

© 2026 Synopsys, Inc and/or ANSYS, Inc. All rights reserved.
Unauthorized use, distribution, or duplication is prohibited.

Icepak® 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 2026 R1
January 2026

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

Copyright and Trademark Information

© 1986-2026 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 2026 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 Icepak Help **1-1**

Icepak General Quick Links 1-1

Icepak User Interface Quick Links 1-2

Modeling Quick Links 1-3

Materials Quick Links 1-4

Meshing Quick Links 1-4

Analysis Quick Links 1-5

Optimetrics Quick Links 1-6

Results Quick Links 1-7

Icepak Getting Started Guides 1-7

Example Projects 1-11

Finding Information in the Online Help 1-12

Using the Search Function in the Help 1-13

Help Conventions 1-17

Getting Help from Ansys Technical Support 1-19

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 2-5

Icepak Student Limitations 2-5

System Requirements 2-5

Launching Ansys Electronics Desktop 2-5

Windows: 2-6

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High DPI Systems 2-7
Linux: 2-7
Launching AEDT from the command line: 2-7
Starting from GNOME desktop: 2-7
Ansys Product Improvement Program 2-8
Ansys Electronics Desktop Overview 2-11
Working with Ribbons 2-12
IC Mode Layout Ribbon 2-16
Customizing the Automation Tab 2-16
Creating a Toolkit Gallery 2-18
Ansys Electronics Desktop Windows 2-19
Showing and Hiding Windows 2-20
Auto Hiding Windows 2-21
Moving and Resizing Windows 2-22
Project Manager Window 2-24
Working with the Project Tree 2-25
Setting the Project Tree to Expand Automatically 2-26
Viewing Material Definitions 2-27
Viewing Ansys Electronics Desktop Design Details 2-28
Message Manager Window 2-29
Setting the Message Manager to Open Automatically 2-30
Showing New Messages 2-30
Automatically Expanding the Message Manager Tree 2-30
Action Messages 2-31
Clearing Messages 2-31
Hiding the Message Manager Window until Messages Appear 2-31
Properties Window 2-32
Showing and Hiding the Properties Window 2-33

Ansys Electromagnetics Suite 2026 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 2-34
Modifying Object Command Properties Using the Properties Window 2-34
Properties Window for Schematic and Layout Editors 2-35
General Tab 2-35
Parameters Tab 2-35
Icepak Tab 2-35
Visualization Tab 2-36
Closing the Properties Window 2-37
Properties Dialog Box 2-37
Opening the Properties Dialog Box 2-37
Setting the Properties Dialog Box to Open Automatically 2-38
General Tab 2-38
Icepak Tab 2-38
Parameters Tab 2-39
Visualization Tab 2-39
Progress Window 2-40
Stopping or Aborting Simulation Progress 2-41
ACT Extensions Window 2-41
Ansys Electronics Desktop Menus 2-46
Menu Bar 2-47
File Menu 2-48
Edit Menu 2-49
Edit Menu for 3D Modeler 2-49
Edit Menu for Report Window 2-52
View Menu 2-52
Basic View Menu 2-53
3D Modeler View Menu 2-54
Project Menu 2-60

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editor and Design Specific Menus 2-60
Layout Editor Draw Menu 2-61
Tools Menu 2-62
Understanding Registry Tools 2-65
Adding External Tools to the Tools Menu 2-69
Window Menu 2-71
Help Menu 2-74
Context-Sensitive Help 2-75
Obtaining Information about the Software and Release 2-75
Shortcut Menus 2-76
Shortcut Menu in the 3D Modeler Window 2-77
Shortcut Menus in the Project Manager Window 2-78
Customizing Ansys Electronics Desktop Menus 2-80
Ansys Electronics Desktop Design Area 2-90
3D Modeler Window 2-91
Report Window 2-91
Ansys Electronics Desktop History Tree 2-92
Organizing the History Tree 2-95
Controlling the View of Objects in the History Tree 2-96
Group Commands for Modeler Objects 2-96
Accessing Group Commands 2-98
Group Command Descriptions 2-98
Operations on Groups 2-99
Group Properties 2-100
Selecting Objects in the History Tree 2-101
Viewing and Editing Commands on History Tree Objects 2-104
Upgrading Version in the History Tree 2-105
Ansys Electronics Desktop Status Bar 2-106

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyboard Shortcuts 2-107
Desktop Shortcuts 2-107
3D Modeler Window Shortcuts 2-108
Report Shortcuts 2-109
Choosing the View Navigation Options 2-110
Custom Keyboard Shortcuts 2-112
Auto-Complete for Variables and Properties in Electronics Desktop 2-113
Using the Password Manager to Control Access to Resources 2-117
Running Ansys Electronics Desktop from a Command Line 2-118
Command-Line Syntax 2-118
Run Commands 2-119
Job Management from the Command Line 2-125
Run Command Examples 2-125
Specifying Project Files 2-126
Options 2-126
-Batchoptions 2-128
-Batchoptions Examples 2-128
Export Options Files 2-129
Examples and Further Explanations of -batchoptions Use 2-129
For -batchoptions Use: Project Directory and Lib Paths 2-131
For -batchoptions Use: TempDirectory 2-132
For -batchoptions Use: Various Desktop Settings 2-132
Batchoptions: Override Registry Entry 2-135
Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta 2-136
Command-line Syntax 2-136
Run Commands 2-137
Run Command Examples 2-141
Specifying Project Files 2-141

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options ... 2-142
-Batchoptions ... 2-143
-Batchoptions Examples ... 2-143
Export Options Files ... 2-144
Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta ... 2-144
Command-line Syntax ... 2-145
Run Commands ... 2-145
Run Command Examples ... 2-149
Specifying Project Files ... 2-150
Options ... 2-150
-Batchoptions ... 2-151
-Batchoptions Examples ... 2-152
Export Options Files ... 2-152
Running Ansys Electronics Desktop with a JSON File ... 2-152
JSON File Format ... 2-153
Running Ansys Electronics Desktop from a Windows Remote Terminal ... 2-155
Using Windows HPC Commands ... 2-156
Customizing Ansys Electronics Desktop with Ansys ACT ... 2-156
Installing PyAEDT (Beta) ... 2-157
Debug Logging ... 2-158
3 - Working with Ansys Electronics Desktop Projects ... 3-1
Ansys Electronics Desktop Files ... 3-1
Getting Started with Icepak ... 3-2
Setting up an Icepak Design ... 3-2
Inserting an Icepak Design ... 3-3
Selecting the Solution Type for Icepak ... 3-3
Setting Project Options ... 3-4
Setting General Options ... 3-5

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Options: Desktop Configuration 3-7
General Options: Project Options 3-9
General Options: Miscellaneous Options 3-10
General Options: User Interface Options 3-11
Choosing a Color Scheme 3-15
General Options: Directories Options 3-17
General Options: Desktop Performance 3-18
General Options: Default Units 3-21
General Options: Remote Analysis Options 3-22
General Options: Component Libraries Options 3-23
Setting Icepak Options 3-25
Setting 3D Modeler Options 3-26
3D Modeler Options: Drawing Options 3-27
3D Modeler Options: Operation Options 3-29
3D Modeler Options: Snap Options 3-30
3D Modeler Options: Display Options 3-31
Display Options: General Options 3-32
Display Options: Rendering Options 3-34
Display Options: History Tree Options 3-36
3D Modeler Options: Group Options 3-37
3D Modeler Options: CAD Link Options 3-37
Overriding CAD Link Settings for Individual Models 3-42
3D Modeler Options: Advanced Options 3-43
Setting Reporter Options 3-44
Reporter Options: Report Setup Options 3-45
Reporter Options: Fields Reporter Options 3-45
Reporter Options: Report2D Options 3-46
Report2D Options: Curve 3-47

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report2D Options: Axis 3-47
Report2D Options: Grid 3-48
Report2D Options: Header 3-48
Report2D Options: Note 3-48
Report2D Options: Legend 3-49
Report2D Options: Marker 3-49
Report2D Options: Marker Table 3-50
Report2D Options: X-Y Markers 3-50
Report2D Options: Stacked 3-52
Report2D Options: Digital 3-52
Report2D Options: General 3-52
Curve Tooltip 3-53
Clipboard Option 3-53
Format 3-53
Report2D Options: Table 3-53
Header Row 3-54
Format 3-54
Copy to Clipboard 3-54
Setting HPC and Analysis Options 3-54
Configurations Tab 3-55
Selecting an Available Configuration 3-56
Options Tab 3-57
Specifying the Remote Spawn Command (Linux) 3-61
Licensing Settings Tool 3-63
Setting Options via Configuration Files 3-66
Behavior Examples 3-67
Rules for Modifying Option Settings 3-67
Configuration File Locations 3-67

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Products with Multiple Desktop Versions 3-67
Table of Directories and Files 3-68
Setting or Removing Option Values in Configuration Files: UpdateRegistry Command 3-70
UpdateRegistry -Set Command 3-70
UpdateRegistry -Get Command 3-71
UpdateRegistry -GetKeys Command 3-71
UpdateRegistry -Delete Command 3-72
UpdateRegistry -FromFile Command 3-73
UpdateRegistry File Format 3-73
Example Uses for Export Options Features 3-74
Options that Apply to All Users 3-74
Example: Searching for a Registry Key Pathname 3-75
Example: Setting an Installation Default Value 3-75
Example: Setting a Host-Dependent Default Value 3-76
Example: Reverting from a User-Defined Option Value to the Administrator Default 3-76
User Options and the UpdateRegistry Tool 3-77
Example: Removing a Host-Dependent User Option Setting 3-78
Example: Getting a Value from a Specific Configuration File 3-78
Example: Getting a Value Using Precedence Rules 3-79
Example: Adding a Host-Independent User Option Setting 3-79
Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml 3-79
Example: Change a Newly Set Registry Key 3-79
Setting the Temporary Directory 3-80
Temporary Directory Configuration File Format 3-82
UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files 3-83

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Batchoptions Command Line Examples ... 3-84
Batchoptions File Format ... 3-84
Example -BatchOptions with -Remote (Windows) ... 3-85
Example -Batchsolve with -Machinelist (Windows) ... 3-86
Example -Batchsolve with -Machinelist (Linux) ... 3-86
Example -Batchsolve for Local (Windows) ... 3-87
Batch Options and Analysis Configurations in the Registry ... 3-88
Managing Projects and Designs ... 3-92
Design Settings ... 3-93
Design Settings for Icepak ... 3-93
Ambient Conditions Tab ... 3-94
Gravity Tab ... 3-95
Validations Tab ... 3-97
Export Settings Tab ... 3-98
Mesh Tab ... 3-100
Post Tab ... 3-102
Advanced Tab ... 3-104
Altitude Effects ... 3-105
Ideal Gas Law Effects ... 3-106
Joule Heating ... 3-107
Opening Projects ... 3-107
Opening Recent Projects ... 3-110
Opening Example Projects ... 3-111
Opening Icepak Classic Projects ... 3-112
Inactive Objects ... 3-113
Linux System Configuration ... 3-113
Accelerate Classic Icepak File Import ... 3-113
Export Classic Icepak File ... 3-113

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import a Classic Icepak File ... 3-113
Import Multiple Classic Icepak Files with Non-Graphical Batch Converter ... 3-114
Icepak Classic Block Conversion ... 3-115
Icepak Classic Network Conversion ... 3-116
Icepak Classic PCB Conversion ... 3-116
Icepak Classic Wall Conversion ... 3-117
Icepak Classic Plate Conversion ... 3-117
Icepak Classic Heatsink Conversion ... 3-118
Icepak Classic Fan Conversion ... 3-118
Fan Flow Direction ... 3-118
Icepak Classic Grille Conversion ... 3-120
Icepak Classic Source Conversion ... 3-120
Icepak Classic Enclosure Conversion ... 3-120
Icepak Classic Opening Conversion ... 3-120
Icepak Classic Resistance Conversion ... 3-121
Icepak Classic Cabinet Conversion ... 3-121
Icepak Classic Parametric Trials Conversion ... 3-121
Icepak Classic Project Conversion Limitations ... 3-121
Naming ... 3-122
Project Settings ... 3-122
Mesh Settings ... 3-122
Mesh Operation ... 3-123
Mesh Reuse Settings ... 3-123
Object Priority Settings ... 3-123
Solver Settings ... 3-124
High Performance Computing Settings ... 3-124
Transient Settings ... 3-124
Materials ... 3-124

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry ... 3-124
Icepak Classic Objects ... 3-125
Blocks ... 3-125
Networks ... 3-125
Walls ... 3-125
Plates ... 3-125
Resistances ... 3-125
PCBs ... 3-126
Packages ... 3-126
Heatsinks ... 3-126
Fans ... 3-126
Grilles ... 3-126
Sources ... 3-127
Openings ... 3-127
Chip Thermal Models ... 3-127
Assemblies ... 3-127
Monitors ... 3-127
Post-processing ... 3-127
ECAD ... 3-128
Creating Projects ... 3-128
Creating a Target Design from 3D Designs ... 3-130
Saving Projects ... 3-132
Path Name Length Issues for Windows ... 3-133
Changing Auto-Save Settings ... 3-134
Save Before Solving Option ... 3-135
Recovering Project Data from an Auto-Save File ... 3-135
Choosing a Preview Image ... 3-136
Importing and Exporting Projects and Data ... 3-138

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Files ... 3-139
Importing 2D Model (GDSII) Files ... 3-140
Creating GDSII Layer Mapping Files ... 3-143
Creating GDSII Layer Mapping Files ... 3-144
Tech File Colors with RGB Values ... 3-145
Importing 3D Model Files ... 3-159
Lightweight Geometry for Icepak Designs ... 3-162
Permitted Modeler Operations on Faceted Objects ... 3-164
Other Characteristics of Faceted Objects ... 3-164
Importing DXF and DWG Format Files ... 3-164
Importing Plot Data ... 3-167
Importing IDF Files ... 3-168
Overview of Importing IDF Files ... 3-168
Import an IDF File ... 3-168
Device Properties File Formats ... 3-176
Tab-Delimited (.tab) Files ... 3-177
Comma-Separated Value (.csv) Files ... 3-177
Importing IDX Files ... 3-178
Overview of Importing IDX Files ... 3-178
Import an IDX File ... 3-178
Importing Electronics Cooling XML Projects ... 3-180
Importing JEDEC JEPJPTD and JEP 30 Files ... 3-181
PTD IFF File Format ... 3-183
Units and Network Models ... 3-183
Resistances ... 3-184
JEP 30 File Format ... 3-185
Units ... 3-185
Network Models ... 3-186

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resistance and Capacitance ... 3-186
Importing Network CSV Files ... 3-187
Importing AEDTI Files ... 3-190
AEDTI File Object Conversion ... 3-191
Fluid Flow Boundary ... 3-191
Solid Thermal Boundary ... 3-191
Exporting Files ... 3-191
Exporting 2D Model Files ... 3-192
Exporting 3D Model Files ... 3-192
GDSII Files ... 3-193
Parasolid Files ... 3-194
State-Space Characterization ... 3-194
When to Use State-Space Characterization ... 3-195
Overview of State-Space Characterization ... 3-195
Run a Steady-State Analysis ... 3-195
Run a Transient Optimetrics Analysis ... 3-195
Export LTI ROM ... 3-197
Exporting Electronics Cooling XML Files ... 3-198
EC XML File Format ... 3-198
Region ... 3-199
Blower ... 3-199
Plate ... 3-199
Wall ... 3-200
Block (Box) ... 3-201
Block (Cylinder) ... 3-202
Source (3D Box) ... 3-203
Source (2D Rectangle) ... 3-204
Grille ... 3-205

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resistance ... 3-206
Opening ... 3-207
Network ... 3-207
PCB ... 3-208
Heat sink ... 3-209
Fan (2D) ... 3-210
Fan (3D) ... 3-211
Mesh Region ... 3-213
Material ... 3-214
Monitor Point ... 3-216
Exporting SIwave Temperature Files ... 3-216
Exporting JEDEC JEP 30 - PTD Files ... 3-218
Exporting Network CSV Files ... 3-219
Network CSV File Format ... 3-220
Creating and Exporting LPV ROM Files ... 3-221
Overview of LPV ROM Creation and Export ... 3-222
Create a Steady-State Analysis ... 3-222
Run a Transient Optimetrics Analysis ... 3-223
Export LPV ROM ... 3-226
Exporting Graphics Files ... 3-226
Exporting Options Files ... 3-228
Exporting Options Files Using the Desktop UI ... 3-228
Exporting Options Files Using a Script ... 3-228
Validating Projects ... 3-229
Modeler Validation Settings ... 3-230
Copying and Pasting a Project or Design ... 3-231
Renaming Projects ... 3-232
Archiving Projects ... 3-232

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Restoring an Archived Project ... 3-236
Deleting a Project or Design ... 3-239
Setting Read Only Designs ... 3-239
Updating Design Components ... 3-240
Undoing and Redoing Commands ... 3-241
Inserting a Documentation File ... 3-242
Working with Design Notes ... 3-242
Printing ... 3-243
Example Projects ... 3-244
Working with Variables ... 3-245
Working with Project Variables ... 3-246
Adding a Project Variable ... 3-246
Importing and Exporting Project Variables ... 3-249
Deleting a Project Variable ... 3-250
Editing a Project Variable ... 3-250
Adding a Project Variable Array ... 3-251
List of Intrinsic Variables ... 3-255
List of Constants ... 3-256
Working with Design Variables ... 3-257
Defining Local Variables at the Design Level ... 3-258
Adding a Design Variable Array ... 3-263
Fixed vs. Non-Fixed Variables ... 3-268
Non-Fixed Variable Sweeps ... 3-268
Orphaned Sweeps ... 3-269
Choosing a Variable to Optimize ... 3-270
Choosing a Variable to Tune ... 3-271
Including a Variable in a Sensitivity Analysis ... 3-272
Including a Variable in a Statistical Analysis ... 3-273

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying and Pasting a Variables List ... 3-274
Working with Datasets ... 3-275
Adding Datasets ... 3-275
Adding 3D Datasets ... 3-279
Importing Datasets ... 3-284
Editing Datasets ... 3-286
Cloning Datasets ... 3-288
Exporting Datasets ... 3-290
Removing Datasets ... 3-290
Changing Dataset Plot Properties ... 3-292
Using SheetScan ... 3-294
SheetScan Menus and Settings ... 3-294
SheetScan View Options ... 3-295
SheetScan Right-click Menu ... 3-296
SheetScan Settings ... 3-296
Loading a Datasheet Picture into SheetScan ... 3-296
Loading a Datasheet Picture Directly ... 3-297
Loading a Datasheet Picture Using the HTML Viewer ... 3-297
Defining a SheetScan Coordinate System ... 3-297
Defining a Characteristic Curve in SheetScan ... 3-299
Performing Operations on SheetScan Curves ... 3-300
Checking for Monotonicity in X ... 3-301
Importing Characteristic Data into SheetScan ... 3-301
Exporting SheetScan Data ... 3-303
Defining an Expression ... 3-304
Valid Operators for Expressions ... 3-304
Using Intrinsic Functions in Expressions ... 3-305
clp Formula ... 3-307

