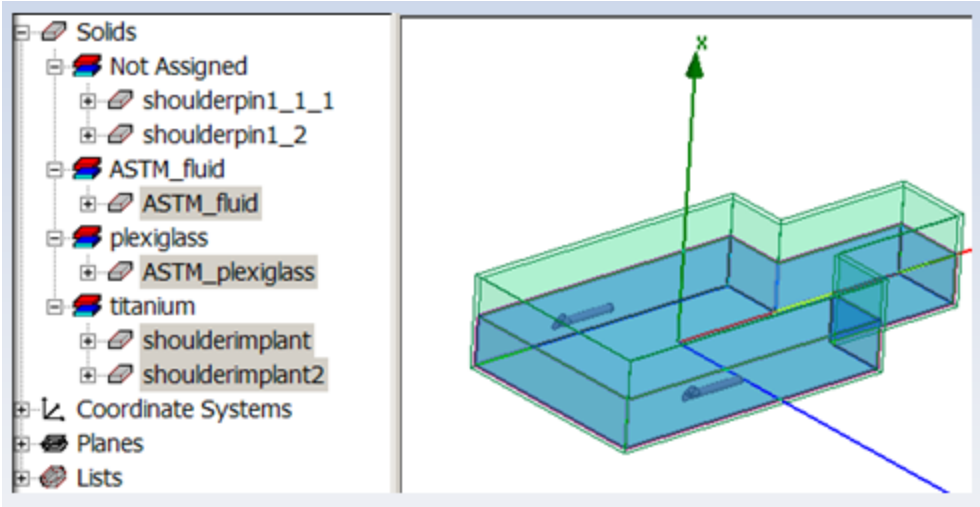
- DX analysis
- WB Update Project
- WB Update All Design Points
    - Parametric analysis with DSO
    - Animation
    - Geometry
    - Field plots



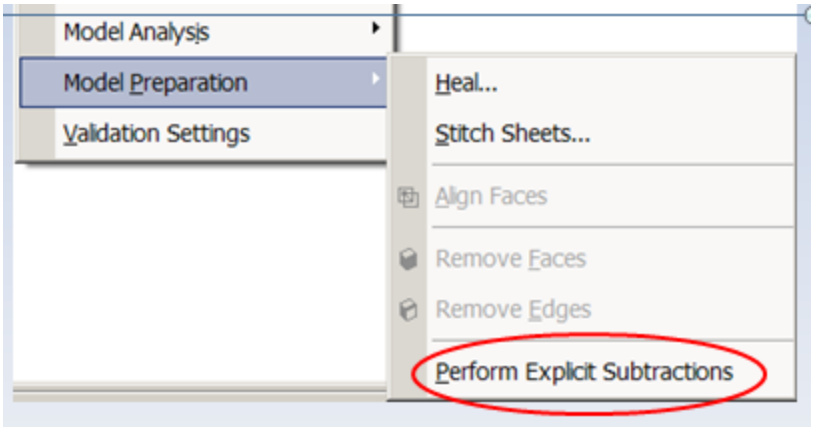
Ansys Mechanical does not do implicit subtraction. Ansys Electromagnetics products can do explicit subtractions before sending geometry to Ansys using **Modeler > Model Preparation > Perform Explicit Subtraction**.

For example, consider the following model.

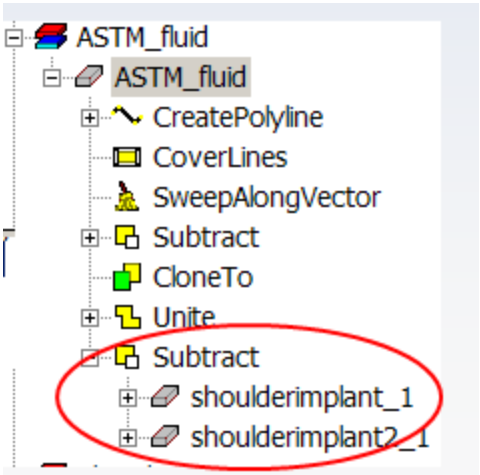
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Perform Explicit Subtractions can be performed.

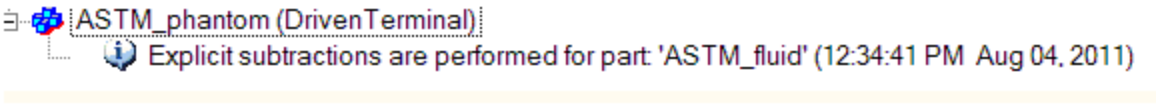


Results appear in the History tree, as shown:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Messages** window reports this action.

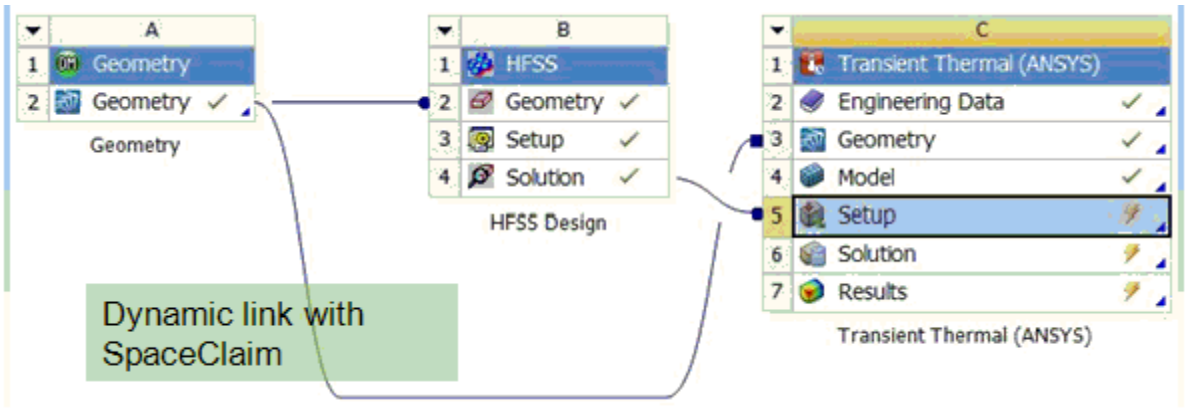


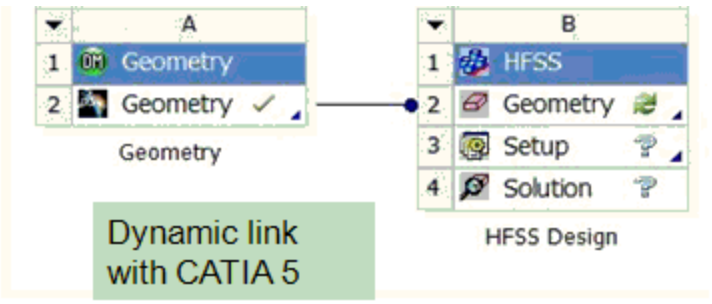
CAD Integration and Geometry Sharing

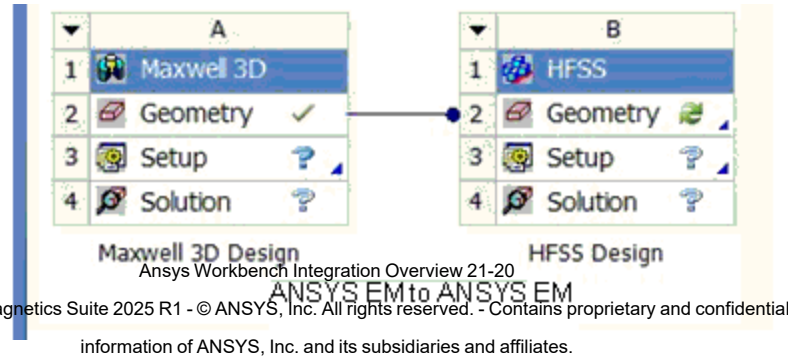
A CAD model comes into Ansys Electromagnetics as a User Defined Model (UDM).

The inputs to Ansys Electromagnetics from CAD are:

- Geometry/Topology with persistent IDs
- CAD parameters
- Material assignment
- Attributes, such as name and color

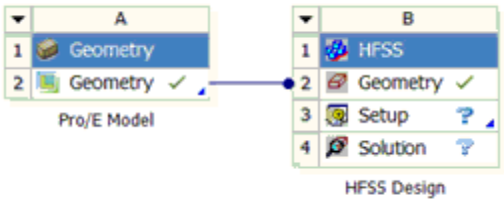




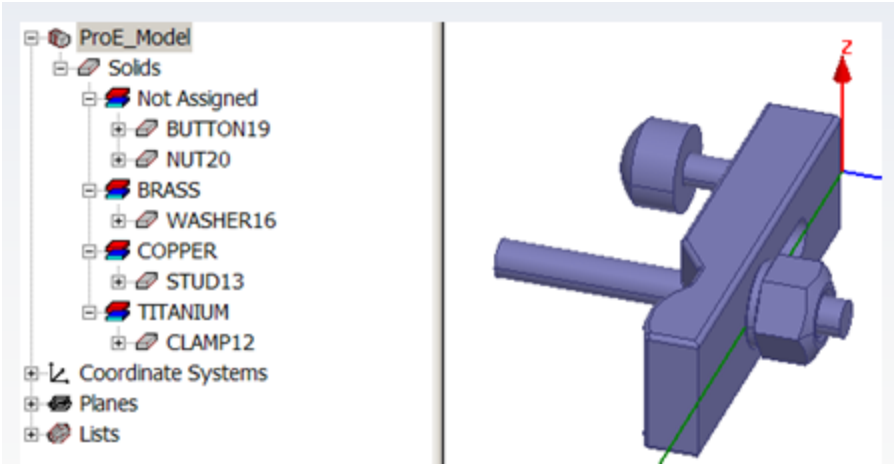


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in Workbench, a Pro/E Model can be linked to Ansys.



The geometry can then be viewed in HFSS as a UDM.



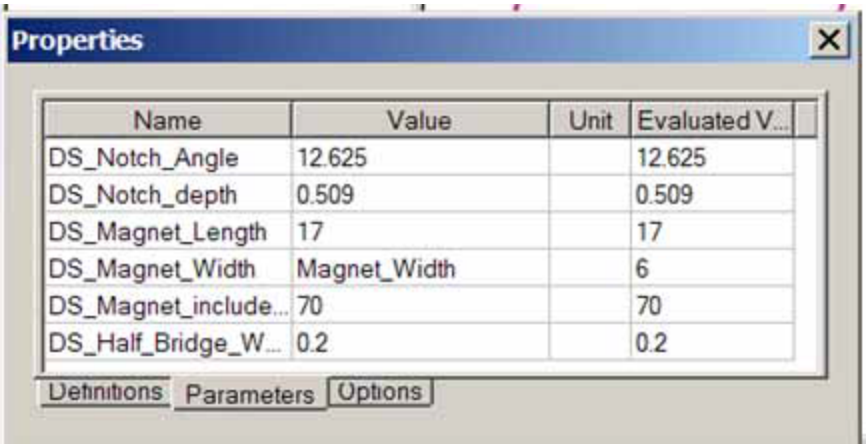
The CAD or WB model parameters appear in the Workbench:

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	DM Geometry (A1)			
4	P5	DS_Notch_Angle	12.625	
5	P6	DS_Notch_depth	0.509	
6	P7	DS_Magnet_Length	17	
7	P8	DS_Magnet_Width	6	
8	P9	DS_Magnet_included_Angle	70	
9	P10	DS_Half_Bridge_Width	0.2	
*	New input parameter	New name	New expression	

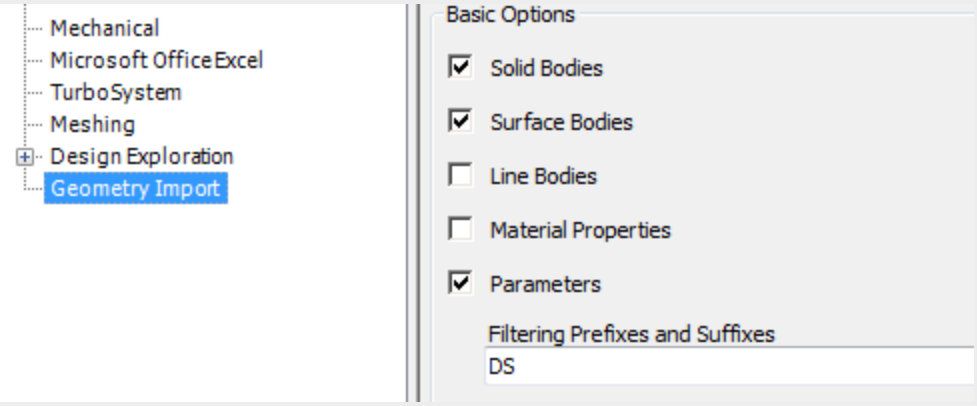
Through the Ansys Electromagnetics CAD integration, the linked UDM includes the same parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



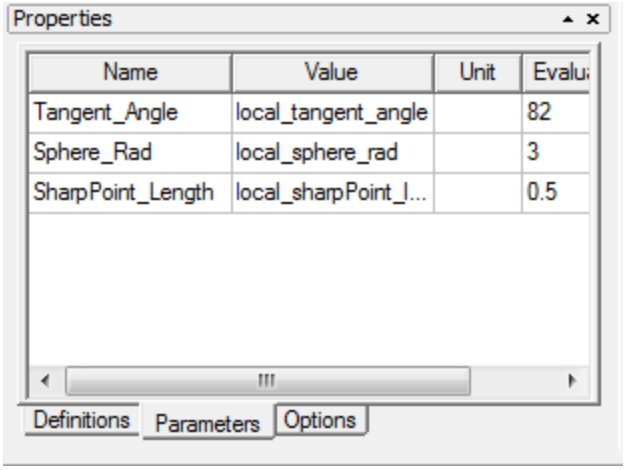
Note:

The parameters shown in these examples all have a DS prefix. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Once you import a geometry with parameters to an Electromagnetics application in the desktop, you need to map to local variables. In the target Electromagnetics application, select the geometry associated with the parameters. Update the geometry and view the **Parameters** tab in the **Properties** window. The **Value** column shows the values of the imported parameters. Type names for local variables in the **Name** column.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have local variables for the imported Design Modeler parameters, you are ready to use Design Xplorer for multiple design parameters.

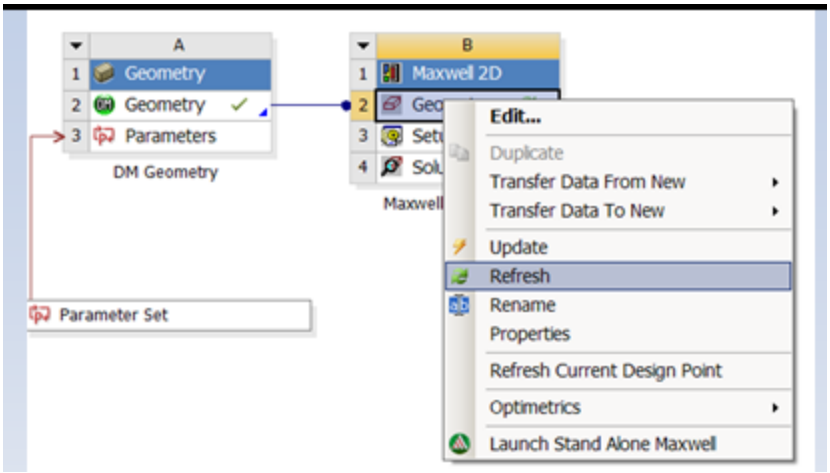
Table of Schematic D2: Design of Experiments (Custom)

	A	B	C
1	Name	P3 - Tangent_Angle (degree)	P4 - Sphere_Rad (in)
2	1	80	3
3	2	75	3
4	3	85	3
5	4	80	2.7

Bi-Directional CAD Integration

Ansys Electromagnetics CAD Integrations uses **Refresh** or **Generate** to pass updates.