Ansys Electromagnetics Suite 2026 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 ... 3-307
Using a Piecewise Constant (pwc) Function ... 3-308
Using Dataset Expressions ... 3-310
Handling Delta Temperature Units in Expressions ... 3-312
Steady State Boundary Condition Thermal Properties Dependency ... 3-313
Defining Mathematical Functions ... 3-315
Modeling Quick Links ... 3-315
Toolkits ... 3-316
Icepak Toolkits ... 3-318
Icepak Geometry Toolkits ... 3-319
Approximate Circular Object as a Polygon ... 3-320
Approximate Cylinder with Plates ... 3-322
Approximate Cylinder with Polygonal Block ... 3-324
Create Polygonal Enclosure ... 3-326
Approximate 1/4 Cylinder with Polygonal Block ... 3-328
Create Adiabatic Enclosure with Thin Sides ... 3-330
Data Center Components - CRAC ... 3-332
Data Center Components - PDU ... 3-335
Data Center Components - Rack (Recirc) ... 3-338
Data Center Components - Tile ... 3-342
Specifying Varying Free Area Ratios ... 3-343
Heatsinks - AngledFinHS ... 3-345
Heatsinks - Heatsink Creation ... 3-347
Other - Create Network Representation Heat Exchanger ... 3-350
Packages - Cavity_Down_BGA ... 3-353
Packages - DUAL ... 3-360
Packages - Flipchip_BGA ... 3-364
Packages - FPBGA ... 3-370

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Packages - LCC ... 3-376
Packages - PBGA ... 3-384
Packages - QFN ... 3-391
Packages - QFP ... 3-396
Packages - SDBGA ... 3-401
Packages - SOT ... 3-407
Packages - Network Model - BGA_Network ... 3-412
Packages - Network Model - Delphi_3Node_Network ... 3-417
Packages - Network Model - Delphi_BGA_Network ... 3-422
Packages - Network Model - Delphi_QFP_Network ... 3-426
Packages - Network Model - GeneralNetwork ... 3-431
Packages - Network Model - General_3Node_Network ... 3-436
Printed Circuit Board (PCB) ... 3-441
Source From CSV ... 3-448
CSV File ... 3-451
Write Average Metal Fraction ... 3-451
Icepak Modeling Toolkit ... 3-454
Blower ... 3-454
Type 1 Blowers ... 3-455
Type 2 Blowers ... 3-459
DTM - Thermostat for Source Control ... 3-462
IC Packages - Extract Delphi Network ... 3-465
Extract Geometries Tab ... 3-465
External Faces Editor ... 3-467
Internal Faces Editor ... 3-470
Generate Data Tab ... 3-472
New Parametric Setup Editor ... 3-475
Predefined ... 3-475

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sample ... 3-477
Stratified LHS ... 3-480
Import from File ... 3-481
Optimize STM Tab ... 3-484
Node Selection ... 3-487
Gradient-Based Algorithm Settings ... 3-489
Differential Evolutionary Algorithm Settings ... 3-490
Training Results ... 3-490
IC Packages - JEDEC ... 3-490
IC Packages - Extract JB JC ... 3-493
LPV ROM Parametric Setup ... 3-496
LTI ROM Parametric Setup ... 3-498
R-Matrix ... 3-500
Combining R-Matrix Files ... 3-502
Solar Flux Calculator ... 3-503
Static ROM ... 3-505
STM Network Extraction ... 3-509
External Faces Editor ... 3-513
Active Setup Options ... 3-514
Configure Uniform Sampling Ranges ... 3-516
Solve the Steady-State Design for Training Data ... 3-517
Create the Steady-State Surrogate Thermal Model ... 3-518
Create a Transient Design ... 3-520
Solve the Transient Design for Training Data ... 3-522
Create the Transient Surrogate Thermal Model ... 3-522
Thermoelectric Cooler - Create TEC ... 3-523
Thermoelectric Cooler - Run TEC ... 3-525
Icepak Productivity Toolkit ... 3-527

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AEDT to Fluent ... 3-527
Contour File Export ... 3-529
Create Conduction Design ... 3-530
Supported and Unsupported Geometry ... 3-531
External Faces ... 3-531
Debug Divergence ... 3-531
Dimensionless Number Calculator ... 3-532
Debug Divergence ... 3-534
Fluent Mesh Import ... 3-535
HTC Coefficient Calculator ... 3-535
Randomize Object Colors ... 3-537
Snapping ... 3-538
Snap Faces ... 3-539
Align Face Centers ... 3-539
Morph Faces ... 3-540
Align Object Centers ... 3-541
Align and Morph Vertices ... 3-541
Align and Morph Edges ... 3-542
Zoom-In Modeling ... 3-543
Icepak Reporting Toolkits ... 3-544
Cut Plane ... 3-544
Write Detailed Report ... 3-549
Min/Max Locations ... 3-550
Power Budget ... 3-551
Variations ... 3-554
4 - Assigning Materials and Surface Materials ... 4-1
Materials and Surface Materials ... 4-6
Searching for Materials ... 4-6

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Searching by Material Property ... 4-10
Adding New Materials ... 4-12
Assigning Material Property Types ... 4-16
Thermal Properties for Solid Materials ... 4-17
Thermal Properties for Fluid Materials ... 4-18
Defining Anisotropic Tensors ... 4-19
Assigning Anisotropic Tensors ... 4-19
Defining Anisotropic Thermal Conductivity ... 4-20
Geometry Axis and Point of Origin ... 4-21
Limitations for Applying Cylindrical Thermal Conductivity ... 4-22
Surface Material Properties ... 4-22
Roughness ... 4-22
Emissivity ... 4-23
Solar Behavior ... 4-23
Solar Absorption ... 4-23
Solar Transmittance ... 4-24
Surface Diffuse Fraction ... 4-24
Defining Variable Material Properties ... 4-24
Specifying Thermal Modifiers ... 4-24
Defining Material Properties as Expressions ... 4-31
Defining Functional Material Properties ... 4-32
Assigning Materials from the Object Properties Window ... 4-32
Viewing and Modifying Material Attributes ... 4-33
Validating Materials ... 4-36
Copying Materials ... 4-36
Removing Materials ... 4-36
Exporting Materials to a Library ... 4-37
Sorting Materials ... 4-37

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filtering Materials ... 4-37
Working with Material Libraries ... 4-39
Editing Libraries ... 4-39
5 - Creating a Model ... **5-1**
Setting Units of Measurement for the Model ... 5-2
Drawing Objects ... 5-3
Drawing a Line Segment ... 5-5
Drawing a Three-Point Arc Line ... 5-10
Drawing a Center-Point Arc Line ... 5-12
Drawing a Spline ... 5-16
Drawing a Polyline ... 5-19
Assigning a Cross Section and Dimensions to a Polyline ... 5-21
Inserting Line Segments ... 5-23
Drawing an Equation-Based Curve ... 5-24
Functions, Operators and Quantities for the Edit Equation Dialog Box ... 5-28
Drawing a Circle ... 5-29
Drawing an Ellipse ... 5-32
Drawing a Rectangle ... 5-35
Drawing a Regular Polygon ... 5-38
Drawing an Equation-Based Surface ... 5-41
Drawing a Sphere ... 5-45
Drawing a Cylinder ... 5-48
Drawing a Box ... 5-53
Drawing a Regular Polyhedron ... 5-56
Drawing a Cone ... 5-59
Drawing a Torus ... 5-63
Drawing a Helix ... 5-66
Drawing a Segmented Helix with Polygon Cross-Section ... 5-69

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Segmented Helix with Rectangular Cross-Section ... 5-70
Drawing a Spiral ... 5-71
Drawing a Spiral Using User-Defined Primitives ... 5-73
Drawing a Bondwire ... 5-74
Drawing a Point ... 5-75
Drawing a Plane ... 5-76
Creating Segmented Geometry ... 5-77
Segmented Objects ... 5-78
Drawing Non-Model Objects ... 5-79
Selecting Non-Model Drawing Mode ... 5-80
Changing an Object to Non-Model ... 5-80
Drawing a Region ... 5-80
Drawing a SubRegion ... 5-83
Reassigning a SubRegion ... 5-88
Suppressing for Physics ... 5-91
Considerations for Suppressing Objects ... 5-93
Snapping Geometry ... 5-94
Snapping 3D and 2D Geometry ... 5-95
Snapping Faces ... 5-97
Snapping a Polyline Vertex ... 5-99
Snapping Fan Components ... 5-100
Group Commands for Modeler Objects ... 5-105
Accessing Group Commands ... 5-107
Group Command Descriptions ... 5-107
Operations on Groups ... 5-108
Group Properties ... 5-109
Prioritizing Objects ... 5-111
Model Analysis ... 5-113

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Options Dialog Box ... 5-114
Analyzing the Surface Mesh ... 5-115
Model Analysis Dialog Box ... 5-116
Objects Tab ... 5-117
Object Misalignment Tab ... 5-119
Surface Mesh (Single/Pairs) Tab ... 5-119
Mesh Feedback Tab ... 5-120
Simplify Command ... 5-125
Heal ... 5-133
Validation and Healing Options for Import File ... 5-135
Healing Non-Manifold Objects ... 5-135
Setting the Healing Options ... 5-137
Stitch Sheets ... 5-138
Align Selected Faces ... 5-139
Remove Selected Faces ... 5-140
Remove Selected Edges ... 5-141
Creating a User-Defined Primitive ... 5-141
User Customization through User-Defined Primitives (UDPs) ... 5-143
User-Defined Model (UDM) ... 5-143
UDM Compared to User-Defined Primitives ... 5-144
Insert UDM Command on Draw Menu ... 5-146
UDM - Icepak Heatsink ... 5-146
UDM Properties ... 5-148
UDM Parameters ... 5-149
UDM Part Edits ... 5-150
SpaceClaim Integration ... 5-150
3D Modeler CAD Link ... 5-151
Configuring Desktop for CAD Interfaces ... 5-152

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Link CAD Geometry to Ansys Electronics Desktop 5-154
Connecting to an Active CAD Session 5-158
Preserving IDs During Parametric Variations 5-160
Property Window of CAD-Link UDM 5-161
Archive a Project Containing CAD Link 5-163
Break Connection with CAD Link 5-164
3D Component Library 5-164
Creating a 3D Component from an Existing Model 5-165
Create 3D Component Info Tab 5-169
Create 3D Component: Model Tab 5-171
Create Component 3D Object Priority Tab 5-172
Create Component 2D Object Priority Tab 5-173
Create Component Thermal Tab 5-174
Create 3D Component Coordinate Systems Tab 5-175
Create 3D Component Parameters Tab 5-177
Non-Geometry Variables in 3D Components 5-178
Dependent Variables in 3D Components 5-178
Create 3D Component Encryption Tab 5-179
Set Passwords 5-182
Hiding Component Content 5-183
Inserting Encrypted Components 5-183
Editing Encrypted Components 5-183
Opening a 3D Component File 5-185
Copying and Pasting Encrypted Components 5-185
Create 3D Component Licensing Tab 5-185
Create 3D Component Image Tab 5-186
Save 3D Component File 5-189
Using the Component Libraries Window 5-190

Ansys Electromagnetics Suite 2026 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 5-193
Inserting a 3D Component into a Design 5-194
Viewing Properties of 3D Components 5-198
3D Components in the Project Manager Window 5-201
3D Components in the History Tree 5-202
Editing 3D Component Properties 5-203
Editing a 3D Component Definition 5-207
Verifying Updates to Components 5-210
Updating Definitions 5-210
Replacing 3D Components 5-211
Updating a 3D Component Definition 5-212
Components 2.0 (Beta) 5-213
Creating a 2.0 Component 5-214
Create Component 2.0 Representation Model Tab 5-217
Create Component 2.0 Representation Coordinate Systems Tab 5-219
Create Component 2.0 Representation Parameters Tab 5-221
Non-Geometry Variables in 2.0 Components 5-222
Dependent Variables in 2.0 Components 5-222
Create Component 2.0 Representation Encryption Tab 5-223
Set Passwords 5-226
Hiding Representation Content 5-227
Inserting Encrypted Representations 5-228
Copying and Pasting Encrypted Representations 5-228
Create Component 2.0 Representation Image Tab 5-229
Create Component 2.0 Representation Thermal Tab 5-231
Create 2.0 Component Representation Solar Loading Tab 5-232
Create Component 2.0 Representation Mesh Tabs 5-233
Component 2.0 Window 5-235

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Representations ... 5-237
Adding Security to a 2.0 Component ... 5-239
Info ... 5-242
Image ... 5-243
Editing a 2.0 Component ... 5-244
Inserting a Representation from a 2.0 Component ... 5-245
Layout Components [Beta] – Topic Index by Product ... 5-248
Modifying Objects ... 5-250
Object and History Editing ... 5-250
Modify Object Appearance ... 5-250
Modify Object Dimensions ... 5-250
Modify Object Location or Orientation ... 5-251
Surface or Edge Operations for Objects ... 5-251
Boolean Operations on Objects ... 5-251
Assigning Color to Objects ... 5-252
Setting the Default Color of Objects ... 5-252
Assigning Transparency to an Object ... 5-253
Setting the Default Transparency of Objects ... 5-253
Copying and Pasting Objects ... 5-254
Import a Model from the Clipboard ... 5-255
Copy Image ... 5-256
Deleting Objects ... 5-257
Deleting Start Points and Endpoints ... 5-257
Delete Last Operation ... 5-258
Cutting Objects ... 5-258
Moving Objects ... 5-259
Rotating Objects ... 5-260
Changing the Orientation of an Object ... 5-262

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mirroring Objects 5-262
Offsetting Objects 5-264
Duplicating Objects 5-265
Duplicating Objects Along a Line 5-266
Duplicating Objects Around an Axis 5-267
Duplicating and Mirroring Objects 5-269
Scaling Objects 5-269
Sweeping Objects 5-271
Sweeping Around an Axis 5-271
Sweeping Along a Vector 5-273
Sweeping Along a Path 5-275
Best Practices 5-275
Sweeping Faces Along Normal 5-280
Draft Type Options 5-280
Thicken Sheet 5-283
Wrap Sheet 5-284
Covering Lines 5-289
Covering Faces 5-289
Uncovering Faces 5-290
Detaching Faces 5-290
Detaching Edges 5-290
Creating a Cross-Section 5-291
Connecting Objects 5-291
Project Sheet Object 5-292
Moving Faces or Edges 5-296
Moving Faces Along the Normal 5-296
Moving Faces Along a Vector 5-297
Moving Edges Along Normal 5-299

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Uniting Objects ... 5-300
Subtracting Objects ... 5-300
Creating Objects from Intersections ... 5-302
Creating an Object from a Face ... 5-303
Creating an Object from an Edge ... 5-304
Splitting Objects ... 5-305
Separating Bodies ... 5-309
Converting Polyline Segments ... 5-309
Rounding the Edge of Objects (Fillet Command) ... 5-310
Flattening the Edge of Objects (Chamfer Command) ... 5-312
Imprinting an Object ... 5-316
Imprint Projection Commands ... 5-319
Purging Object History ... 5-322
Generate History to Reproduce Portions of Model ... 5-322
Selecting Items in the 3D Modeler Window ... 5-323
Selecting Objects ... 5-325
Selecting Several Objects ... 5-326
Selecting Objects by Name or Material ... 5-327
Selection by Object Name ... 5-327
Selection by Object Material ... 5-328
Select By Area ... 5-329
Setting the Default Color and Transparency of Selected Objects ... 5-331
Setting the Default Color of Highlighted Objects ... 5-332
Creating an Object Selection ... 5-332
Reassigning Objects to Another Object Selection ... 5-333
Using or Viewing Selection Objects ... 5-335
Selecting Faces ... 5-335
Selecting All Faces of an Object ... 5-337

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Faces by Name ... 5-338
Selecting Faces by Plane ... 5-339
Creating a Face Selection ... 5-340
Reassigning Faces to Another Face Selection ... 5-341
Selecting Edges ... 5-342
Selecting All Edges of an Object or Face ... 5-343
Selecting Vertices ... 5-344
Selecting Multi ... 5-346
Controlling the Selection in Multi Mode ... 5-347
Selecting Groups and Submodels ... 5-348
Clearing a Selection ... 5-348
Selecting the Face, Edge, Vertex, or Object Behind ... 5-349
Selecting Objects and Surfaces That Lie Inside Other Objects ... 5-349
Selecting Objects by Variable ... 5-352
Assigning Coordinates to New Objects ... 5-352
Setting the Reference Point ... 5-353
Defining Cartesian Coordinates ... 5-353
Defining Cylindrical Coordinates ... 5-354
Defining Spherical Coordinates ... 5-354
Using Absolute Coordinates ... 5-355
Using Relative Coordinates ... 5-356
Choosing the Movement Mode ... 5-356
Moving the Cursor In Plane ... 5-357
Moving the Cursor Out of Plane ... 5-359
Moving the Cursor in 3D Space ... 5-360
Moving the Cursor Along the X-Axis ... 5-361
Moving the Cursor Along the Y-Axis ... 5-362
Moving the Cursor Along the Z-Axis ... 5-363

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Measure Modes ... 5-364
Setting Coordinate Systems ... 5-368
Setting the Working Coordinate System ... 5-370
Creating a Relative Coordinate System ... 5-370
Creating an Offset Relative CS ... 5-371
Creating a Rotated Relative CS ... 5-373
Creating an Offset and Rotated Relative CS ... 5-375
Creating a Face Coordinate System ... 5-377
Automatically Creating Face Coordinate Systems ... 5-380
Creating an Object Coordinate System ... 5-380
Creating an Offset Object CS ... 5-381
Creating a Rotated Object CS ... 5-382
Creating an Object CS that is Both Offset and Rotated ... 5-383
Move CS to End Command for History Tree ... 5-384
Modifying Coordinate Systems ... 5-385
Expressing Cartesian, Cylindrical, or Spherical Coordinates ... 5-386
Deleting Coordinate Systems ... 5-387
Setting the Drawing Plane ... 5-387
Searching for Model Geometry ... 5-388
Cleaning Up Model History ... 5-389
6 - Modifying the 3D Model View ... 6-1
Change the View ... 6-1
Show or Hide Design Objects ... 6-2
Change Object Visualization ... 6-2
Change the Background ... 6-2
View Options: 3D UI Options ... 6-2
Panning the View ... 6-8
Zooming In and Out ... 6-9