You make an edit in a CAD application and run **Refresh** on an Ansys Electromagnetics Geometry Cell.

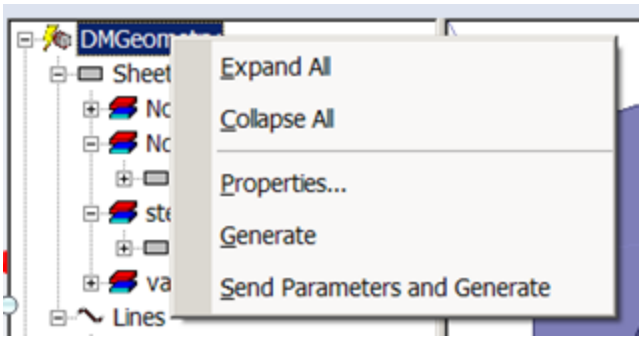


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refresh pulls the current state of CAD model (geometry, parameters, materials etc) and updates the corresponding data in the Ansys Electromagnetics application.

Alternatively, you can run **Generate** on the UDM in the Ansys Electromagnetics window.

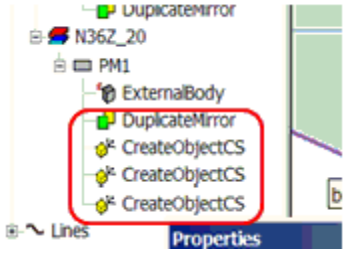
If you edit UDM (CAD) parameters in the Ansys Electromagnetics modeler window you can run the **Send Parameters and Generate** command.



The command passes the edited parameters to the linked CAD application and then pulls corresponding CAD geometry.

CAD Integration Model Edits

Several modeling operations are allowed on a CAD model in the Ansys Electromagnetics Modeler window. Operations are included in the History tree and retained during model **Refresh**.

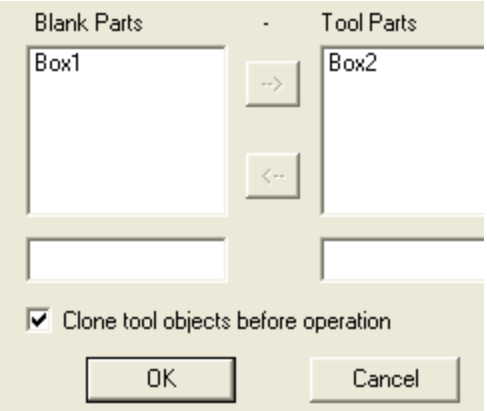


The following operations are not allowed:

- Non-history tree operations like Heal or defeature.
- Operations that use UDM parts as tools, such as Sweep

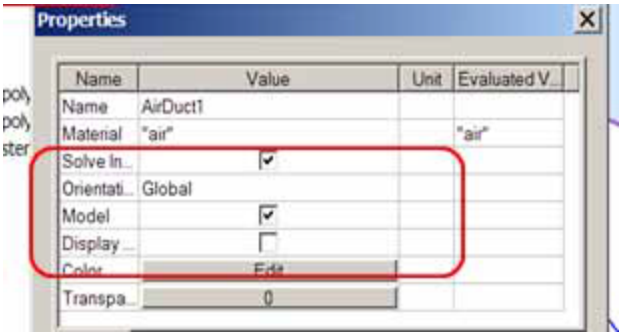
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boolean operations like Split or Unite (allowed when you select the clone tool option)



The following part attributes can be modified for UDM parts:

- Model/Non Model flag
- Solve Inside flag
- Part orientation
- Color
- Display Wireframe



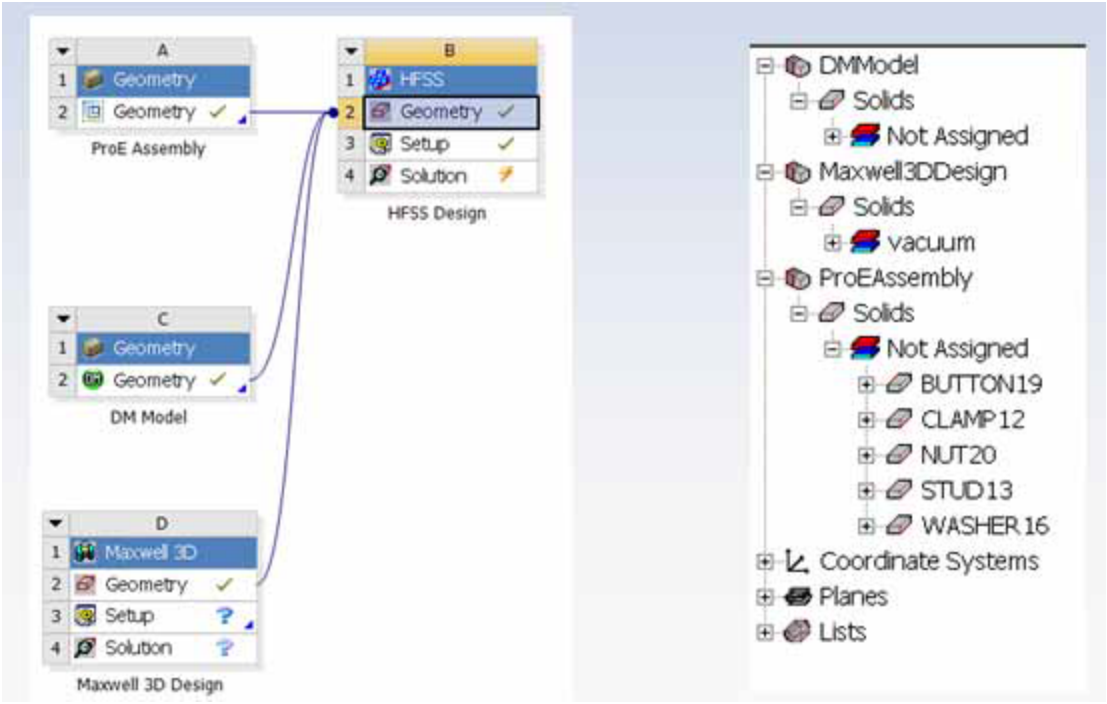
It is not possible to delete individual parts of UDM.

Multiple Geometry Links for CAD Integration

With CAD Integration you can consume geometry from multiple upstream sources. The source can be any of supported CAD or Ansys Electromagnetics products. This creates a UDM for each geometry source.

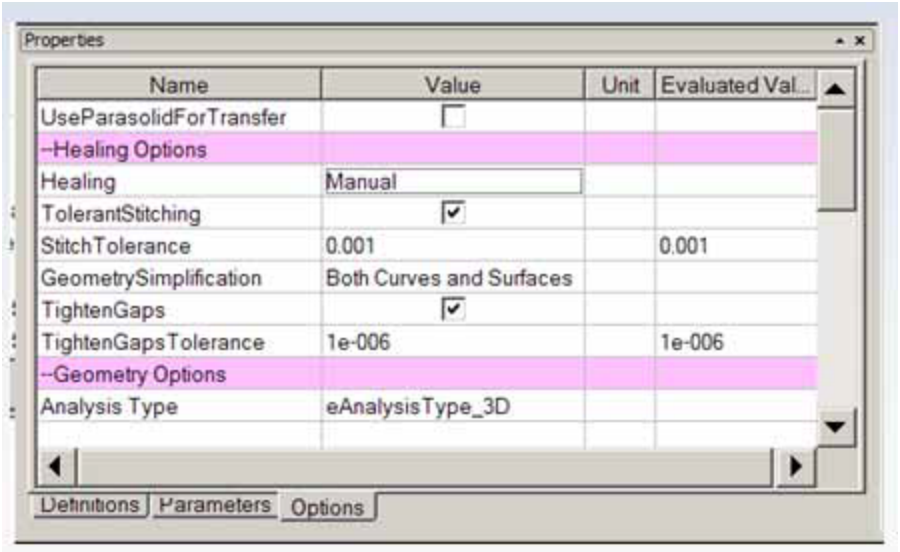
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, the following figure shows a DM Model, a Maxwell model, and ProE model linked to HFSS in Workbench, and displayed in the HFSS History tree as three UDMs.