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zoom In and Zoom Out Commands 6-11
Zoom Previous Command 6-12
Zooming into a Rectangular Area 6-14
Zooming in or out Using a Mouse Wheel 6-15
Zoom to a Selected Boundary or Excitation 6-15
Modifying the View Orientation 6-16
Apply an Orientation to the Current View 6-18
Using the Orientation Gadget 6-20
Controlling Visibility of the Orientation Gadget 6-24
Rotating or Spinning the View 6-25
Changing the Model View with Alt+Double-Click Areas 6-32
Applying a Default View Orientation 6-33
Applying a New View Orientation 6-35
Removing an Orientation 6-36
Fitting Objects in the View Window 6-37
Fitting All Objects in a View Window 6-38
Fitting Selected Objects in a View Window 6-38
Hiding Objects from View 6-39
Showing Objects 6-41
Showing Only Selected Objects in All or Active Views 6-44
Active View Visibility Dialog Box 6-46
Rendering Objects as Wireframes or Solids 6-47
Setting the Default View Rendering Mode 6-48
Enabling Optimized Rendering (Beta) 6-48
Setting the Enhanced Display of Material Color and Transparency 6-49
Setting the Curved Surface Visualization 6-51
Using Clip Planes 6-52
Modifying the Lighting 6-56

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Projection View ... 6-57
Setting the Background Color ... 6-58
Modifying the Coordinate System Axes View ... 6-61
Showing or Hiding the Axes ... 6-61
Show the Axes for Selected Objects ... 6-61
Enlarging or Shrinking the Axes ... 6-62
Showing or Hiding the Triad Axes ... 6-62
Choosing Grid Settings ... 6-62
Setting the Grid Type ... 6-64
Setting the Grid Style ... 6-65
Setting the Grid Density and Spacing ... 6-65
Setting the Grid's Visibility ... 6-67
Setting the Grid Plane ... 6-68
7 - Optimetrics ... 7-1
Parametric Overview ... 7-2
Setting Up a Parametric Analysis ... 7-2
Adding a Variable Sweep Definition ... 7-4
Specifying Variable Values for a Sweep Definition ... 7-4
Synchronizing Variable Sweep Definitions ... 7-5
Modifying a Variable Sweep Definition Manually ... 7-6
Overriding a Variable's Current Value in a Parametric Setup ... 7-7
Specifying a Solution Setup for a Parametric Setup ... 7-7
Specifying the Solution Quantity to Evaluate for Parametric Analysis ... 7-8
Setting up Calculations for Optimetrics ... 7-8
Specifying a Solution Quantity's Calculation Range ... 7-10
Viewing Results for Parametric Solution Quantities ... 7-10
Using Distributed Analysis ... 7-11
Adding a Parametric Sweep from a File ... 7-12

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Fast Optimetrics 7-13
Optimization Overview 7-14
Choosing an Optimizer 7-14
Legacy Optimizers 7-16
Quasi Newton (Gradient) 7-16
Pattern Search (Search-Based) 7-20
Sequential Nonlinear Programming (Gradient) 7-22
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) 7-23
Genetic Algorithm (Random Search) 7-24
MATLAB Optimizer 7-25
Multi-Objective Genetic Algorithm (MOGA) 7-30
MOGA Workflow 7-31
MOGA Steps 7-32
MOGA Steps to Generate a New Population 7-33
Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP 7-36
Mixed-Integer Sequential Quadratic Programming 7-38
Adaptive Multiple Objective Optimization 7-39
AMO Workflow 7-39
AMO Steps 7-40
Adaptive Single Objective Optimization 7-42
ASO Workflow 7-42
AMO Steps 7-44
Optimization Variables and the Design Space 7-45
Setting Up an Optimization Analysis 7-46
Setting Up the Quasi Newton (Gradient) Optimizer 7-47
Setting Up the Pattern Search (Search-based) Optimizer 7-49
Merit-Based Sequential Quadratic Programming(Gradient) Optimizer Setup 7-52
Setting Up the Sequential Nonlinear Programming (Gradient) Optimizer 7-53

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer 7-55
Setting Up the Genetic Algorithm (Random Search) Optimizer 7-57
Setting Up the MATLAB Optimizer 7-59
Screening (Search Based) Optimizer Setup 7-60
Setting Up the Multi-Objective Genetic Algorithm(Random Search) Optimizer 7-63
Setting Up the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer 7-69
Setting the Maximum Iterations for an Optimization Analysis 7-74
Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) 7-74
Adaptive Multiple-Objective (Random Search) Optimizer Setup 7-80
Adaptive Single-Objective (Gradient) Optimizer Setup 7-85
Cost Function 7-90
Acceptable Cost 7-91
Cost Function Noise 7-91
Adding a Cost Function 7-92
Specifying a Solution Quantity for a Cost Function Goal 7-94
Setting the Calculation Range of a Cost Function Goal 7-95
Setting a Goal Value 7-95
Specifying a Single Goal Value 7-96
Specifying an Expression as a Goal Value 7-96
Specifying a Variable-Dependent Goal Value 7-97
Goal Weight 7-97
Modifying the Starting Variable Value for Optimization 7-98
Setting the Min. and Max. Variable Values for Optimization 7-99
Edit Calculation Range 7-100
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 7-102
Changing the Min. and Max. Variable Values for Every Optimization Setup 7-103

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Size 7-103
Setting the Min. and Max. Step Sizes 7-104
Setting the Min and Max Focus 7-105
Equalizing the Influence of Different Optimization Variables 7-105
Setting the Min and Max Focus Values 7-106
Solving a Parametric Setup Before an Optimization 7-106
Solving a Parametric Setup During an Optimization 7-106
Automatically Updating a Variable's Value after Optimization 7-107
Changing the Cost Function Norm 7-107
Explanation of L1, L2, and Max Norms in Optimization 7-107
Example of a More Complex Cost Function 7-110
Advanced Genetic Algorithm Optimizer Options 7-111
Sensitivity Analysis Overview 7-114
Selecting a Primary Output 7-114
Setting Up a Sensitivity Analysis 7-114
Setting the Maximum Iterations Per Variable 7-116
Setting Up an Output Parameter 7-117
Specifying a Solution Quantity for an Output Parameter 7-117
Setting the Calculation Range of an Output Parameter 7-119
Modifying the Starting Variable Value for Sensitivity Analysis 7-119
Setting the Min. and Max. Variable Values 7-120
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 7-120
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 7-121
Setting the Initial Displacement 7-121
Solving a Parametric Setup before a Sensitivity Analysis 7-121
Solving a Parametric Setup during a Sensitivity Analysis 7-122
Performing Worst Case Analysis 7-122
Statistical Analysis Overview 7-126

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up a Statistical Analysis 7-127
Setting the Maximum Iterations for a Statistical Analysis 7-128
Specifying the Solution Quantity to Evaluate for Statistical Analysis 7-129
Setting the Solution Quantity's Calculation Range 7-130
Setting the Distribution Criteria 7-131
Overriding the Distribution Criteria for a Single Statistical Setup 7-131
Changing the Distribution Criteria for Every Statistical Setup 7-132
Statistical Cutoff Probability 7-133
Edit Distribution 7-134
Modifying the Starting Variable Value for Statistical Analysis 7-135
Solving a Parametric Setup During a Statistical Analysis 7-136
Using Design of Experiments 7-136
Setting Up Design of Experiments in Ansys Electronics Desktop 7-139
Add the Design of Experiments Setup 7-139
Design of Experiments Tab 7-143
Optimal Space Filling Design (OSF) 7-145
Central Composite Design (CCD) 7-147
Box Behnken Design (CCD) 7-148
Custom DOE Type 7-148
Latin Hypercube Sampling 7-149
Table Tab for Design of Experiments 7-150
Response Surface Tab for Design of Experiments 7-151
Variables Tab for Design of Experiments 7-153
Viewing Analysis Result for Design of Experiments 7-156
Min-Max Search View for Design of Experiments Result 7-159
Refinement Points Table 7-159
Performing a Manual Refinement 7-160
Response Points Table 7-160

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Response Curve 7-161
Response Curve (2D Slices) 7-164
Verification Points Table 7-166
Goodness of Fit (Predicted vs Observed Chart) 7-167
Response Surface Results 7-168
Local Sensitivity Charts 7-171
Local Sensitivity Curves 7-172
Tuning Overview 7-174
Tuning a Variable 7-174
Applying a Tuned State to a Design 7-177
Saving a Tuned State 7-177
Reverting to a Saved Tuned State 7-178
Resetting Variable Values after Tuning 7-178
Saving Field Solutions for Optimetrics Analyses 7-178
Saving Field Solutions for a Parametric Setup 7-179
Saving Field Solutions for an Optimization Setup 7-180
Saving Field Solutions for a Sensitivity Setup 7-181
Saving Field Solutions for a Statistical Setup 7-182
Saving Field Solutions for a Design of Experiments Setup 7-182
Copying Meshes in Optimetrics Sweeps 7-183
Adding an Expression in the Output Variables Window 7-184
Excluding a Variable from an Optimetrics Analysis 7-184
Modifying the Value of a Fixed Variable 7-185
Linear Constraints 7-186
Setting a Linear Constraint 7-186
Modifying a Linear Constraint 7-187
Deleting a Linear Constraint 7-188
Running an Optimetrics Analysis 7-188

Ansys Electromagnetics Suite 2026 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 7-188
Viewing Solution Data for an Optimetrics Design Variation 7-189
Viewing an Optimetrics Solution's Profile Data 7-189
Viewing Results for Parametric Solution Quantities 7-190
Plotting Solution Quantity Results vs. a Swept Variable 7-191
Viewing Cost Results for an Optimization Analysis 7-191
Plotting Cost Results for an Optimization Analysis 7-192
Viewing Output Parameter Results for a Sensitivity Analysis 7-192
Plotting Output Parameter Results for a Sensitivity Analysis 7-193
Viewing Distribution Results for a Statistical Analysis 7-193
Plotting Distribution Results for a Statistical Analysis 7-194
Link to DesignXplorer 7-194
8 - High Performance Computing 8-1
High Performance Computing in Icepak 8-1
Pre/post Processing Option 8-2
HPC Tasks 8-2
Meshing 8-2
Parametric Analysis 8-3
One Level Distribution 8-3
Two Level Distribution 8-4
COM Engine CPU Usage 8-4
Specifying Tasks and Cores for Job Schedulers 8-5
Specifying Tasks and Cores from the Command Line 8-6
Remote Analysis 8-6
Prerequisites for Remote and Distributed Analysis 8-7
Configuring the Local Machine to Solve Remotely 8-8
Remote Analysis Options 8-8
Running Remote Analysis 8-9

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting up Linux for Icepak ... 8-9
Troubleshooting ... 8-11
Remote Solve Node = Windows ... 8-13
Remote Solve Node = Linux ... 8-14
Distributed Analysis ... 8-15
Beta Feature: Parallel Component Mesh Adapt for 3D Component Array ... 8-15
Configure a Distributed Analysis ... 8-16
Editing Distributed Machine Configurations for Icepak ... 8-16
Test Machines ... 8-20
Job Distribution Tab ... 8-21
Example: ... 8-23
Adding Configurations or Accepting Edits ... 8-24
Selecting Optimal Configurations for Distributed Analysis ... 8-24
Two-Level Distribution Guidelines ... 8-24
Enabling GPU (Beta) ... 8-25
Large Scale DSO for Parametric Analysis ... 8-28
Prerequisites for Large Scale DSO ... 8-30
Job Management Interface for Large Scale DSO ... 8-31
Large Scale DSO Command Line Syntax ... 8-38
Large Scale DSO Job Outputs ... 8-42
Output Location and Organization ... 8-42
CSV File Contents ... 8-42
Postprocessing Large Scale DSO Dataset Solutions ... 8-43
Importing a Large Scale DSO Solution ... 8-43
Viewing Imported Solutions ... 8-44
Creating a Dataset Report ... 8-45
Cloning a Dataset Solution ... 8-45
Large Scale DSO Job Monitoring ... 8-45

Ansys Electromagnetics Suite 2026 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 8-46
Large Scale DSO Deployment/Configuration 8-46
Large Scale DSO Tutorial Example 8-48
Prepare the Model for Large Scale DSO Analysis 8-48
Submit the Large Scale DSO Job: Examples 8-53
Large Scale DSO Example: Post Process the Results 8-55
Large Scale DSO Known Issues/Troubleshooting 8-57
Node Order 8-57
Parallel Task Limitation for LS-DSO Parametric Variations 8-57
Job Restart 8-58
Linux-Only Issues 8-58
Interactive Scheduler Jobs 8-59
Specifying Options for Interactive Scheduler Jobs 8-59
Batchoptions for Interactive Scheduler Jobs 8-60
DSO Configuration for Interactive Scheduler Jobs 8-61
Analysis Configuration - Machines Tab 8-62
Analysis Configuration - Job Distribution Tab 8-63
Analysis Configuration - Options Tab 8-64
Design Type Options for Interactive Scheduler Jobs 8-65
High Performance Computing (HPC) Integration 8-66
Scheduler Terminology 8-66
What a Scheduler Does 8-67
Configuring Electronics Installation for HPC 8-68
Firewall Configuration 8-69
Installation Directory Examples 8-69
MPI Vendor and MPI Version 8-69
Integration with Microsoft Windows® HPC Scheduler 8-71
Submitting and Monitoring Ansys EM HPC Jobs 8-72

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Submitting and Monitoring Jobs for Windows HPC 8-74
Windows® HPC Job Templates 8-93
Selecting Computation Resource Units (Job Unit Type) 8-93
Windows® HPC Non-Exclusive Jobs 8-94
Windows® HPC Job Credentials 8-106
Integration with Platform's Load Sharing Facility (LSF) 8-107
LSF Job Management 8-107
LSF Job Submission Guidelines 8-108
Integration of Ansys EM Products with LSF 8-108
Installing Ansys EM Tools on LSF Cluster 8-109
Using the bsub Command to Submit Batch Jobs 8-111
bsub Arguments 8-112
Example LSF bsub Command Lines (Linux Only) 8-113
Serial Job 8-113
Serial Job Requiring a Minimum of 4GB 8-113
Multi-processing Job using 4 Cores 8-113
Distributed Processing Job using 4 Engines 8-113
Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing 8-114
Shell Script (~/projects/OptimTee.csh): 8-114
Monitoring LSF Batch Jobs 8-114
Terminating LSF Batch Jobs 8-115
LSF Known Issues and Workarounds 8-116
LSF Troubleshooting 8-116
Integration with PBS (Portable Batch System) 8-117
PBS Job Management 8-117
Non Standard Installations for PBS 8-118
PBS Limitations 8-118

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Submitting Ansys EM PBS Batch Jobs ... 8-120
Monitoring PBS Batch Jobs ... 8-121
qsub Arguments ... 8-122
Example PBS qsub Command Lines ... 8-123
Integrating Ansys EM Tools with Third-Party Schedulers ... 8-125
Introduction ... 8-125
Serial Jobs ... 8-126
Parallel Jobs ... 8-126
Common Requirements for Running Jobs ... 8-126
Using a Shared Library (Linux) or a DLL (Microsoft Windows) ... 8-127
Build Information for Scheduler Proxy Library ... 8-128
Implementation Details for Custom Scheduler Integration ... 8-128
IsProductLaunchedInYourEnvironment ... 8-129
GetTempDirectory ... 8-129
GetMachineListAvailableForDistribution ... 8-130
GetMessageStringToRegisterForSigTerm ... 8-131
LaunchProcess ... 8-132
GetUseRsmForEngineLaunch ... 8-133
GetThisJobID ... 8-134
GetSchedulerDisplayName ... 8-135
Using the Command Line to Submit HPC Jobs ... 8-136
Distributed Jobs ... 8-136
Serial Job on a Single Processor ... 8-136
Distributed Analysis and Multiprocessing in the Same Job ... 8-137
Scheduler Proxy Interfaces ... 8-137
Testing Scheduler Integration ... 8-143
Testing IsProductLaunchedInYourEnvironment ... 8-143
Testing GetSchedulerDisplayName and GetThisJobID ... 8-143

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Testing GetTempDirectory 8-144
Testing GetMachineListAvailableForDistribution 8-144
Testing LaunchProcess 8-144
Testing GetUseRsmForEngineLaunch 8-145
Troubleshooting Custom Scheduler Integration 8-145
None of the Proxy Functions are Called 8-145
Troubleshooting IsProductLaunchedInYourEnvironment Function 8-146
Troubleshooting GetSchedulerDisplayName 8-146
Troubleshooting GetThisJobID 8-146
Troubleshooting GetTempDirectory 8-147
Troubleshooting GetMachineListAvailableForDistribution 8-147
Troubleshooting LaunchProcess 8-147
Troubleshooting GetUseRsmForEngineLaunch 8-148
Using an IronPython Program for Integration with a Scheduler 8-148
GetName [IronPython] 8-150
GetDescription [IronPython] 8-150
IsProductLaunchedInYourEnvironment [IronPython] 8-151
GetSchedulerDisplayName [IronPython] 8-151
GetThisJobID [IronPython] 8-152
GetUseRsmForEngineLaunch [IronPython] 8-152
GetTempDirectory [IronPython] 8-153
GetMessageStringToRegisterForSigTerm [IronPython] 8-154
GetMachineListAvailableForDistribution [IronPython] 8-154
LaunchProcess [IronPython] 8-155
Selecting a Scheduler 8-156
Submitting a Job 8-158
Using Advanced Job Submission Options 8-163
Batchoptions 8-164

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment Variables ... 8-165
Batch Extract ... 8-166
Customize Job Submission ... 8-167
Using the Command Line to Submit HPC Jobs ... 8-167
Distributed Jobs ... 8-167
Serial Job on a Single Processor ... 8-167
Distributed Analysis and Multiprocessing in the Same Job ... 8-168
Job Import and Export ... 8-168
Scripting ... 8-168
Multi-Step Job Submission ... 8-169
Windows to Linux Job Submission ... 8-174
Prerequisites for Job Submission ... 8-175
Prerequisites for Job Monitoring ... 8-176
Supported Schedulers ... 8-176
Job Submission Scripting ... 8-182
Monitoring Jobs ... 8-183
Monitor Job Window ... 8-183
Monitoring Ansys Cloud Burst Jobs ... 8-186
Monitoring Large Scale DSO Jobs ... 8-187
Web Client for Batch Solve Monitoring and Reporting ... 8-187
Aborting an Analysis ... 8-193
Distribution Command Line Options ... 8-194
Integration with Ansys Remote Simulation Management (RSM) ... 8-196
Changing the AnsoftRSMService Listening Port ... 8-210
AnsoftRSMService Configuration ... 8-211
Ansoft Electromagnetics Desktop Configuration ... 8-212
Support for Open MPI on Linux [Beta] ... 8-214
Running HPC Diagnostics ... 8-219

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Site-Specific Diagnostics Job ... 8-222
Changing Solution Priority for System Resources ... 8-224
desktopproxy Utility to Abort or Cleanstop a BatchSolve ... 8-224
9 - Assigning Thermal Boundary Conditions ... 9-1
Temperature-Dependent Thermal Properties ... 9-2
Using Datasets to Define Temperature-Dependent Parameters ... 9-3
Defining a Dataset for Temperature-Dependent Parameters ... 9-5
3D Datasets for Thermal Properties ... 9-6
Joule Heating in Icepak ... 9-7
Joule Heating in Blocks ... 9-7
Joule Heating in Sources ... 9-9
Assigning Block Properties ... 9-10
Assigning Solid Block Properties ... 9-11
Assigning a Solid Block and Defining Thermal Properties ... 9-13
Assigning Hollow Block Properties ... 9-14
Assigning a Hollow Block and Defining Thermal Properties ... 9-16
Assigning Fluid Block Properties ... 9-17
Assigning a Fluid Block and Defining Thermal Properties ... 9-20
Moving Reference Frame (MRF) ... 9-22
Assigning Grille Properties ... 9-24
Modeling Vents in Icepak ... 9-24
Modeling Planar Resistances in Icepak ... 9-25
Pressure Drop Calculations for Grilles ... 9-25
Loss Coefficient ... 9-26
Loss Curve ... 9-26
Assign Grille Properties ... 9-27
Assigning a Grille and Defining Thermal Properties ... 9-28
Assigning Network Properties ... 9-29