Healing with CAD Integration

It is not possible to use the Heal command in Ansys Electromagnetics Modeler. Instead similar healing options are available under UDM properties option tab.

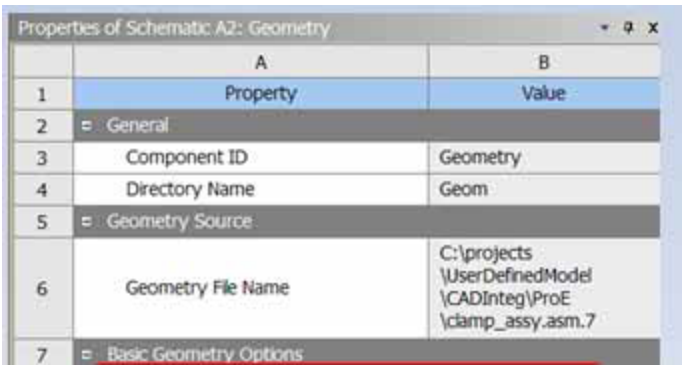


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

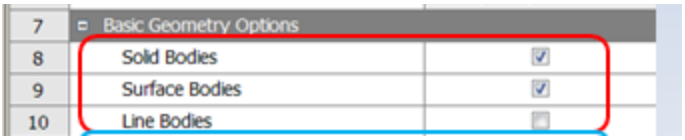
Healing options are None, Auto and Manual. By default, healing is off (None) and should be turned on only if required.

Important Geometry Options for CAD Integration

Select a Geometry Cell in Workbench to see options in the Properties window.

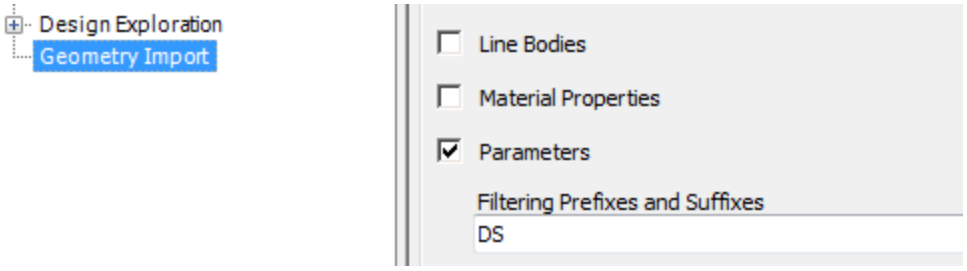


Control dimension of bodies coming from CAD



Make sure **Parameters** is checked and that the parameter key (filter) is appropriate for CAD parameters.

The default parameter key filtering means that only parameters whose names start with DS will come through. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



The **Attribute Key** should be empty or "Color" to bring in CAD colors.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

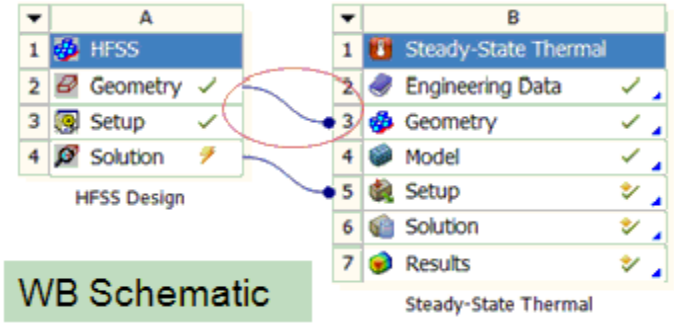
Material Properties must be checked to bring in the material assignment.

The **Mixed Import Resolution** option is used to resolve parts with mixed dimension (typically from Pro/E)

Refer to Ansys Help for details.

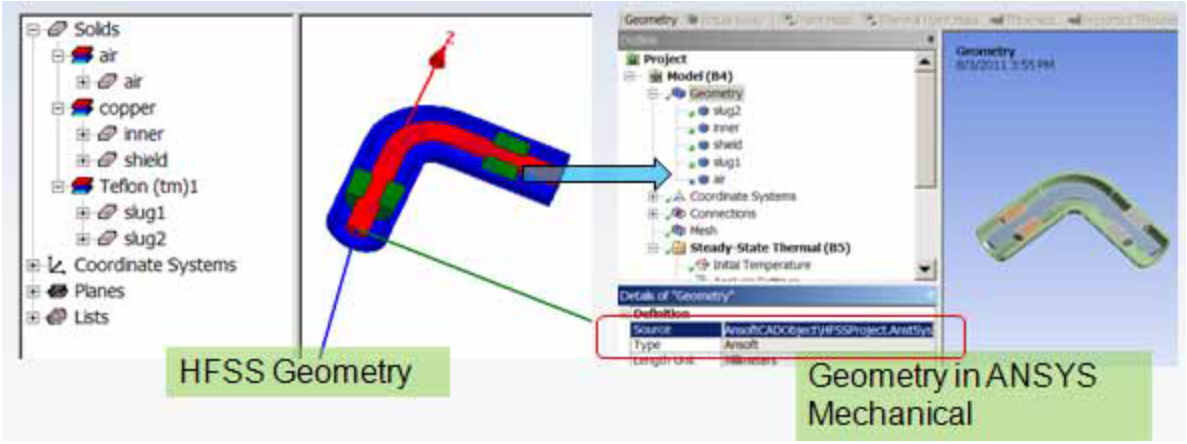
Ansys EM to Ansys Geometry Transfer

Ansys Electromagnetics CAD Integration transfers model information based on Workbench links.



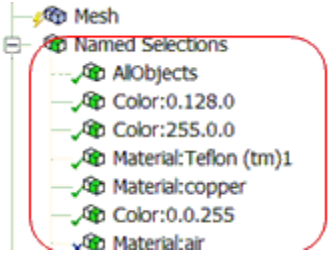
The following figure shows how the information is transferred between simulators.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

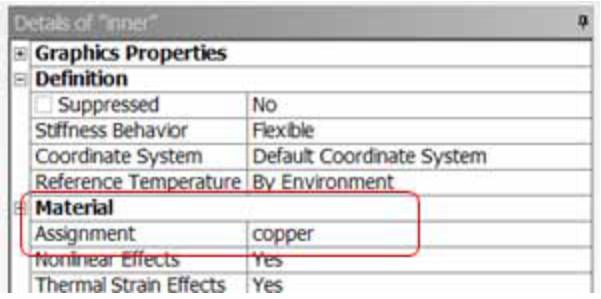


Information transferred includes:

- Geometry
- Ansys Electromagnetics lists and material assignment as Named Selection



- Material assignment



The CAD Integration geometry link is dynamic (you can get updated geometry from Ansys Electromagnetics) and associative (IDs persist between Ansys Electromagnetics model and Ansys model during model refresh).

Boundary conditions in Ansys are preserved. Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

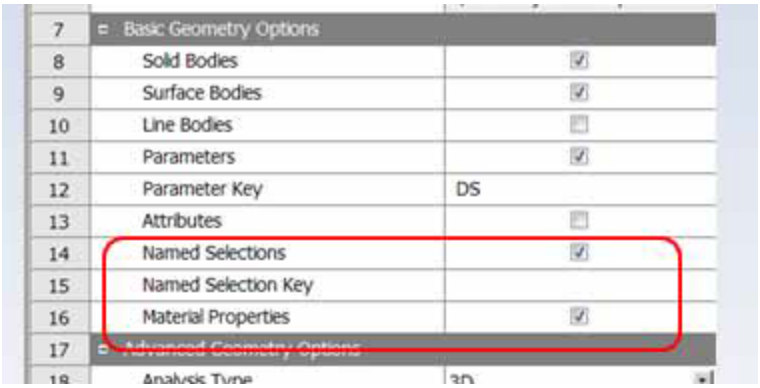
Material Assignment Transfer

There are two prerequisites for material assignment transfer:

- Engineering Data should have materials used in Ansys Electromagnetics model and material names should match (including case)
- Material Properties in Geometry Options must be checked

Before transferring as a named selection, **Named Selections** must be checked in **Basic Geometry Options**.