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network Nodes and Links 9-31
The Network Thermal Model Dialog 9-31
Network Node and Link Properties 9-33
Face Node Properties 9-33
Boundary Node Properties 9-35
Internal Node Properties 9-36
R Link Properties 9-37
MC Link Properties 9-38
Network Properties Table 9-38
Assign Network Properties 9-39
Assigning Opening Properties 9-40
Assigning Free Opening Properties 9-40
Assign a Free Opening and Define Thermal Properties 9-43
Assigning Recirculating Opening Properties 9-44
Recirculating Opening Project Manager Options 9-46
Flow Direction for Recirculation Openings 9-48
Recirculation Mass Flow Rate 9-49
Recirculating Opening Thermal Specifications 9-49
Assign a Recirculating Opening and Defining Thermal Properties 9-50
Assigning Plate Properties 9-51
Adiabatic Properties 9-52
Assigning an Adiabatic Plate and Defining Thermal Properties 9-54
Conducting Plate Properties 9-54
Assigning a Conducting Plate and Defining Thermal Properties 9-58
Assigning Resistance Properties 9-59
Pressure Drop Calculation for a 3D Resistance 9-59
Device/Approach 9-59
Power Law 9-60

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure Loss Curve 9-60
Assign Resistance Properties 9-61
Assigning a Resistance and Defining Thermal Properties 9-63
Assigning Source Properties 9-64
Assigning a Source and Defining Thermal Properties 9-67
Assigning Wall Properties 9-68
Stationary Wall Properties 9-68
Assigning a Stationary Wall and Defining Thermal Properties 9-74
Wall Heat Transfer Coefficient Correlation Formulas 9-75
Forced Convection 9-75
Laminar Flow 9-75
Turbulent Flow 9-75
Natural Convection 9-76
Vertical Wall (Side Wall) 9-76
Top Wall 9-76
Bottom Wall 9-76
Symmetry Wall Properties 9-77
Assigning EM Loss Properties 9-77
Assign EM Loss Properties 9-78
Preserve source design solution 9-79
Modifying an EM Loss Boundary Condition 9-83
Reassign a Surface or Volume Assignment 9-84
Add to a Surface or Volume Assignment 9-84
Remove from an EM Loss Assignment 9-84
Delete an EM Loss Boundary Condition or Assignment 9-84
Clearing Linked Data 9-84
Assigning Blower Properties 9-85
Impellers - Type 1 Blowers 9-86

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Centrifugal Blowers - Type 2 Blowers 9-86
Specifying Swirl 9-86
Assign Blower Properties 9-88
Assigning a Blower and Defining Thermal Properties 9-89
Assigning Heat Exchanger Properties 9-90
Modeling a Planar Heat Exchanger 9-90
Modeling the Pressure Loss through a Heat Exchanger 9-90
Modeling the Heat Transfer through a Heat Exchanger 9-91
Calculating the Heat Transfer Coefficient 9-91
Assign Heat Exchanger Properties 9-93
Polynomial Pressure Loss and Heat Transfer Coefficient 9-94
Assigning a Heat Exchanger and Defining Thermal Properties 9-95
Identifying Objects 9-96
Identifying Grilles 9-97
Grille Shapes 9-98
Rectangular 9-98
Automatic 9-98
Circular 9-99
Inclined Rectangular 9-100
Polygonal 9-100
Identifying Openings 9-102
Identifying Fans 9-103
Reassigning Thermal Boundary Conditions 9-104
Adding to Thermal Boundary Conditions 9-104
Removing from Thermal Boundary Conditions 9-105
Showing and Hiding Thermal Boundary Conditions 9-105
Reprioritizing Thermal Boundary Conditions 9-106
10 - Native 3D Components in Icepak **10-1**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fans 10-1
Fan Characteristic Curve 10-2
Fans in Series 10-3
Fans in Parallel 10-3
Geometry, Location, and Dimensions 10-4
Simple Fans 10-4
Flow Direction 10-5
Specifying Swirl 10-6
Swirl Magnitude 10-6
Fan RPM 10-7
Fan Efficiency 10-7
Creating Fans 10-7
Creating a Fan 10-15
Heatsinks 10-16
Creating Heatsinks 10-17
Creating a Heatsink 10-25
PCBs 10-26
Coupling HFSS 3D Layout AC and DC Loss 10-27
Coupling SIwave DC IR Loss 10-28
Displaying DC IR Loss 10-29
Convert a Powermap to Sources 10-31
Inclined PCBs 10-33
Create a Relative Coordinate System 10-34
Create a Relative Coordinate System Based on Inclined Geometry 10-34
Creating PCBs 10-37
PCB Grid Resolution 10-55
Creating a PCB 10-57
Bondwire Sheets 10-59

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Surface Materials from SIwave 10-60
Clearing Linked Data 10-60
Chip Thermal Models (CTMs) 10-61
CTMs with Substrate and Routing Layers 10-62
Text File Format 10-64
Creating CTMs 10-68
Creating a CTM 10-70
CTM Visualization 10-71
Display Options 10-72
Export a CTM Field Plot 10-75
3D Integrated Circuits (3D-ICs) 10-77
Creating 3D-ICs 10-79
Creating a 3D-IC 10-81
Packages (Beta) 10-82
Detailed Packages 10-84
Detailed Features 10-84
Approximated Features 10-85
Compact Conduction Model (CCM) Packages 10-86
Lead Frame Packages 10-87
Ball Grid Array (BGA) Packages 10-87
Junction-to-Case Characterization Model 10-87
Junction-to-Board Characterization Model 10-88
Creating a Package 10-89
Defining PGBA Parameters 10-90
Defining Cavity Down BGA Parameters 10-96
Defining FPGBA Parameters 10-103
Defining Flip-chip Parameters 10-110
Defining QFP Parameters 10-117

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining QFN Parameters 10-123
Defining Dual Parameters 10-129
Defining Native Component Instance Parameters 10-135
Define an Instance Parameter 10-136
Map an Instance Parameter to a Project or Design Variable 10-140
Using an Instance Parameter in an Expression 10-144
Defining Variables for Component Parameters 10-145
Create a Heatsink with a Fin Count Variable 10-146
Create a Parametric Sweep 10-148
Viewing Design Variations 10-150
Editing Component Definitions 10-152
Inserting Component Instances 10-154
Making Component Instances Independent 10-155
Project Manager 10-155
History Tree 10-156
11 - Layout Components [Beta] 11-1
Adding a Layout Component 11-3
Two-Way Coupling 11-12
Inserting a Layout Component Instance 11-14
Instance Variation 11-14
Editing a Layout Component Definition 11-16
Replacing or Editing Layout Components 11-22
Edit/Replace Procedure 11-24
Layout Component Properties 11-24
General Tab 11-25
Parameters Tab 11-25
Component Data Tab 11-26
Visualization Tab 11-26

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trace Mapping Using a Layout Component ... 11-27
PCB Grid Resolution ... 11-28
12 - Generating a Mesh ... 12-1
Enabling Concurrent Meshing ... 12-1
Hybrid Mesh ... 12-3
Mesh Quality and Type ... 12-5
Adding a Mesh Linked Solution Setup ... 12-6
Defining Global Mesh Settings ... 12-8
Part By Part Meshing (Beta) ... 12-15
Assigning Mesh Regions ... 12-17
Mesh Parameters for Mesh Regions ... 12-24
Mesh Fusion ... 12-27
Assigning Mesh Operations ... 12-30
Creating a Mesh Size Region ... 12-32
Local Mesh Parameters ... 12-33
Mesh Parameters for Geometry ... 12-34
Box Object - Object Selection Mode ... 12-34
Cylinder Object - Object Selection Mode ... 12-36
Circle Object - Object Selection Mode ... 12-38
2D Rectangular Sheet Object - Object Selection Mode ... 12-40
2D Polygon Object - Object Selection Mode ... 12-41
3D Polygon Object - Object Selection Mode ... 12-42
Object Face - Face Selection Mode ... 12-43
Object Edge - Edge Selection Mode ... 12-44
Mesh Parameters for Heat Sinks ... 12-45
Mesh Reuse ... 12-46
Reassigning a Mesh Region ... 12-47
Enable or Disable a Mesh Region ... 12-48

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing a List of Mesh Regions 12-48
Viewing Mesh Region Properties 12-48
Deleting All Previously Assigned Mesh Regions 12-48
Mesh Region Intersections 12-48
Mesh Region Intersection Limitations 12-49
Mesh Region Intersecting No Objects 12-49
Mesh Region Intersecting Objects 12-50
Mesh Region Intersecting the Outer Region 12-50
Mesh Region Touching Other Mesh Regions 12-51
Mesh Region with No Padding Touching 2D Geometry 12-52
Mesh Region with No Padding Touching 3D Geometry 12-53
Mesh Priority 12-54
Nested Mesh Regions 12-54
Overlapping Mesh 12-54
Generating the Mesh 12-55
Displaying the Mesh 12-57
Reduced Mesh and Full Mesh 12-61
Checking the Mesh 12-63
Checking the Face Alignment 12-63
Checking the Element Volume 12-66
Checking the Skewness 12-68
Equilateral-Volume-Based Skewness 12-69
Skewness-Normalized Equiangular 12-69
Check the Skewness 12-70
13 - Assigning Monitors 13-1
Assign a Face Monitor 13-2
Assign a Point Monitor 13-4
Edit Monitor Definitions 13-5

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reassign a Face Monitor 13-7
Reassign a Point Monitor 13-8
Deleting Monitors 13-8
Monitor Visualization Options 13-8
Setting Monitor Base Names 13-10
14 - Assigning Solar Loading 14-1
Assign Solar Loading 14-1
Solar Loading Context Menus 14-1
15 - Specifying Solution Settings 15-1
The Icepak Solve Setup Dialog 15-1
Icepak Solve Setup Defaults 15-17
Forced Convection 15-18
Natural Convection 15-19
Mixed Convection 15-19
Conduction Only 15-20
Partition Methods 15-20
Adding a Solution Setup to an Icepak Design 15-21
Renaming a Solution Setup 15-23
Copying a Solution Setup 15-24
Deleting a Solution Setup 15-24
Importing Mesh 15-24
Preserve source design solution 15-27
Adding Two-Way Coupling to an Icepak Design 15-28
Disabling and Enabling an Analysis Setup 15-31
Specifying the Number of Processors 15-31
16 - Running Simulations 16-1
Solving a Single Setup 16-2
Running More than One Simulation 16-4

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Submitting a Job ... 16-4
Monitoring Queued Simulations ... 16-9
Monitoring the Solution Process ... 16-10
Solution Overview ... 16-15
Thermal Tab ... 16-16
Flow Tab ... 16-17
Fan Tab ... 16-18
Changing a Solution Priority for System Resources ... 16-19
Reverting an Icepak Simulation ... 16-20
Aborting an Analysis ... 16-22
Re-solving after Modifying a Design ... 16-22
Writing Solver Files ... 16-22
17 - Post Processing and Generating Reports ... 17-1
Viewing Solution Data ... 17-1
Viewing a Solution Profile ... 17-1
Viewing Solution Residuals ... 17-5
Viewing Thermal and Flow Monitors ... 17-5
Deleting Solution Data ... 17-7
Deleting Reports ... 17-8
Creating Animations ... 17-8
Creating Design of Experiments Animations ... 17-10
Creating Geometry Animations ... 17-15
Creating Parametric Animations ... 17-18
Controlling the Animation's Display ... 17-22
Modifying an Animation Setup ... 17-23
Exporting Animations ... 17-26
Exporting Ensight from the Select Animation Dialog Box ... 17-26
Exporting from the Animation Control Dialog Box ... 17-28

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Reports 17-30
Creating a Quick Report 17-31
Creating a New Report 17-32
Context Section for Reports 17-36
Using Families Tab for Reports 17-36
Filtering Quantity Selections for the Reporter 17-39
Modifying Reports 17-40
Modify Report: Selecting Use All Values or Making Selection 17-43
Modify Report: Using the Edit Sweep Dialog Box 17-44
Creating a Report from an Ansoft Report Data File 17-45
Zooming and Fitting Reports 17-45
Modifying the Background Properties of a Report 17-46
Modifying the Legend in a Report 17-48
Creating Custom Report Templates and Defaults 17-51
Exporting Ansoft Report Data Format Files 17-52
Exporting Reports as Graphics 17-54
Report File Formats 17-56
Selecting the Report Type 17-57
Selecting the Display Type 17-59
Creating 2D Rectangular Plots 17-60
Creating 2D Rectangular Stacked Plots 17-62
Creating 3D Rectangular Plots 17-64
Creating Rectangular Contour Plots 17-71
Creating 2D Polar Plots 17-74
Creating Data Tables 17-75
Delta Markers in 2D Reports 17-77
Working with Traces 17-77
Editing Trace Properties 17-79

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing the Display Properties of Traces 17-80
Adding Data Markers to Traces 17-83
Y Markers in Stacked XY Plots 17-85
Discarding Report Values Below a Specified Threshold 17-92
Adding Characteristics to a Trace 17-93
Adding a Recently Used Trace Characteristic 17-93
Adding a Trace Characteristic from Favorites 17-93
Adding Trace Characteristics to your Favorites 17-94
Adding Characteristics Using the Add Trace Characteristics Window 17-96
Removing All Trace Characteristics 17-98
Removing Traces 17-98
Copy and Paste of Report and Trace Definitions 17-98
Copy and Paste of Report and Trace Data 17-99
Limit Lines in Cartesian Plots 17-100
Sweeping a Variable in a Report 17-108
Sweeping Values Across a Distance 17-108
Selecting a Function for a Plot 17-109
Selecting Solution Quantities to Plot 17-115
Selecting a Field Quantity to Plot 17-116
Plotting Imported Solution Data 17-117
Setting a Range Function 17-117
Range Functions 17-118
Eye Measurement Range Function Parameters 17-123
Specifying Output Variables 17-123
Function List for Output Variables 17-126
Plotting Field Overlays 17-126
Plotting Streamlines in Icepak 17-130
Display Streamline Markers 17-133

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Animating the Streamlines Field Plot 17-135
Icepak Post Processing Quantities 17-139
Pressure 17-140
Temperature 17-140
Velocity 17-140
Turbulence 17-140
Thermal Conductivity 17-141
Joule Heating 17-141
EM Mapping 17-141
Others 17-141
Reporting Sides 17-143
Plotting Derived Field Quantities 17-144
Overlaying DC Temperature Fields (General Mode Only) 17-145
Creating 2D Reports from Named Expressions 17-147
Creating Scalar Field Plots 17-148
Working with Scalar Field Plot Markers 17-149
Adding a Field Plot Marker 17-149
Deleting a Field Plot Marker 17-150
Exporting a Field Plot Marker Table 17-151
Clearing All Field Plot Markers 17-151
Editing Field Plot Markers 17-151
Showing Field Plot Markers 17-153
Creating Vector Field Plots 17-153
Modifying Field Plots 17-156
Setting Field Overlay Attributes 17-160
Modifying Field Plot Colors 17-171
Color Key Settings 17-176
Modifying the Field Plot Scale 17-177

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Vector Field Plot Arrows 17-181
Setting the Mesh Visibility on Field Plots 17-185
Modifying Scalar Field Plot Isovalues 17-185
Mapping Scalar Field Plot Transparency to Field Values 17-189
Modifying Markers on Point Plots 17-189
Modifying Line Plots 17-190
Function Command 17-191
Setting a Plot's Visibility 17-192
Saving a Field Overlay Plot 17-194
Opening a Field Overlay Plot 17-195
Deleting a Field Overlay Plot 17-195
Setting Field Overlay Plot Defaults 17-195
Using the Fields Calculator 17-196
Scripting Support 17-198
Opening the Fields Calculator 17-198
Context Area 17-199
The Calculator Stack 17-201
Stack Commands 17-202
Pop 17-203
RlUp 17-203
RlDn 17-203
Exch 17-203
Clear 17-203
Undo 17-203
Registers 17-203
Enlarging the Register Display Area 17-206
Units of Measure 17-206
Input Commands 17-206

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quantity Command 17-208
Geometry Command 17-210
Constant Command 17-214
Number Command 17-215
Geom Settings Command 17-216
Read Command 17-216
General Commands 17-217
Steps for Calculating the Complex Vector Electric Field 17-220
Scalar Commands 17-220
Vec? Command 17-222
1/x (Inverse) Command 17-222
Pow Command 17-222
Square Root Command 17-223
d/d? (Partial Derivative) Command 17-223
Integral Command 17-223
Min Command 17-223
Max Command 17-224
Ln Command 17-225
Log Command 17-225
Std Command 17-225
Vector Commands 17-225
Scal? Command 17-226
Matl Command 17-226
Mag Command 17-227
Dot Command 17-227
Cross Command 17-227
Divg Command 17-227
Curl Command 17-227

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tangent Command 17-227
Normal Command 17-228
Unit Vec Command 17-229
X Form 17-229
Output Commands 17-231
Value Command 17-231
Eval Command 17-233
Write Command 17-234
Export Command 17-236
Calculating Derived Field Quantities Using Named Expressions 17-241
Named Expression Library 17-243
Exiting the Fields Calculator 17-244
Creating a Fields Summary 17-245
Mean and Standard Deviation Calculations 17-250
Mean 17-250
Surfaces 17-250
Volumes 17-250
Standard Deviation 17-250
Surfaces 17-250
Volumes 17-251
Reporting Sides 17-251
Filtering the Entities List 17-252
Type 17-252
Boundary 17-252
Object 17-253
Monitor 17-254
Entity 17-255
Exporting Fan Operating Point Data 17-255

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Outputs (UDOs) 17-257
Named Probes and Properties in User Defined Outputs 17-257
17-262
Computation of Traces Based UDO Calculations 17-262
Dimensions Reduction by UDO Calculations 17-263
Dynamic Probes 17-263
User-Defined Outputs: Python Script API 17-263
UDO Extension Implementation 17-264
Import Statements 17-264
UDOExtension Class 17-264
Validate 17-265
IUDOPluginExtension Abstract Class 17-266
GetUDSName 17-266
GetUDSDescription 17-267
GetUDSSweepNames 17-267
GetCategoryNames 17-268
GetQuantityNames 17-268
GetQuantityInfo 17-269
GetInputUDSParams 17-269
GetDynamicProbes 17-272
Compute 17-273
Data Types Used in UDO Python Scripts 17-275
Constants 17-275
Abstract Classes 17-276
IUDSInputData 17-276
GetDoubleProbeData 17-276
GetSweepsDataForProbe 17-277
GetComplexProbeData 17-277

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetSweepNamesForProbe 17-278
GetRequiredQuantities 17-279
GetVariableValues 17-279
GetInterpolationOrdersData 17-280
IUDSOutputData 17-280
SetSweepsData 17-281
SetDoubleQuantityData 17-281
SetComplexQuantityData 17-282
Working With Properties for UDO 17-282
IPropertyList Abstract Class 17-283
IProperty Abstract Class 17-283
INumberProperty Abstract Class 17-283
ITextProperty Abstract Class 17-284
IMenuProperty Abstract Class 17-284
Other Application-Specific Classes Used in Python Scripts 17-285
Constants Class 17-285
UDSProbeParams Class 17-285
UDSDynamicProbes Class 17-286
QuantityInfo Class 17-286
IProgressMonitor Abstract Class 17-287
SetTaskName 17-288
SetSubTaskName 17-288
BeginTask 17-289
SetTaskProgressPercentage 17-289
CheckForAbort 17-289
EndTask 17-290
Using .NET Collection Classes and Interfaces in Python Scripts 17-290
User-Defined Outputs: Messaging Methods 17-292

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AddErrorMessage 17-297
AddInfoMessage 17-298
AddWarningMessage 17-298
User-Defined Outputs: Script Organization 17-298
Using Script Libraries 17-299
Using Additional .NET Assemblies 17-299
User-Defined Documents (UDDs) 17-300
Create User-Defined Document Inputs 17-301
UDD Document Creation and Display 17-303
Managing Documents Listed in the Project Window Under Results 17-304
Documents Folder's Context Menu 17-305
Document Folder's Property Window 17-305
Viewing UDDs with an HTML Web Browser 17-306
UDD Script Libraries 17-307
User-Defined Documents: Python Script API 17-308
Import Statements 17-308
Data Types Used in UDD Python Scripts 17-309
Constants 17-309
Abstract Classes 17-309
IProgressMonitor Abstract Class 17-309
SetTaskName 17-310
SetSubTaskName 17-310
BeginTask 17-311
SetTaskProgressPercentage 17-311
CheckForAbort 17-312
EndTask 17-312
IUDDPluginExtension Abstract Class 17-313
GetUDDName 17-313

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetUDDDescription 17-314
ShowDefaultSetupDialog 17-314
GetUDDInputParams 17-314
Generate 17-316
SetupUDDInputParams 17-318
HandleUDDEvents 17-319
GetUDDSchema 17-320
GetUDDStyleSheetForHtml 17-320
GetUDDStyleSheetForPdf 17-321
GetFopExecutable 17-321
GetUDDAppContext 17-322
GetUDDDesignContext 17-322
UDDExtension Class 17-323
UDDInputData Class 17-323
UDDInputBool 17-324
UDDInputDouble 17-324
UDDInputSolution 17-324
UDDInputText 17-325
UDDInputTrace 17-325
UDDInputParams Class 17-326
UDD Input Interfaces 17-326
Document Generator Interfaces 17-328
Chip Model Analyzer (CMA) 17-333
PinToPinUtility 17-333
18 - Transient Simulations 18-1
Selecting the Transient Solution Type for Icepak 18-1
Specifying Parameters as a Function of Time 18-2
Linear 18-3

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Power Law 18-4
Exponential 18-4
Sinusoidal 18-5
Piecewise Linear 18-5
Square Wave 18-5
Using Datasets to Define Transient Parameters 18-6
Defining a Dataset for Transient Parameters 18-8
Transient Thermal Properties Dependency 18-9
Icepak Setting 18-10
Icepak 3D Native Component 18-10
Boundary Conditions 18-10
Assigning Initial Temperatures 18-11
Assign an Initial Temperature 18-12
Initial Temperature Context Menus 18-13
Initial Temperature Visualization Options 18-14
Setting Initial Temperature Base Names 18-15
The Icepak Solve Setup Dialog (Transient) 18-17
Specifying Varying Time Steps for Transient Solution Setups 18-31
Linear 18-33
Square Wave 18-34
Piecewise Constant 18-35
Piecewise Linear 18-35
Automatic 18-36
Specifying Parameters for Automatic Time Stepping 18-37
Transient Import Options Requirements 18-38
Icepak Solve Setup Defaults 18-40
Forced Convection 18-41
Natural Convection 18-42

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixed Convection 18-42
Conduction Only 18-43
LTI ROM 18-43
Using Solved Simulation Data 18-44
Preserve source design solution 18-46
Monitoring a Transient Solution 18-47
Solution Overview 18-52
Thermal Tab 18-53
Flow Tab 18-54
Fan Tab 18-55
Postprocessing Transient Solutions 18-56
Create a Fields Summary 18-56
Create a Field Plot 18-57
Animate a Field Plot 18-59
Create a Report 18-61
Use the Fields Calculator 18-63
19 - Ansys Workbench Integration Overview 19-1
Integrating Ansys EM Suite with Ansys Workbench 19-1
Workbench Data Integration Overview 19-3
Electronics Desktop Icepak Systems in Workbench 19-5
Adding New Analysis Systems 19-6
Importing Existing Ansys EM Projects into Workbench 19-8
Editing Ansys EM Models in Workbench 19-9
Analyzing Ansys EM Models in Workbench 19-10
Performing Parameter Studies in Workbench 19-12
Scripting in Workbench 19-16
Ansys EM CAD Integration through Workbench 19-17
CAD Integration Functionality 19-19

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CAD Integration and Geometry Sharing 19-21
Bi-Directional CAD Integration 19-24
CAD Integration Model Edits 19-25
Multiple Geometry Links for CAD Integration 19-27
Healing with CAD Integration 19-27
Important Geometry Options for CAD Integration 19-28
Ansys EM to Ansys Geometry Transfer 19-29
Material Assignment Transfer 19-31
Geometry Transfer through Ansys DesignModeler (DM) 19-32
Workbench Material Data Transfer 19-32
Detailed Behavior 19-32
20 - Icepak Technical Notes 20-1
Governing Equations 20-1
The Mass Conservation Equation 20-1
Equation 1 20-1
Equation 2 20-1
Momentum Equations 20-1
Equation 3 20-1
Equation 4 20-2
Energy Conservation Equation 20-2
Equation 5 20-2
Equation 6 20-2
Turbulence 20-3
Zero-Equation Turbulence Model 20-3
Equation 7 20-3
Equation 8 20-3
Equation 9 20-3
Equation 10 20-3

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Turbulence Models 20-3
Reynolds (Ensemble) Averaging 20-4
Equation 11 20-4
Equation 12 20-4
Equation 13 20-4
Equation 14 20-5
Equation 15 20-5
Boussinesq Approach 20-5
Equation 16 20-5
Choosing an Advanced Turbulence Model 20-6
The Spalart-Allmaras Model 20-6
The Standard Model 20-6
The RNG Model 20-6
The Realizable Model 20-7
The Enhanced Two-Equation Models 20-7
Enhanced Wall Treatment 20-8
Computational Effort: CPU Time and Solution Behavior 20-8
The Spalart-Allmaras Model 20-9
Transport Equation for the Spalart-Allmaras Model 20-9
Equation 17 20-9
Modeling the Turbulent Viscosity 20-9
Equation 18 20-10
Equation 19 20-10
Equation 20 20-10
Modeling the Turbulent Production 20-10
Equation 21 20-10
Equation 22 20-10
Equation 23 20-10

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 24 20-11
Equation 25 20-11
Equation 26 20-11
Equation 27 20-11
Equation 28 20-11
Modeling the Turbulent Destruction 20-12
Equation 29 20-12
Equation 30 20-12
Equation 31 20-12
Equation 32 20-12
Model Constants 20-12
Equation 33 20-12
Equation 34 20-13
Wall Boundary Conditions 20-13
Equation 35 20-13
Equation 36 20-13
Convective Heat and Mass Transfer Modeling 20-13
Equation 37 20-13
Equation 38 20-14
Two-Equation (Standard) Turbulence Model 20-14
Transport Equations for the Standard Model 20-14
Equation 39 20-14
Equation 40 20-14
Modeling the Turbulent Viscosity 20-15
Equation 41 20-15
Model Constants 20-15
Equation 42 20-15
The RNG Model 20-15

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transport Equations for the RNG ε Model 20-15
Equation 43 20-16
Equation 44 20-16
Modeling the Effective Viscosity 20-16
Equation 45 20-16
Equation 46 20-16
Equation 47 20-17
Calculating the Inverse Effective Prandtl Numbers 20-17
Equation 48 20-17
The Term in the Equation 20-17
Equation 49 20-17
Equation 50 20-17
Equation 51 20-18
Model Constants 20-18
Modeling Turbulent Production in the Models 20-18
Equation 52 20-18
Equation 53 20-18
Equation 54 20-19
Equation 55 20-19
Effects of Buoyancy on Turbulence in the Models 20-19
Equation 56 20-19
Equation 57 20-19
Equation 58 20-20
Convective Heat Transfer Modeling in the Models 20-20
Equation 59 20-20
Equation 60 20-20
Equation 61 20-20
SST Model 20-21

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transport Equations for the Transition SST Model 20-21
Equation 62 20-21
Equation 63 20-21
Equation 64 20-22
Equation 65 20-22
Equation 66 20-22
Equation 67 20-22
Equation 68 20-22
Separation-Induced Transition Correction 20-22
Equation 69 20-23
Equation 70 20-23
Equation 71 20-23
Equation 72 20-23
Equation 73 20-24
Equation 74 20-24
Equation 75 20-24
Equation 76 20-24
Equation 77 20-25
Coupling the Transition Model and SST Transport Equations 20-25
Equation 78 20-25
Equation 79 20-25
Equation 80 20-25
Specifying Inlet Turbulence Levels 20-25
Equation 81 20-26
Equation 82 20-26
Equation 83 20-26
Equation 84 20-26
Equation 85 20-26

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: Exemplary Decay of Turbulence Intensity (Tu) as a Function of Streamwise Distance (x) 20-27
Buoyancy-Driven Flows and Natural Convection 20-27
Equation 86 20-27
Equation 87 20-27
Equation 88 20-28
The Boussinesq Model 20-28
Equation 89 20-28
Incompressible Ideal Gas Law 20-29
Equation 90 20-29
Definition of the Operating Density 20-29
Equation 91 20-29
Equation 92 20-29
Radiation 20-29
Overview 20-29
Gray-Diffuse Radiation 20-30
The Discrete Ordinates (DO) Radiation Model 20-30
The DO Model Equations 20-30
Equation 93 20-30
Angular Discretization and Pixelation 20-31
The Ray Tracing Radiation Model 20-35
Solution Procedures 20-35
Overview of Numerical Scheme 20-35
Overview of Solution Method 20-37
Linearization 20-37
Spatial Discretization 20-38
Equation 106 20-38
Equation 107 20-38

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Control Volume Used to Illustrate Discretization of a Scalar Transport Equation 20-39
First-Order Upwind Scheme 20-40
Second-Order Upwind Scheme 20-40
Equation 108 20-40
Equation 109 20-40
Linearized Form of the Discrete Equation 20-40
Equation 110 20-41
Under-Relaxation 20-41
Equation 111 20-41
Discretization of the Momentum and Continuity Equations 20-41
Equation 112 20-41
Equation 113 20-42
Discretization of the Momentum Equation 20-42
Equation 114 20-42
Pressure Interpolation Schemes 20-42
Discretization of the Continuity Equation 20-43
Equation 115 20-43
Equation 116 20-43
Pressure-Velocity Coupling with SIMPLE 20-44
Equation 117 20-44
Equation 118 20-44
Equation 119 20-44
Equation 120 20-44
Equation 121 20-45
Equation 122 20-45
Equation 123 20-45
Coupled Pressure-Velocity Formulation 20-45
Equation 124 20-46

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 125 20-46
Equation 126 20-46
Equation 127 20-46
Equation 128 20-46
Equation 129 20-47
Equation 130 20-47
Pseudo Transient Under-Relaxation 20-47
Equation 131 20-47
Time Discretization 20-47
Equation 132 20-48
Equation 133 20-48
Equation 134 20-48
Equation 135 20-48
Multigrid Method 20-49
Approach 20-49
The Need for Multigrid 20-49
The Basic Concept in Multigrid 20-49
Equation 136 20-49
Equation 137 20-50
Equation 138 20-50
Equation 139 20-50
Equation 140 20-50
Restriction and Prolongation 20-50
Equation 141 20-50
Equation 142 20-51
Unstructured Multigrid 20-51
Multigrid Cycles 20-51
The V and W Cycles 20-51

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V-Cycle Multigrid 20-52
W-Cycle Multigrid 20-53
The Flexible Cycle 20-54
Logic Controlling the Flex Multigrid Cycle 20-55
The F Cycles 20-55
The Residual Reduction Rate Criteria 20-56
Equation 143 20-56
The Termination Criteria 20-56
Equation 144 20-57
Restriction, Prolongation, and Coarse-Level Operators 20-57
AMG Restriction and Prolongation Operators 20-57
Equation 145 20-57
AMG Coarse Level Operator 20-58
Equation 146 20-58
Equation 147 20-58
Equation 148 20-58
Equation 149 20-58
Solution Residuals 20-59
Equation 150 20-59
Equation 151 20-59
Equation 152 20-59
Equation 153 20-60
Equation 154 20-60
Equation 155 20-60
21 - Icepak Troubleshooting 21-1
Icepak Troubleshooting - Meshing 21-1
Icepak Troubleshooting - Boundary Conditions 21-2
22 - Scripting 22-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

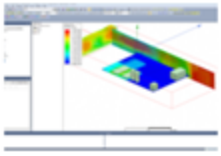
23 - Glossary **23-1**
Glossary: A 23-1
Glossary: B 23-4
Glossary: C 23-6
Glossary: D 23-10
Glossary: E 23-14
Glossary: F 23-17
Glossary: G 23-20
Glossary: H 23-22
Glossary: I 23-24
Glossary: J 23-25
Glossary: K 23-26
Glossary: L 23-27
Glossary: M 23-29
Glossary: N 23-32
Glossary: O 23-35
Glossary: P 23-37
Glossary: Q 23-41
Glossary: R 23-41
Glossary: S 23-43
Glossary: T 23-49
Glossary: U 23-52
Glossary: V 23-53
Glossary: W 23-54
Glossary: X 23-55
Glossary: Y 23-56
Glossary: Z 23-56
Index **Index-1**

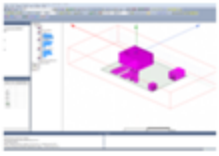
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

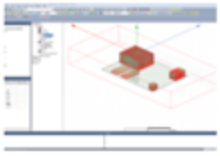
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Welcome to Icepak Help

Click for help on these topics.

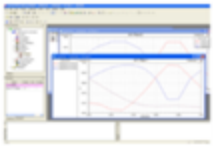

Getting Started Icepak


Modeling


Meshing


Icepak Boundary Conditions


Icepak Native Components


Optimetrics

Click for help on Icepak **Process Flow** topics:



Click for Icepak General Quick Links including The Ansys Electronics Desktop and Working with Projects.

Icepak General Quick Links

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

- Icepak User Interface
- Setting Up an Icepak Design
- Modeling
- Analysis
- Results
- Solution Types
- Materials
- Meshing

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimetrics
Example Projects
Ansys Web site

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

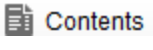
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.
- **Ansys Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Icepak User Interface Quick Links

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

The Ansys Electronics Desktop
Setting Options in Icepak
Keyboard Shortcuts for General Purposes
Moving, Showing, and Hiding Windows
Working with Ribbons
Working with Short Cut Menus
Running Icepak From a Command Line
Working with Variables

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.
- **Ansys Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

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
Creating 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* – 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

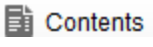
- **Ansys Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Materials Quick Links

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

- Assigning Materials
- Searching for Materials
- Assigning Material Property Types
- Defining Functional Material Properties
- Adding New Materials
- Defining Variable Material Properties
- Defining Material Properties as Expressions
- Viewing and Modifying Material Attributes
- Specifying Thermal Modifiers

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 Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Meshing Quick Links

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

- Defining Global Mesh

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings
Mesh Quality and Type
Assigning Mesh Regions
Mesh Priority
Mesh Region Intersections
Generating the Mesh
Displaying the Mesh
Checking the Mesh

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* – 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 Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Analysis Quick Links

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

Specifying the HPC and Analysis Options
Configuring Distributed Analysis
Selecting Optimal Configurations for Distributed Analysis
Running Simulations
Remote Analysis
Specifying Solution Settings
High Performance Computing Integration

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

Ansys Electromagnetics Suite 2026 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.
- **Ansys Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Optimetrics Quick Links

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

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

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 Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Results Quick Links

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

View solution data
Plot field overlay
Create 2D or 3D reports of S-parameters
Create animations
Creating Reports
Working with Traces
Using the Fields Calculator

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 Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Icepak Getting Started Guides

Icepak documentation includes the following Getting Started Guide:

- *Getting Started with Icepak* - This guides you through the following tasks involved with running an Icepak simulation:
  - Setting up project options and settings
  - Creating and assigning materials

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Assigning boundary conditions
    - Creating mesh regions and setting global mesh settings
    - Creating solution monitors
    - Adding a solution setup and running the simulation
    - Creating post-processing objects
    - Creating a fields summary report
    - Creating variables and running a design of experiments
- *Classic Icepak File Conversion* - This guides you through opening a Classic Icepak project and reviews how the following project elements are represented in the Electronics Desktop:
    - Geometry and Boundary Conditions
    - Global Mesh Settings
    - Mesh Regions
    - Monitors
    - Design Settings and Solution Setup
    - Design Variables and Parametric Setup
    - Postprocessing
    - Fields Summary Report
- *Getting Started with Icepak - Coil and Plate* - This guides you through the following tasks involved with running an Icepak simulation:
    - Setting up a project for Maxwell-Icepak coupling
    - Assigning boundary conditions
    - Adding a solution setup and running the simulation
    - Creating post-processing objects
    - Creating a fields summary report
- *Getting Started with Icepak - Waveguide Filter* - This guides you through the following tasks involved with running an Icepak simulation:
    - Setting up a project for HFSS-Icepak coupling
    - Assigning boundary conditions
    - Adding a solution setup and running the simulation
    - Creating post-processing objects
    - Creating a fields summary report
- *Getting Started Guide - Transient Simulation* - This guides you through the following tasks involved with running a transient Icepak simulation:
    - Selecting the Transient solution type
    - Reviewing a dataset
    - Assigning a piecewise linear transient power to a block boundary condition