The Named Selection Key should either be empty or contain "Material."

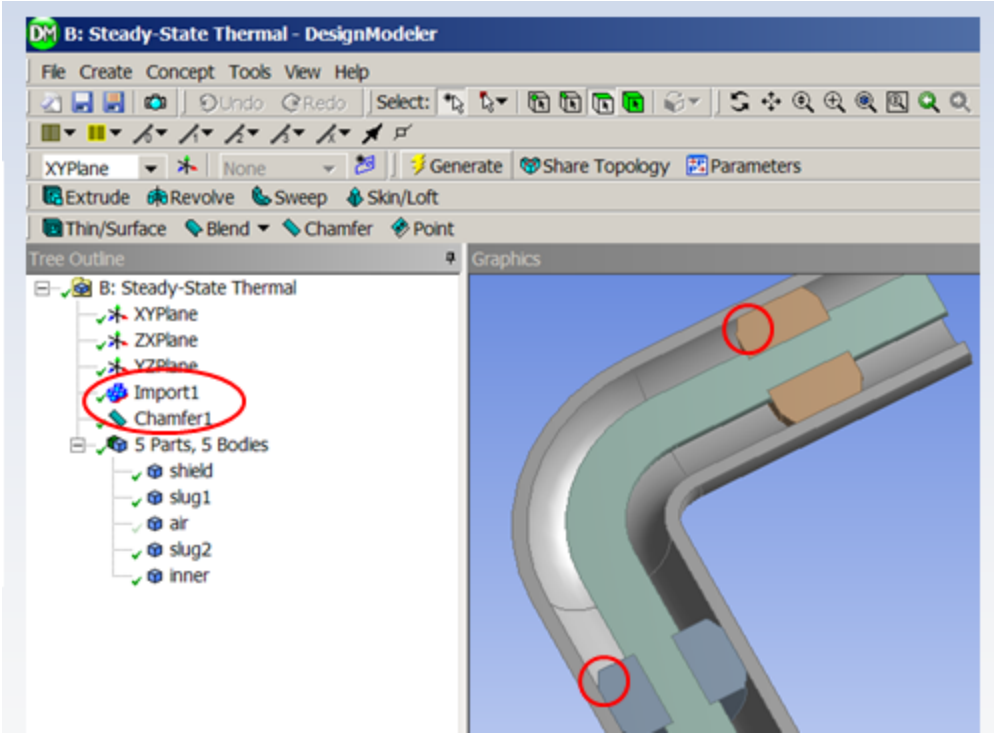


Geometry Transfer through Ansys DesignModeler (DM)

It is possible to edit geometry in Ansys DesignModeler (DM) before consuming it in Mechanical. This is useful when geometry requires pre-processing for Ansys simulation.

Consider an HFSS model linked to DM through the Workbench. In the following example, a chamfer operation on geometries is being imported.

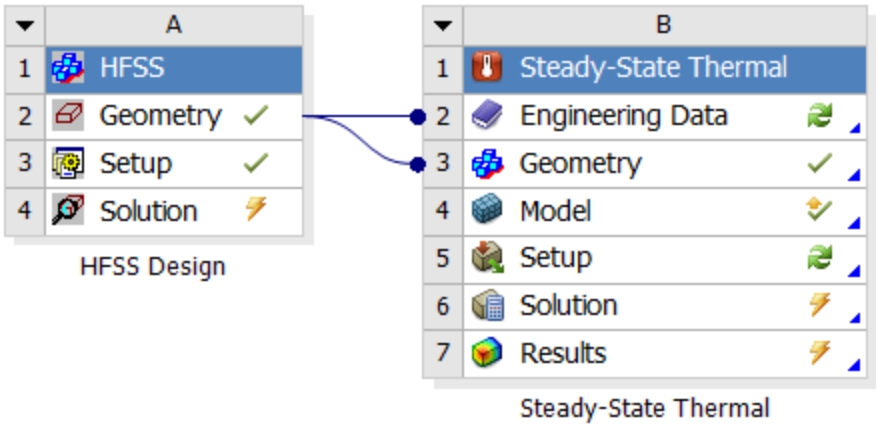
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Workbench Material Data Transfer

A link can be created from an Ansys Electromagnetics Desktop system (e.g., HFSS) geometry cell to a downstream Engineering Data (ED) cell. When the downstream ED cell is refreshed, it updates with materials (names and properties) used by the upstream project. This ensures that the geometry-to-geometry link connecting the same two systems, which specifies the material assignments for various parts, refers to the correct materials and their properties.

Detailed Behavior



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The geometry-to-ED link can be created by dragging the Ansys Electromagnetics Desktop system geometry cell to the ED cell of a downstream system. When the downstream system's ED cell refreshes, the upstream system generates a MatML format XML file, which is consumed by Workbench to update the downstream system. This file is stored in the workbench project files according to design point. Note that Workbench ED cell material names are case-sensitive, as are material names within Ansys Electromagnetics Desktop products.

For material assignments to work with the ED in the downstream model, the downstream geometry cell must have the **Material Properties** check box selected. A convenient way to do this is to have this property checked by default, by setting this in **Tools > Options > Geometry Import**. Systems created after this default option is set will have **Material Properties** automatically checked for the geometry cell.

When the downstream model cell is refreshed or updated, material properties from the ED in the same system are used in the model's materials, based on material assignments set during the geometry cell refresh. Note that if material assignment fails, the Workbench default material (Structural Steel) is used.

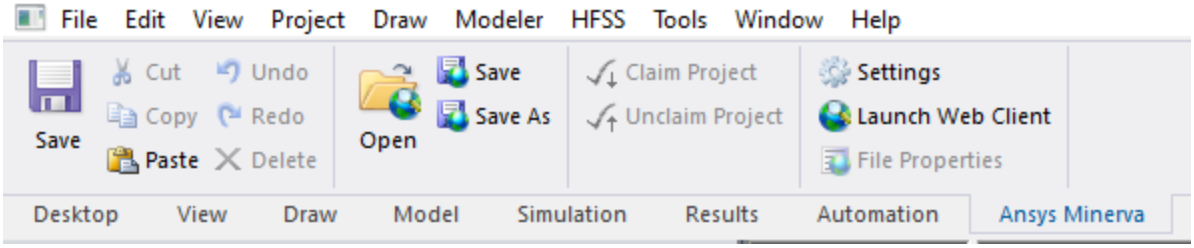
Workbench resolves name conflicts by changing the name of any materials being transferred over from the upstream system. For example, if the upstream Ansys Electromagnetics Desktop system uses a material named "Structural Steel", this will conflict with the existing downstream default material of the same name. The update will result in the Ansys Electromagnetics Desktop material being named "Structural Steel 2" in the downstream ED, but the downstream model will still use the properties supplied by Ansys Electromagnetics Desktop for the material.