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Reviewing transient solve setup options
    - Monitoring a transient simulation
    - Post-processing transient simulation results
- *Getting Started Guide - Toolkits* - This guides you through the following tasks involved with using Icepak toolkits:
    - Accessing Icepak toolkits
    - Creating a package using a toolkit
    - Solving the model and extracting the JB theta value using a toolkit
- *Getting Started Guide - LTI ROM* - This guides you through the following tasks involved with using the LTI ROM toolkit and couple with Twin Builder:
    - Assign boundary conditions.
    - Create an LTI ROM and run a parametric setup.
    - Export the LTI ROM .
    - Use the LTI ROM to prepare and run a Twin Builder analysis.
- *Getting Started Guide - LPV ROM* - This guides you through the following tasks involved with using the LPV ROM toolkit and couple with Twin Builder:
    - Assign boundary conditions.
    - Create an LPV ROM and run a parametric setup.
    - Export the LPV ROM .
    - Use the LPV ROM to prepare and run a Twin Builder analysis.
- *Getting Started Guide - Scripting Example* - This guides you through the following tasks involved with using Icepak scripting commands:
    - Solving the Graphics Card model.
    - Recording a script that includes creating a plane, plotting temperature data, and exporting an image.
    - Editing the script to produce additional planes, field plots, and images.
- *Getting Started Guide - Finned Heat Sink* - This guides you through the following tasks involved with modeling a finned heat sink:
    - Create a new project.
    - Create a model using geometry and thermal boundary conditions.
    - Generate a mesh
    - Set up a simulation with various physical conditions and parameters, including turbulence.
    - Calculate and monitor a solution.
    - Post-process the results by plotting field overlays
- *Getting Started Guide - RF Amplifier* - This guides you through the following tasks involved with running an Icepak simulation:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Create a new project.
    - Create a model using openings, fans, sources, blocks, heat sinks, and walls.
    - Set up a simulation with various physical conditions and parameters, including turbulence and natural convection.
    - Calculate and monitor a solution.
    - Post-process the results by plotting field overlays.
- *Getting Started with Icepak - Optimization of Fan Location* - This guides you through the following tasks involved with running an Icepak simulation:
    - Create a new project.
    - Create a model using grilles, fans, sources, blocks, networks, heat sinks, and walls.
    - Assign mesh regions and generate a mesh.
    - Set up and run parametric files using a variable for fan location.
    - Post-process the results by plotting field overlays.
- *Getting Started with Icepak - Cold Plate Model* - This guides you through the following tasks involved with running an Icepak simulation:
    - Use a mesh region and mesh operation to mesh complicated model setups
    - Use multiple fluids in a single model.
    - Account for external natural convection and internal forced convection.
    - Create mesh regions to reduce the overall mesh count.
    - Specify per-object meshing parameters.
- *Getting Started with Icepak - Heat Pipe Model* - This guides you through the following tasks involved with running an Icepak simulation:
    - Create anisotropic solid materials.
    - Assign materials to simulate a simplified heat-pipe in a system.
    - Assign nested non-conformal mesh regions.
    - Add a solution setup and run the simulation.
    - Create temperature and velocity vector field plots.
- *Getting Started with Icepak - Non-Conformal Mesh* - This guides you through the following tasks involved with running an Icepak simulation:
    - Generate a non-conformal mesh and related parameters such as slack values and maximum element sizes.
    - Understand the effects of a non-conformal mesh on total mesh count and results.
    - Generate and compare summary reports.
    - Apply non-conformal rules and restrictions.
- *Getting Started with Icepak - Extract Delphi Network* - This guides you through the following tasks involved with running an Icepak simulation:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Use Icepak's QFN toolkit to create a QFN package.
    - Use the Extract Delphi Network to generate an optimized compact thermal model (CTM) 3D component, which can be used to replace a detailed thermal model (DTM) in a system-level design to reduce mesh size and simulation duration, while yielding accurate thermal results.
- *Icepak - IcepakFEA Coupling: Structural Solution - Power Resistor* - This guide describes how to construct, set up, and solve two designs with an identical power resistor:
    - An Icepak design (specifically, a steady-state thermal analysis) in which the resistor body is a heat source, air movement removes heat, and a fixed temperature is assigned to the face of the solder joints (where they would contact a circuit board trace).
    - An IcepakFEA – Structural design which imports the temperature results from the Icepak design and determines the resultant thermal stresses and displacements.

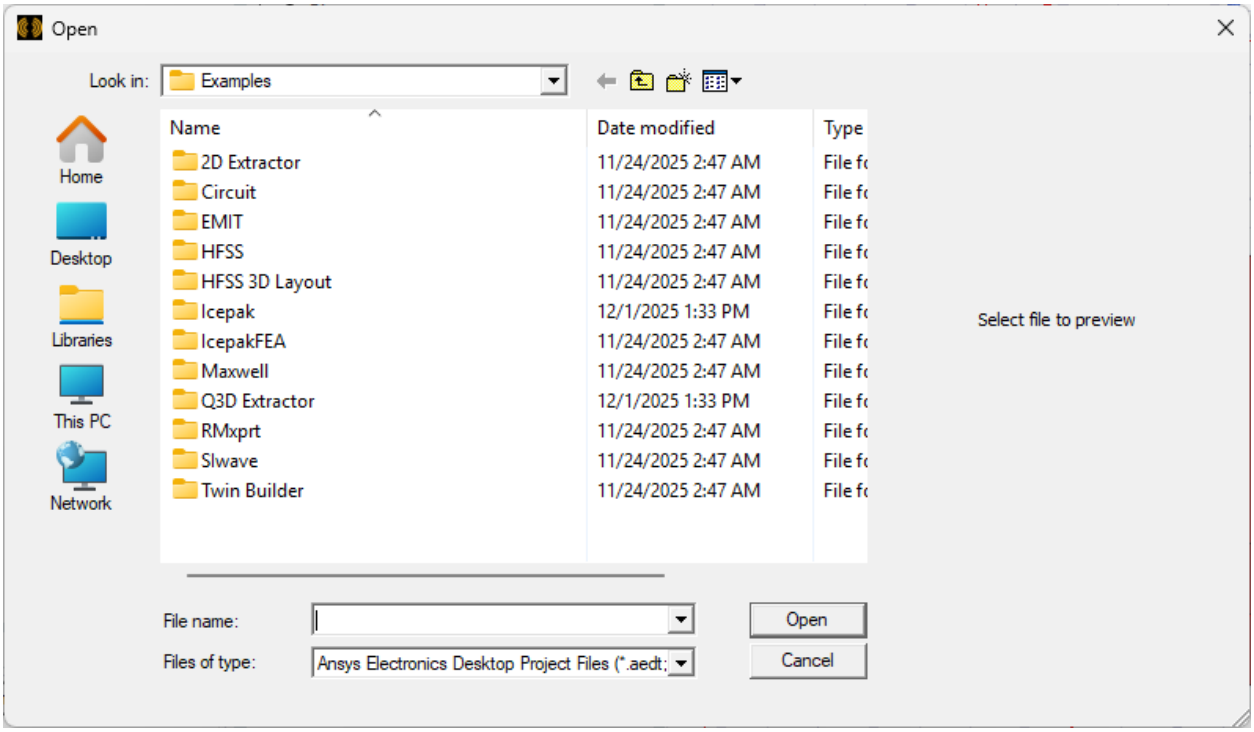
  By following the steps in this guide, you will learn how to perform the following thermal analysis tasks in Ansys Electronics Desktop:

    - Insert an Icepak and an IcepakFEA design
    - Choose the Structural solution type for the IcepakFEA design
    - Set the model units
    - Create the power resistor geometry in IcepakFEA
    - Specify component names, materials, and colors
    - Copy geometry to Icepak
    - Assign thermal sources in Icepak
    - Assign thermal openings in Icepak
    - Adjust the air region size in Icepak
    - Solve the Icepak analysis and review the temperature results
    - Assign fixed supports in IcepakFEA
    - Assign a non-uniform thermal condition in IcepakFEA, linking to the Icepak results
    - Solve the IcepakFEA – Structural analysis
    - Compare imported temperatures in IcepakFEA to Icepak temperature results
    - Create and animate a displacement magnitude overlay
    - Create an equivalent stress overlay in IcepakFEA
    - Define a clipping plane to see the internal stresses

Example Projects

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example projects are organized by the design type.

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

Finding Information in the Online Help

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

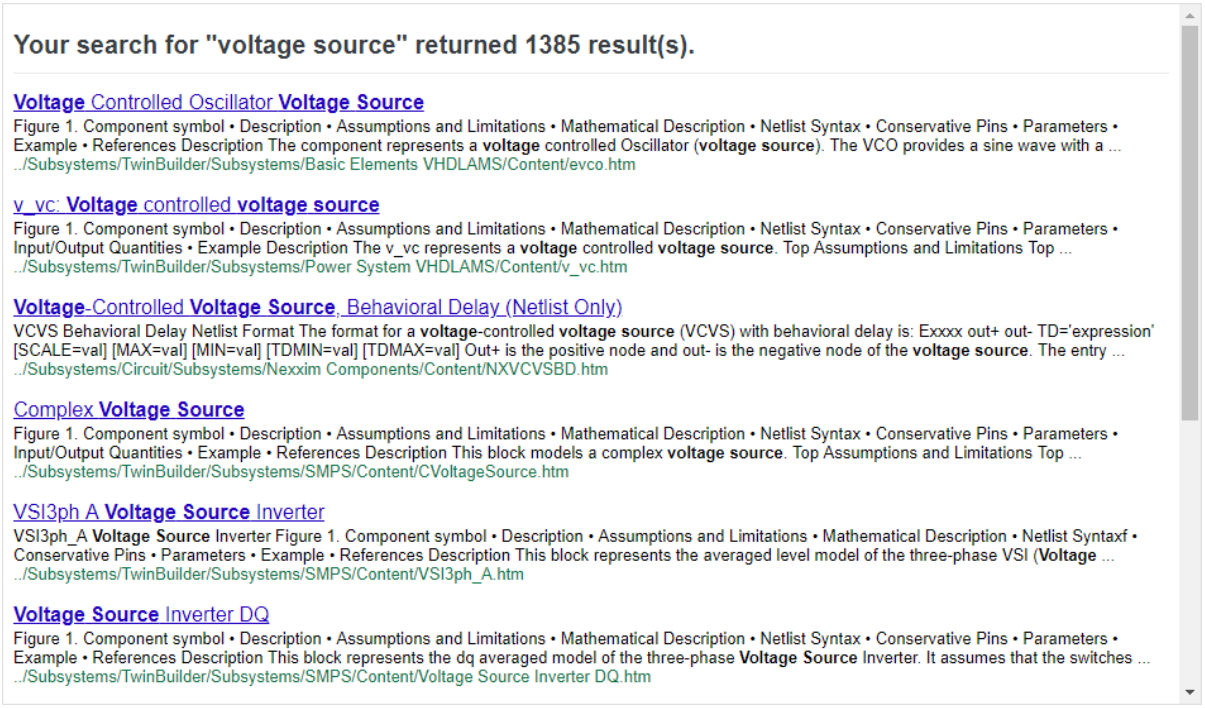
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.
- **Ansys Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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.”

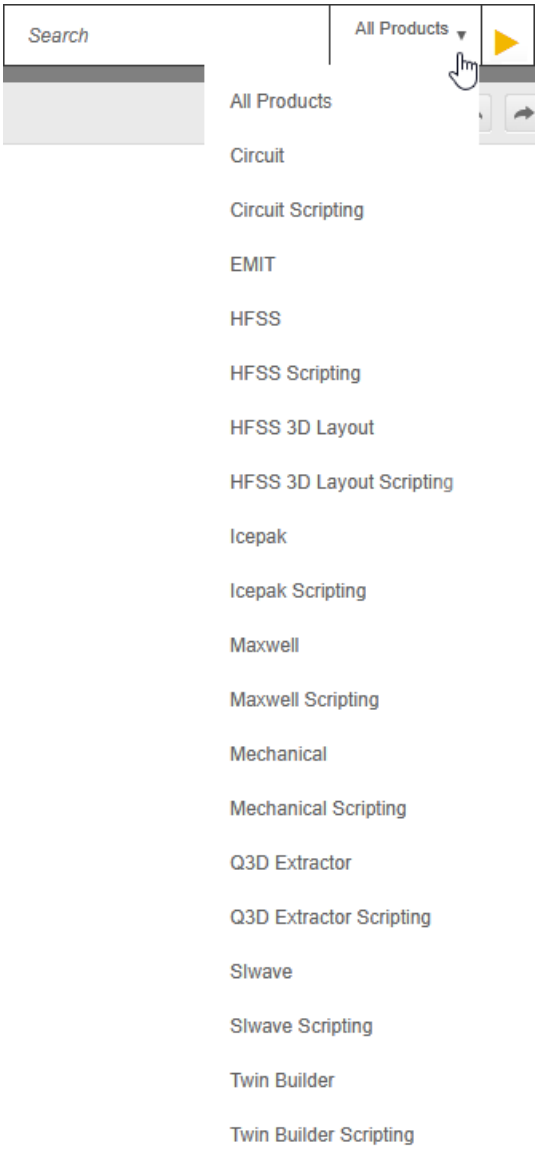


This method probably provides more hits than you want. The search function in the help provides several methods for making searches more specific.

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 2026 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.

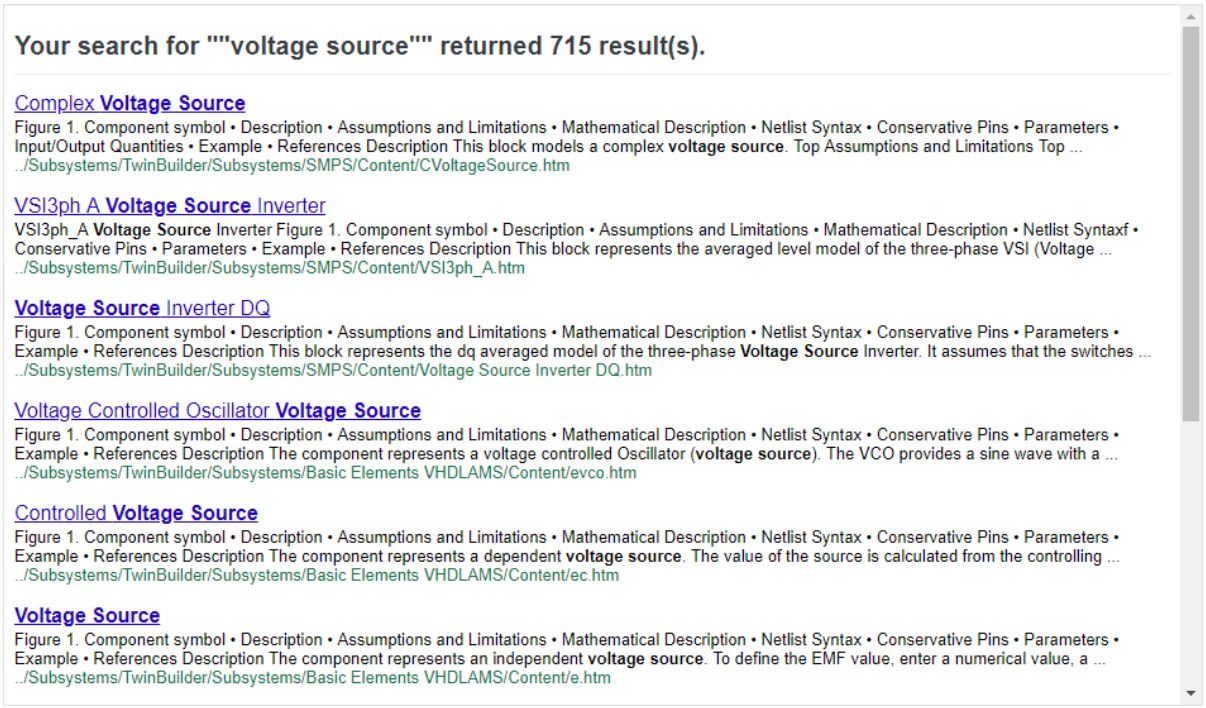
2. Click on the topic you want in the results list.
   - Some topics in different products share the same title. When searching "All Products," check the URL below each link. The URL indicates the product under which the topic falls.
   - After clicking a topic, if you want to view a different topic,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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To turn off highlighting on the page you are viewing, click the Remove Highlights icon ( ).

Searching with Quotation Marks

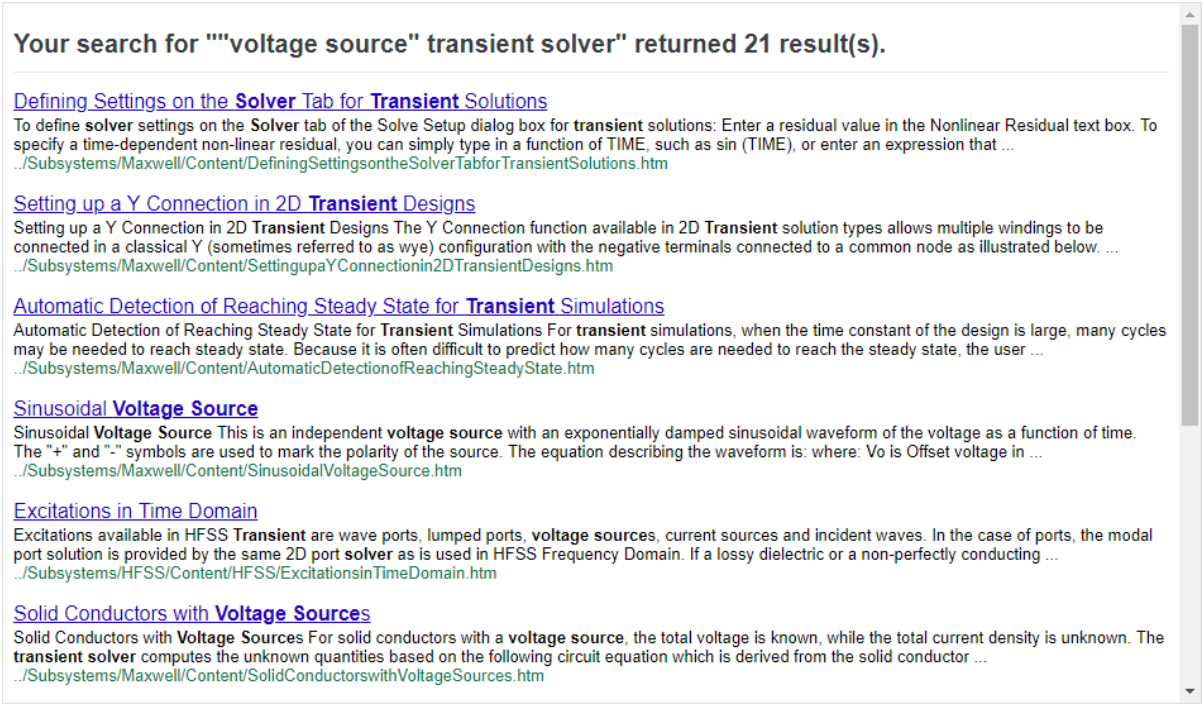
If you enter “voltage source” with quotation marks, the results show all topics that include the phrase.



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 2026 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 2026 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**.

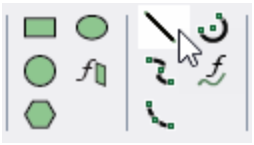
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 2026 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.