Note that some materials may not satisfy all requirements for downstream system physics. For example, vacuum has no thermal conductivity, but downstream steady state thermal systems require a non-zero value for this property. Material properties in the workbench ED cell of the downstream system can be modified after refresh to ensure that validity criteria can be met. Any property added to a material in the downstream ED is preserved in subsequent ED cell updates. However, any existing property that is edited in the downstream ED will be overwritten by a subsequent ED cell update.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

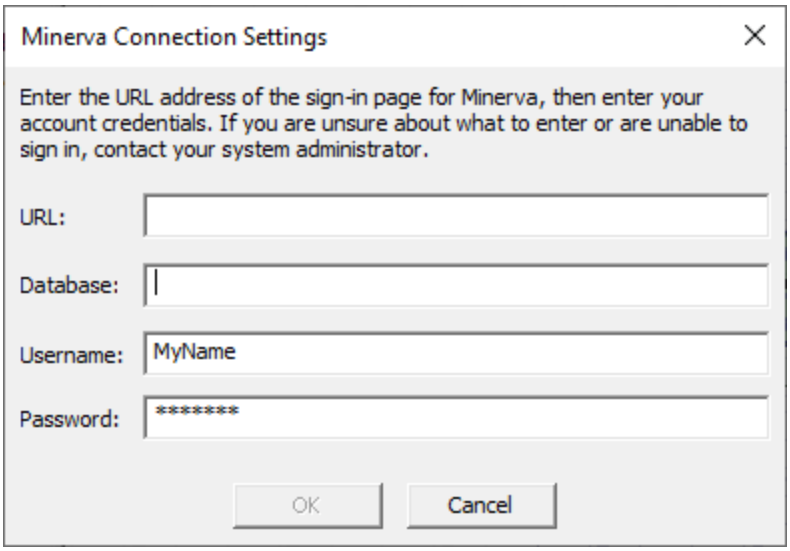
22 - Minerva Remote Storage Environment

Ansys Minerva is a remote storage environment where you can store Ansys project archive files. You can collaborate with other users by downloading a project, making changes, and then uploading the project. One of the features of Ansys Minerva is the ability to store and retrieve Ansys projects from the Minerva server. This ability has been integrated into Electronics Desktop so it is now possible to download and open a project stored on Minerva directly from the Electronics Desktop interface. In the same way, it is also possible to save a project to Minerva. The image below shows the Minerva ribbon containing Minerva commands accessible through the desktop.



Minerva has its own HTML web interface with additional features as a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos.

Minerva Settings

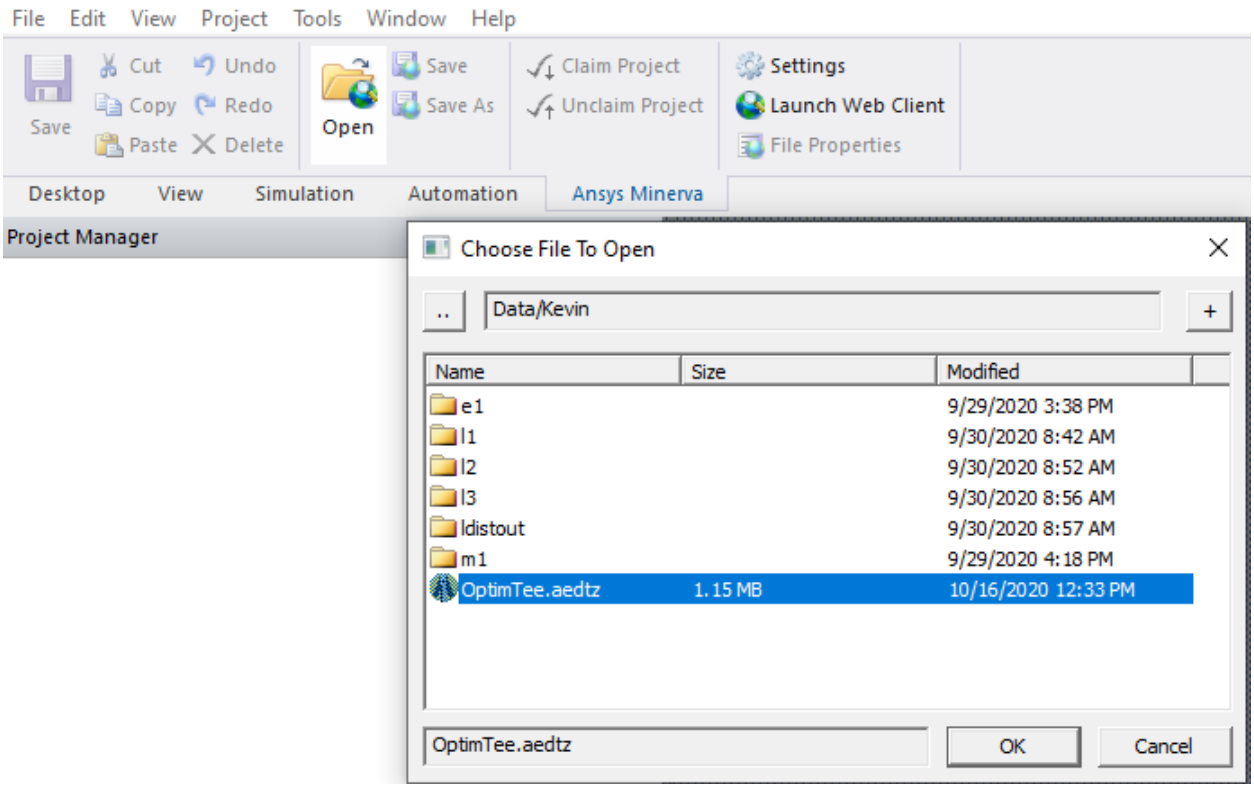


When you first login via the desktop, or to the Web Client, you must provide a Minerva URL, a database name, a user name, and a password. You will need to obtain these from your system

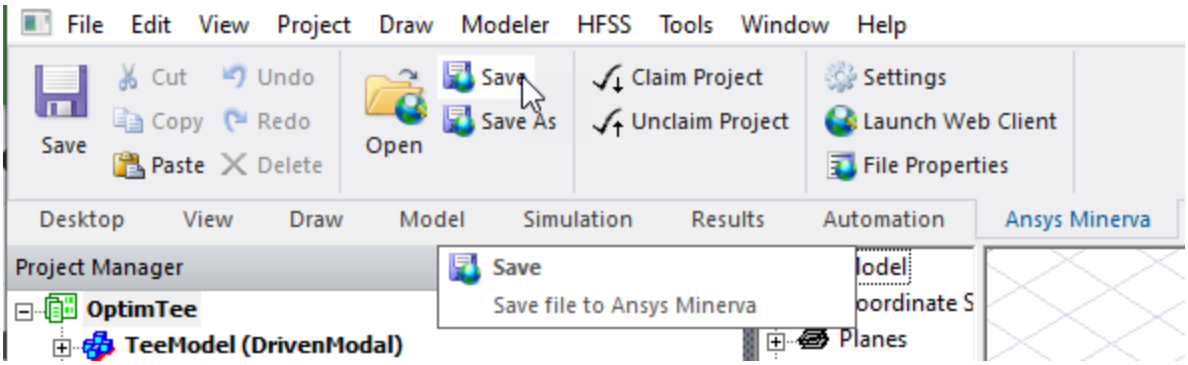
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

administrator. Once you provide the connection settings, the other Minvera commands are enabled.

Open : opens the Choose File to Open browser that lets you select and download an archive file, extract it, and open it. The [..] button moves up one level in the file hierarchy. The [+] button creates a new folder at the current level.

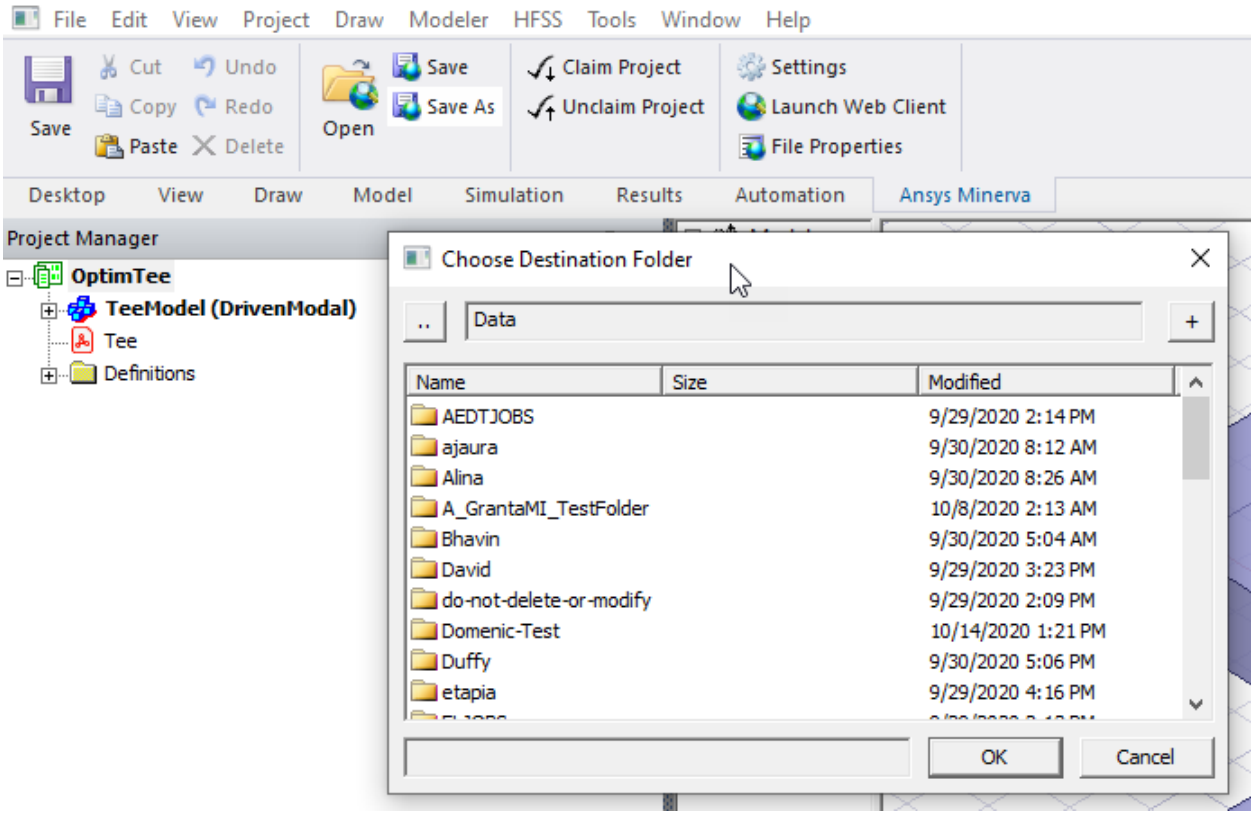


Save : archives the current project, and saves it to a remote Minerva system.



Save As... prompt for remote file, archive project, save to remote Minerva system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In order to prevent users from overwriting each other's work, Minerva has added the concept of claiming and unclaiming files. A file claimed by one user cannot be overwritten on the Minerva server by another user. Unclaimed files can be overwritten by anyone at any time. Once a file has been claimed by one user, it cannot be claimed by a different user until it has been unclaimed by the original claimer. Note: even if a project is claimed by one user, other users are free to download, open and edit the file locally.

The recommended flow when working on a project is:

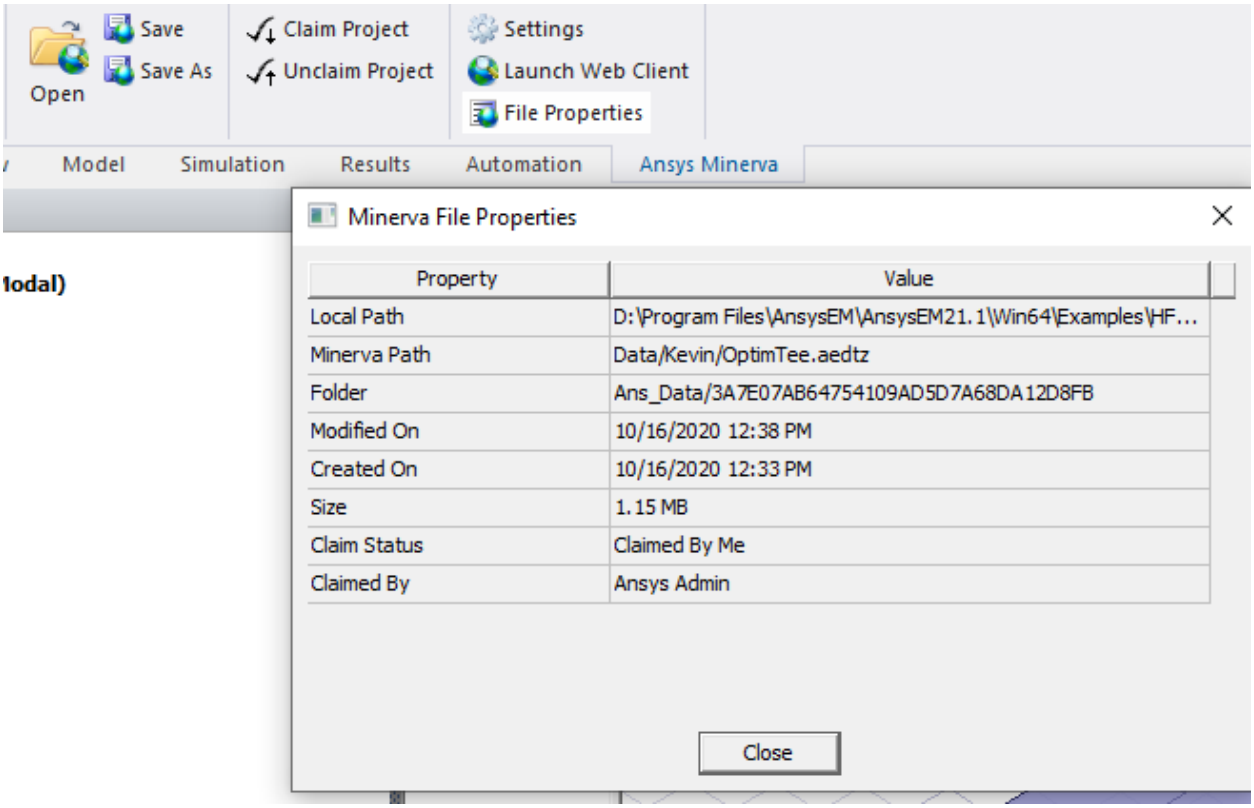
1. Claim the file, make sure you are working with the most recent version
2. Make edits
3. Save the file to Minerva
4. Unclaim the file

Claim Project: claims file so it cannot be modified by other users.

Unclaim Project: releases claimed file.

File properties: opens a dialog letting you view remote file properties. Notice the Claim Status and Claimed By properties.