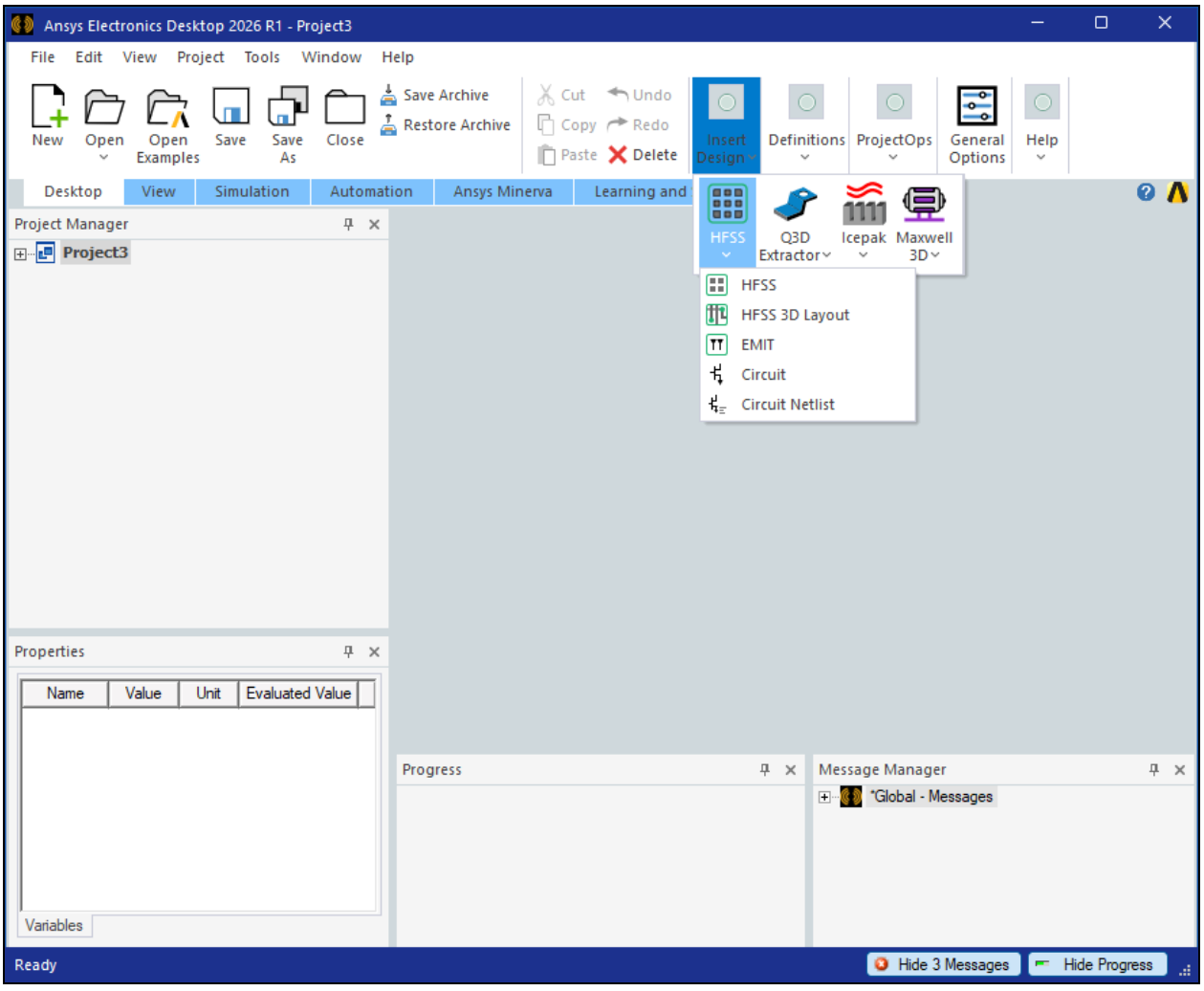
You can also contact Technical Support through Ansys Engineering Copilot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **HFSS 3D Layout** – a full-wave, layout-based electromagnetic simulator with a specialized interface for geometries created in layout.
- **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, AC magnetic (eddy current), and transient problems.
- – uses finite element analysis (FEA) to solve two-dimensional (2D) electrostatic, magnetostatic, AC magnetic (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 AC Magnetic 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.
- **IcepakFEA** – perform modal analyses to determine natural vibration frequencies and thermal analyses to determine temperatures and heat flux.

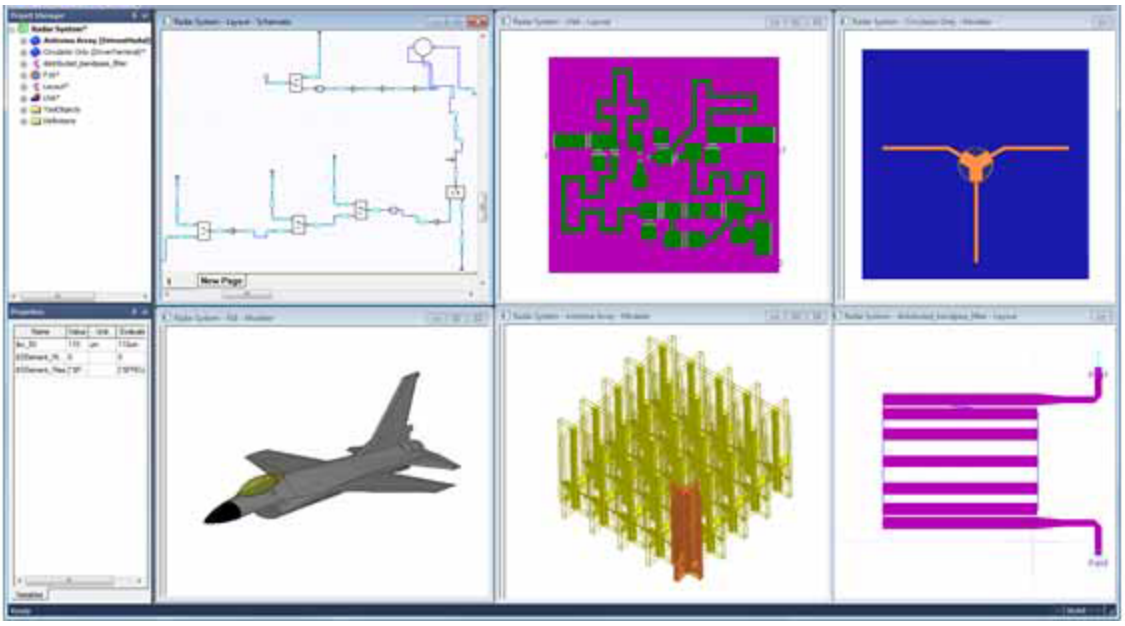
You can access all of these design types and features from the **Project** menu, and any combination of design types can be inserted into a single project file. The schematics can be used to wire up the different field solver models and create a model of a high-level system. Ansys Electronics Desktop 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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

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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Unsupported design types (e.g., IcepakFEA) do not open in 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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

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 2026 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. For up-to-date platform support by product:

1. Go to the Ansys Platform Support website.
2. From the list of documents, click the link, Ansys 2026 R1 - Platform Support by Application / Product (PDF).
3. This document includes platform support information for all Ansys products. Refer to the section labeled Electronics Applications.

Launching Ansys Electronics Desktop

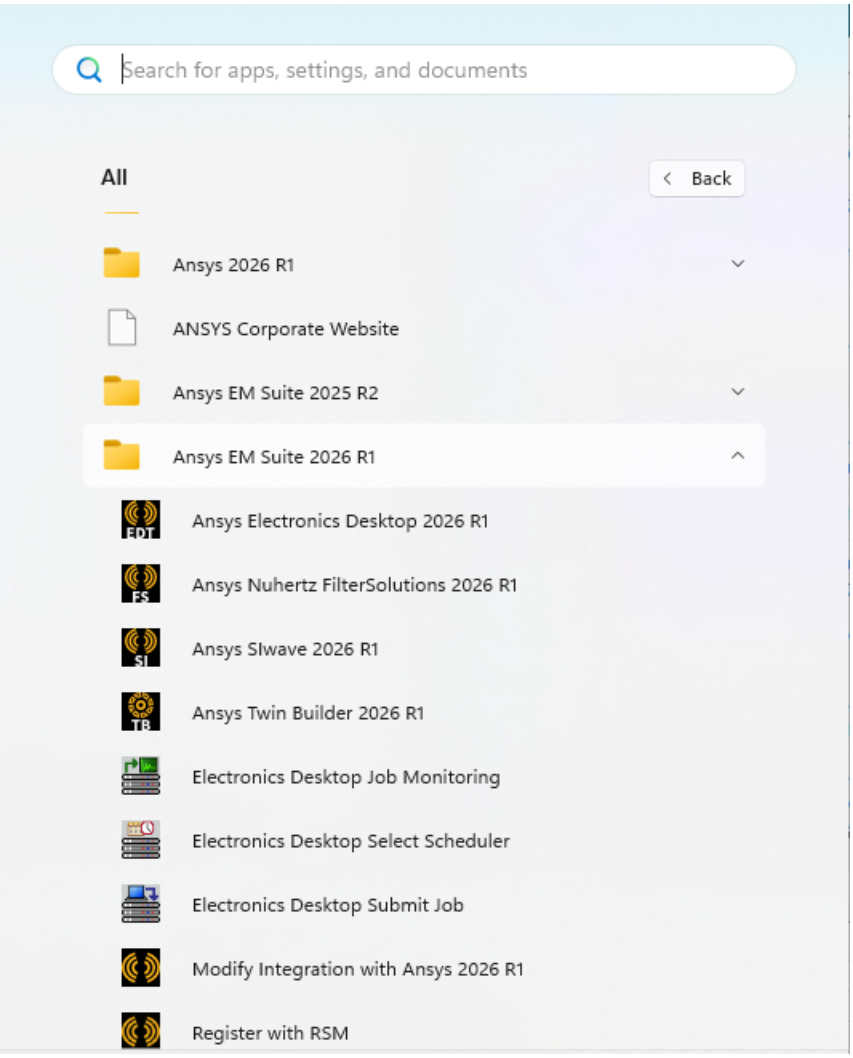
Follow the instructions below to launch Ansys Electronics Desktop in either Windows or Linux.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Windows:

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

- From the Start icon, select **All**, and then select **Ansys EM Suite 2026 R1**> **Ansys Electronics Desktop 2026 R1**.



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 Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

High DPI Systems

If the program starts but graphics seem very small, Windows may be running on a high DPI display. In this case, it may be necessary to change high DPI settings and enable DPI Scaling in the program's properties. Consult Microsoft Windows documentation for the latest instructions for doing so.

Linux:

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

Launching AEDT from the command line:

When AEDT is installed, a sub-directory is created under the installation directory: /v261/AnsysEM/. This directory contains the software executable files and wrapper scripts.

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

- /opt/ansys_inc/v261/AnsysEM/ansysedt

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

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

Starting from GNOME desktop:

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



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the icon for Ansys EM:



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:

- Application
    - Build information
    - System information
        - Country
        - Country code
        - CPU architecture
        - CPU brand
        - CPU identifier
        - Graphics card

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

        - 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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 ElectronicsDesktop2026.1 -
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.

   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?*

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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?*

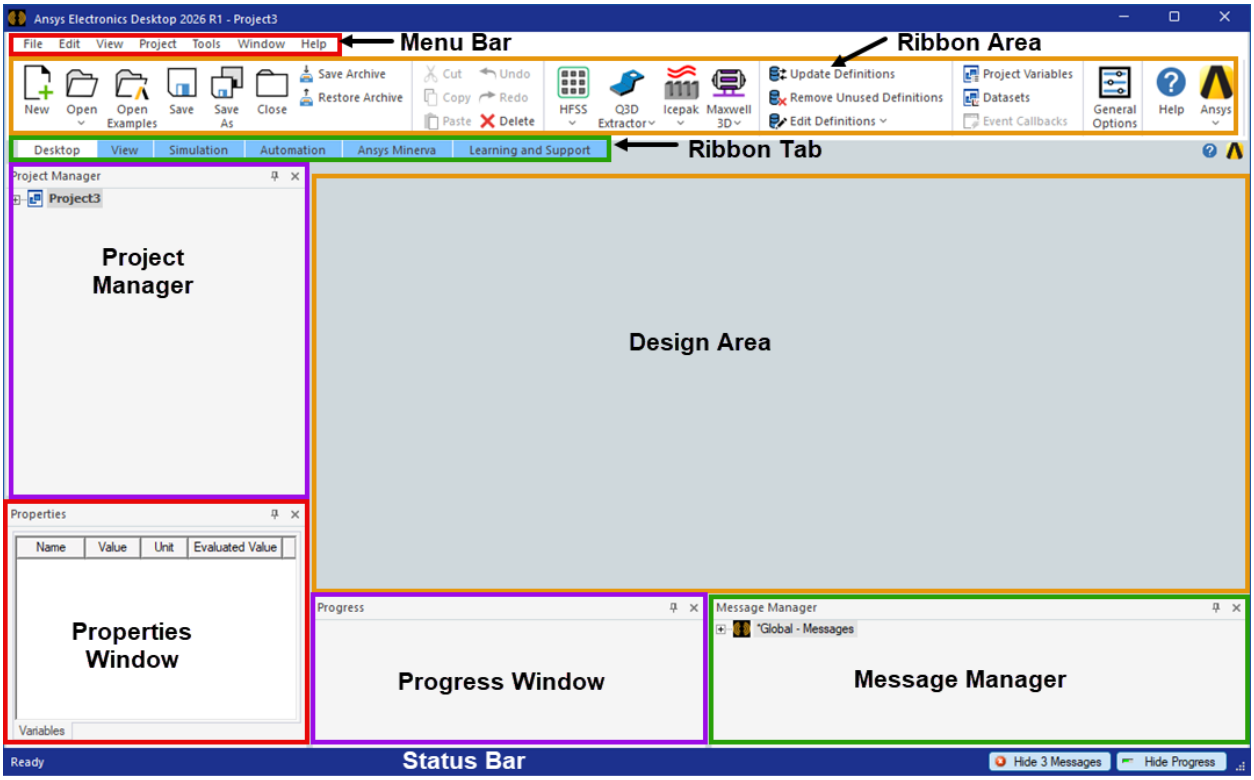
   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

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.

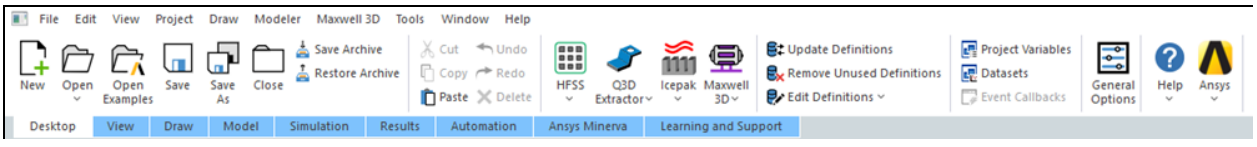
> **Note:**
>
> To ensure that all Electronics Desktop dialog boxes are visible on your monitor, it is recommended to not scale the display resolution above 100%.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



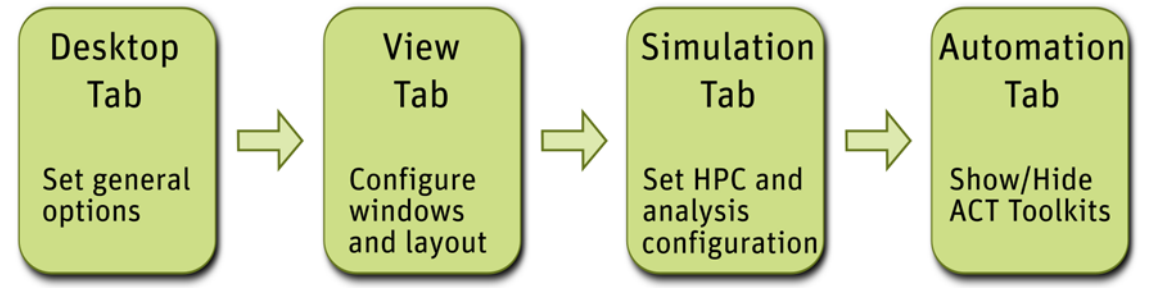
Working with Ribbons

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:

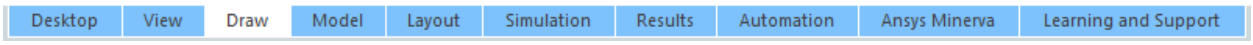
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

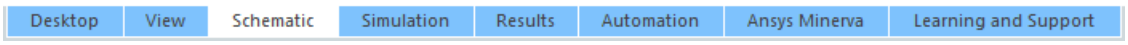
- For HFSS, Icepak, Maxwell, IcepakFEA, and Q3D designs, the **Draw**, **Model**, and **Results** tabs also appear:



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



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

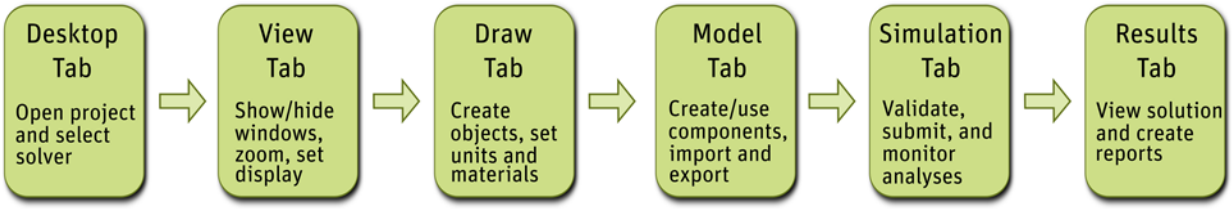


In addition to **Desktop**, **View**, **Minerva** and **Learning and Support** the ribbon tabs that are applicable to Icepak designs are described as follows:

- **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), install PyAEDT (Beta) and customize buttons to access Icepak toolkits. See Customizing the Automation Tab for more information on customizing toolkit buttons.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

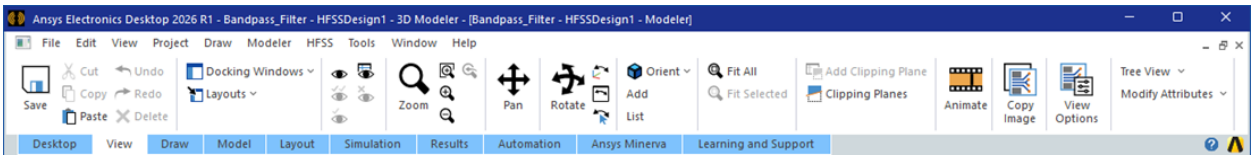
Each tab contains features specific to the design type, and the general workflow is from left to right.