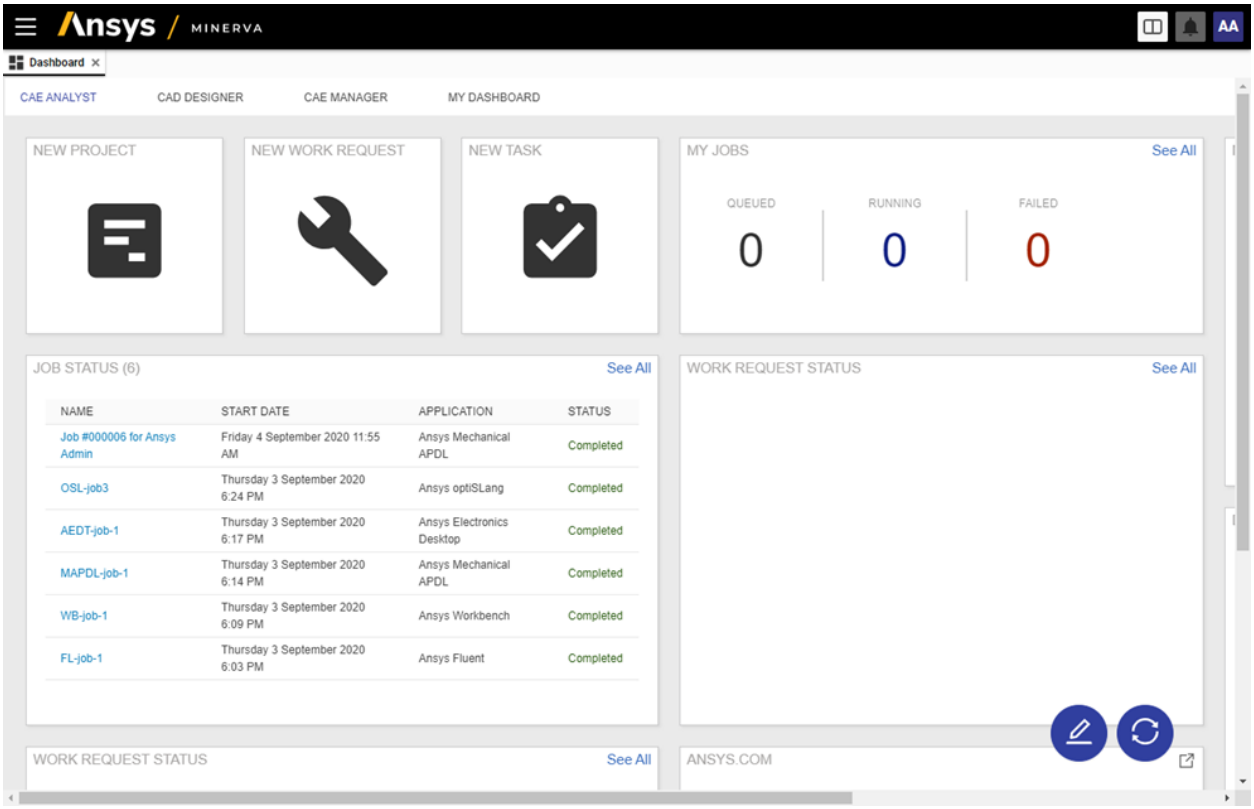
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Launch Web Client:

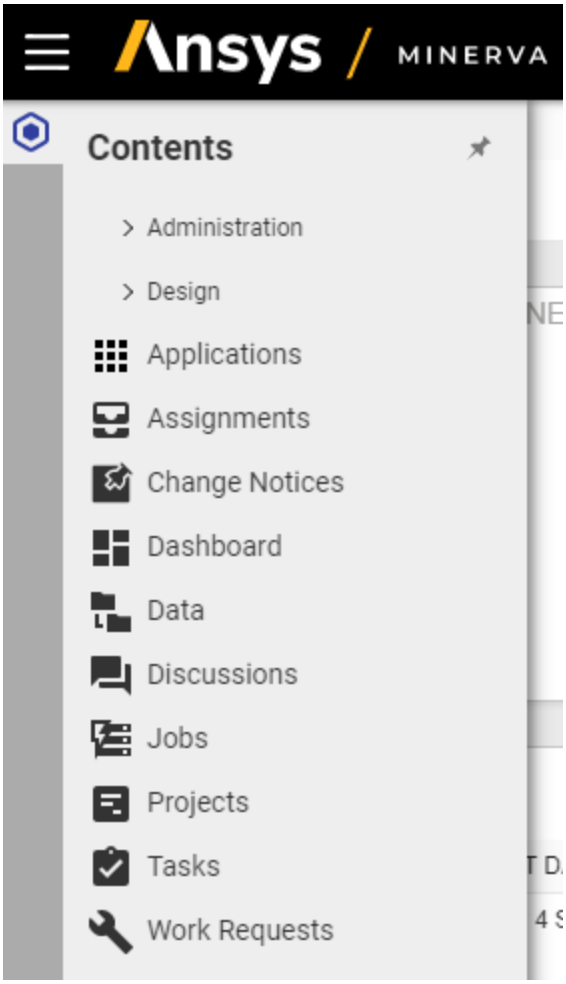
Ansys Minerva, powered by Aras, is a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos. Launching the Minerva Web Client gives access to these features. This opens a dialog calling for you to specify provide login information. It attempts to open a web browser window, so your default browser must be compliant.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



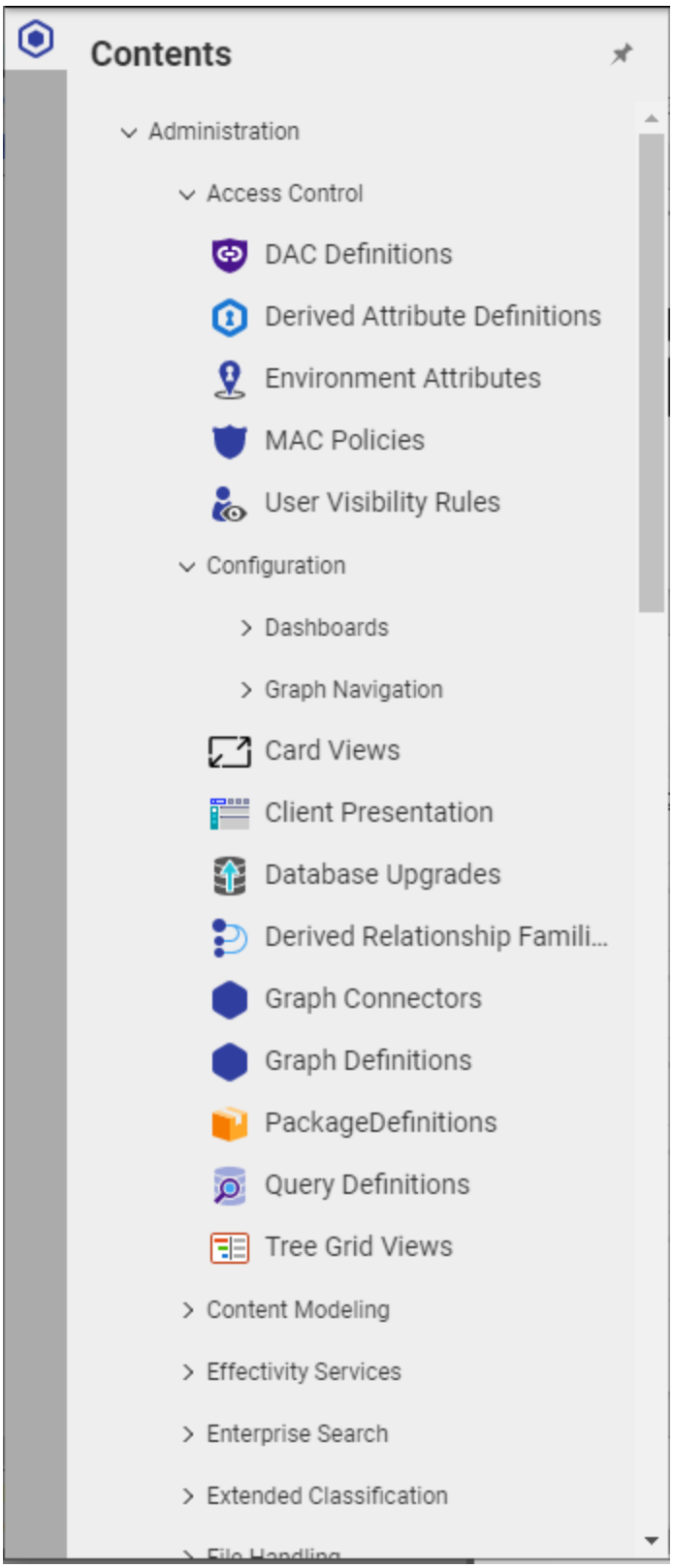
Clicking on the triple bar icon left of the Ansys logo opens up access to the more robust Minerva Web Client functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The angle icons for Administration open up further functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23 - Scripting

Information about creating, editing, and using IronPython scripts and VB scripts is in the product's scripting guide in the help. You can also access PDF versions of the scripting guide from the help system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.