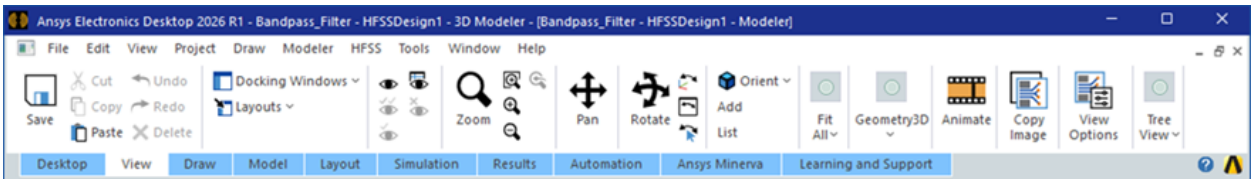
Note that workflow can vary depending on tasks at hand. For example, if you open a completed model in HFSS or use 3D components or the ACT Antenna Design toolkit (Windows only), you may not need the features of the **Draw** tab. If Validation on the **Simulation** tab identifies problems in a model, you may need to use the **Modeler** menu commands for Analysis and Heal, or use the CAD 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:



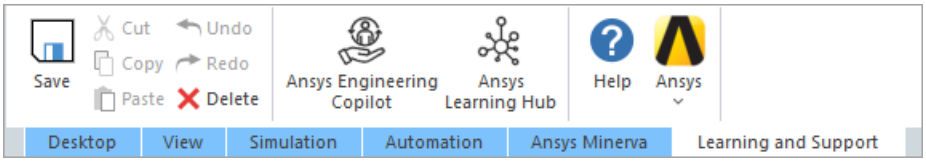
If you reduce the size of the Desktop window, the ribbon tabs become compressed. Fewer icons are shown, and available features are moved into drop-down menus rather than shown separately. The following example shows the **View** tab in a compressed state:



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

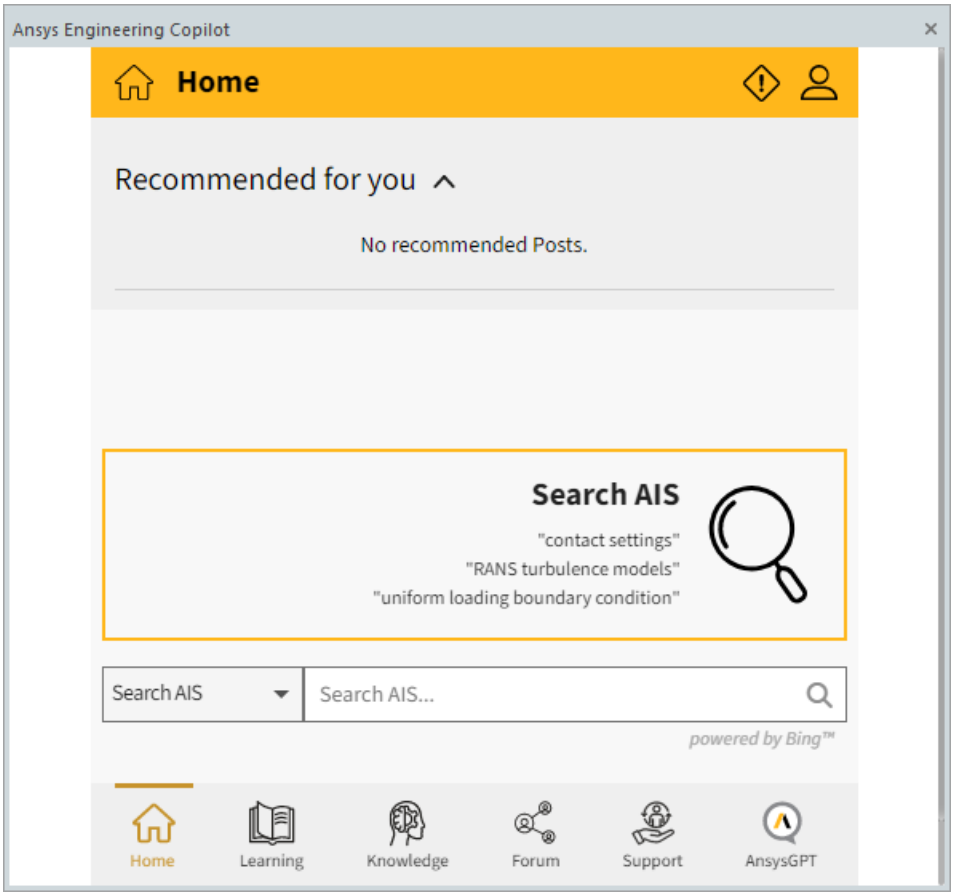
The **Learning and Support** tab provides easy access to technical advice, instruction, and examples on Ansys websites.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Ansys Engineering Copilot** is a single point access to Ansys Innovation Space resources, such as AnsysGPT, Support Cases, Courses, Knowledge, Forum, and more.

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



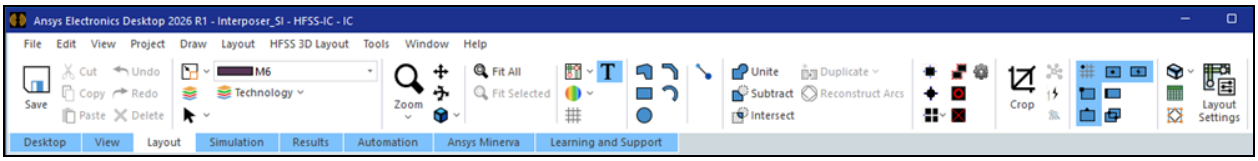
  - **Ansys Learning Courses** – Provide a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.
  - **Ansys Knowledge** – Provides expert curated knowledge materials from FAQs to tutorials on simulation topics.
  - **Ansys Forum** – Provides Ansys blog containing discussion and presentations from Ansys experts, partners and customers.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **AnsysGPT** – AnsysGPT is a Machine Learning (ML) Large Language Model (LLM) which has been trained in Ansys company information, including online help contents.
- **Ansys Learning Hub** opens a web page with subscription based access to, virtual and self-paced learning across the Ansys Software portfolio.
- **About Electronics Desktop** opens a dialog box 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).

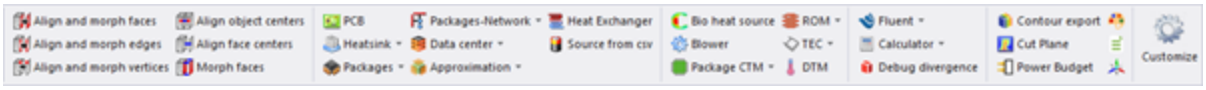


The general workflow remains the same in both environments. Refer to Switching to IC Mode.

Customizing the Automation Tab

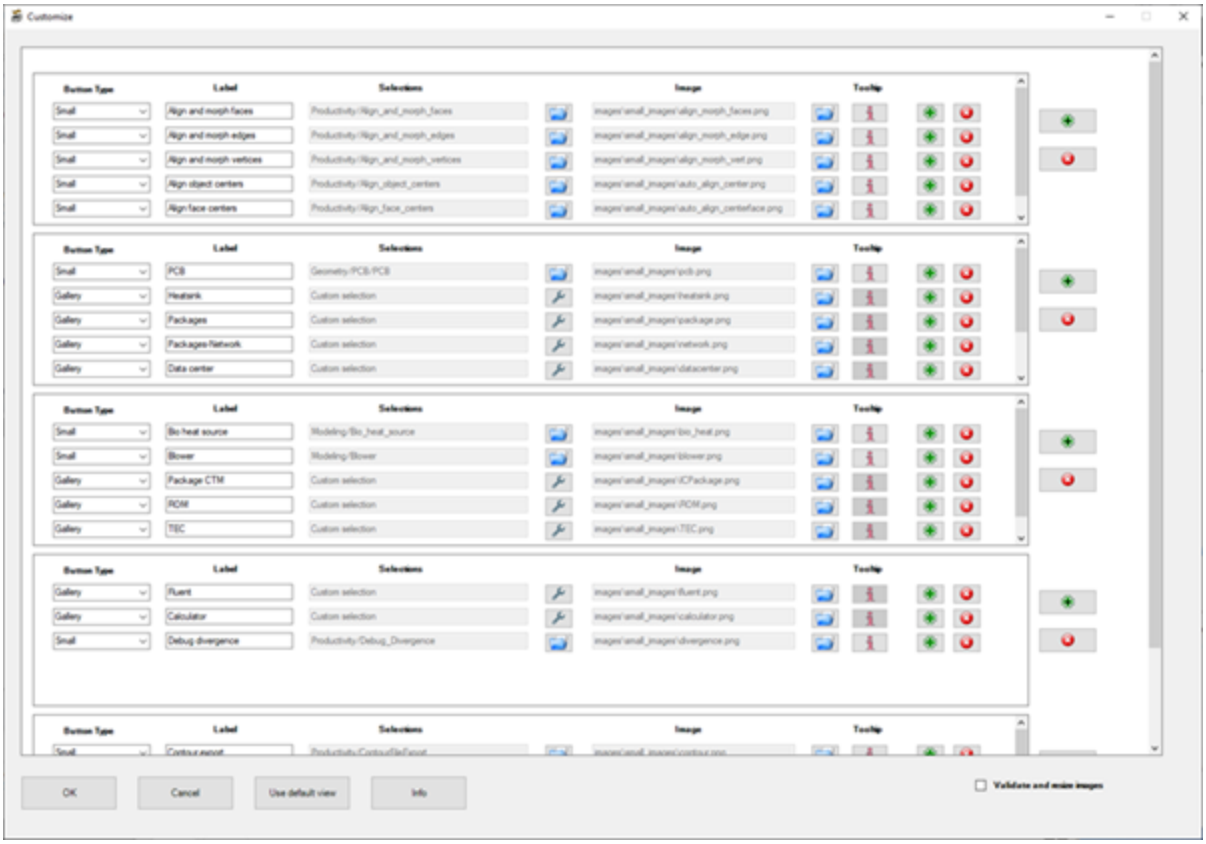
> **Note:** After clicking **OK** in the **Customize** dialog box to save changes to the **Automation** tab, you must insert a new Icepak design for the changes to be updated.

The **Automation** tab displays Icepak toolkit buttons.



You can customize the **Automation** tab to display buttons for your commonly used toolkits. Click **Customize** to open the **Customize** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



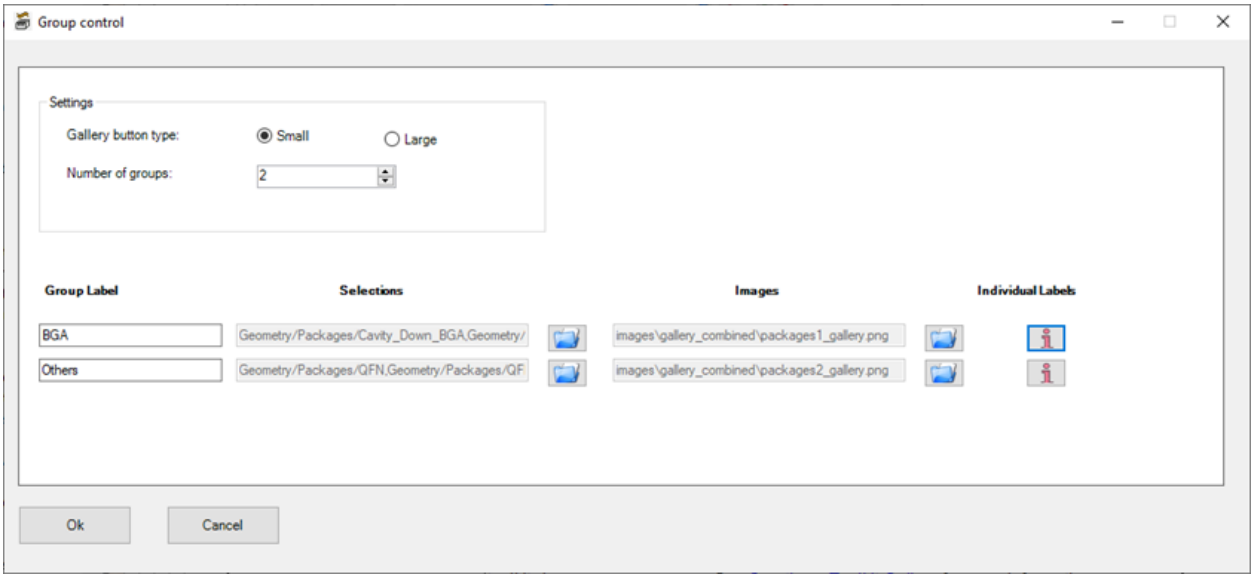
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add/Remove	Click **Add** to add individual toolkits to groups or to add a new toolkit group. Click **Remove** to remove individual toolkits or entire groups.
Button Type	From the **Button Type** drop-down list, select **Small**, **Large**, or **Gallery**.
Label	For **Label**, enter a name to appear on the toolkit or gallery button on the **Automation** tab.
Selections	For **Selections**, a small or large button links to one toolkit. Click the folder button and select a toolkit from the **Select scripts** dialog box. A gallery displays links to multiple toolkits in one or more groups. See "Creating a Toolkit Gallery" below for more information.
Image	For **Image**, click the folder button and select an image from the Select images dialog box. The image is displayed on the **Automation** tab.
Tooltip	For **Tooltip**, click the button and enter a description to be displayed when the cursor hovers over the toolkit button.
OK/Cancel/Use default view/Info	Click **OK** to save your changes and close the dialog box. Click **Cancel** to close the dialog box without saving. Click **Use default view** to restore the default selections. Click **Info** to display image size requirements for each **Button Type**.
Validate and resize images	Enable **Validate and resize images** to automatically scale the selected images.

Creating a Toolkit Gallery

When creating a toolkit gallery, click the wrench button in the **Customize** dialog box to open the **Group control** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Settings	
Gallery button type	For **Gallery button type**, select **Small** or **Large** to specify the button size.
Number of groups	For **Number of groups**, specify the number or groups to appear in the gallery.
Group Label	For **Group Label**, enter a name to appear on the toolkit group on the **Automation** tab.
Selections	For **Selections**, a small or large button links to one toolkit. Click the folder button and select a toolkit from the **Select scripts** dialog box. A gallery displays links to multiple toolkits in one or more groups.
Images	For **Images**, click the folder button and select an image from the **Select images** dialog box. The image is displayed on the **Automation** tab.
Individual Labels	For **Individual Labels**, enter names in the **Label list** dialog box. The labels should match the amount of scripts selected and be separated by commas and listed in the same sequence as the selected scripts.
OK/Cancel	Click **OK** to save your changes and close the dialog box. Click **Cancel** to close the dialog box without saving.

Ansys Electronics Desktop Windows

Ansys Electronics Desktop contains the following windows:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Project Manager
- Message Manager
- Progress
- Properties
- Component Libraries

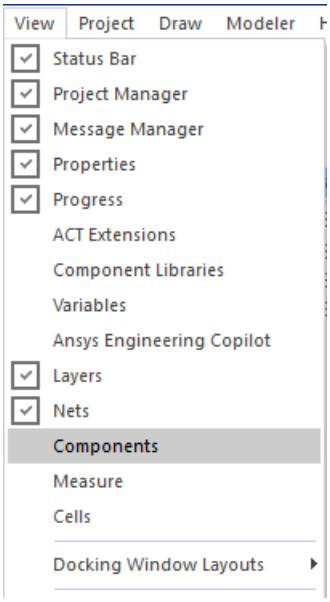
You can toggle the display of each window, auto-hide windows, or move and resize windows. Additionally, you can toggle the status bar or add ACT Extensions (Windows only).

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:

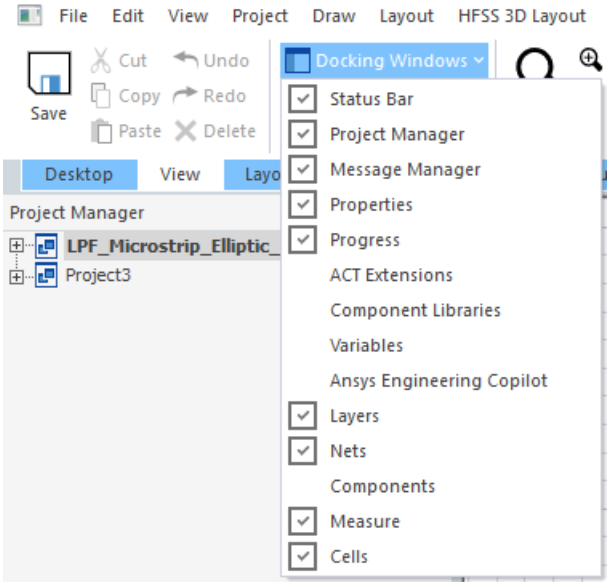
- Click a window to show or hide it.



From the **View** tab:

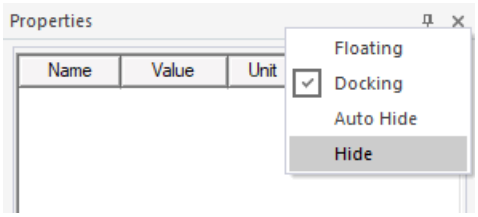
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

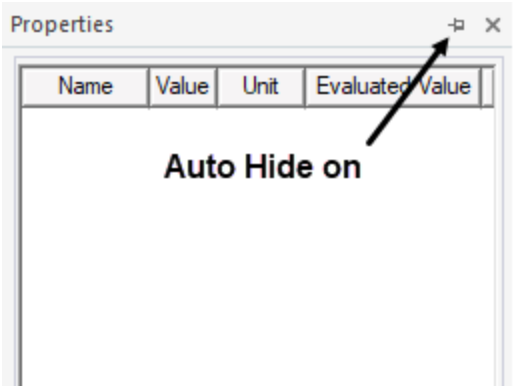
To automatically hide a window until the cursor is hovered over it:

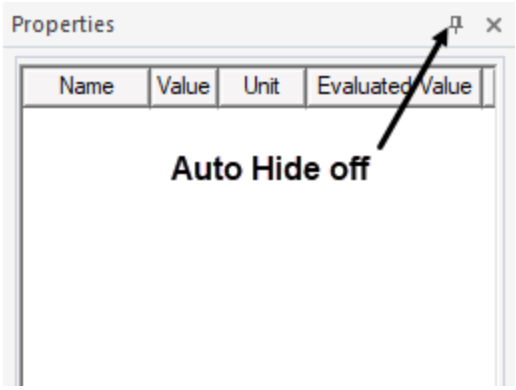
- Click the pin icon in the window title bar, or
- Right-click in the window title bar and select **Auto Hide**.

> **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

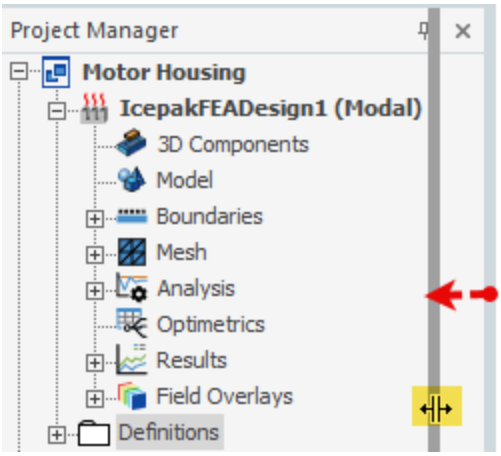
When a window is in Auto Hide mode, the pin icon flips on its side. Click the icon again to disable Auto Hide.





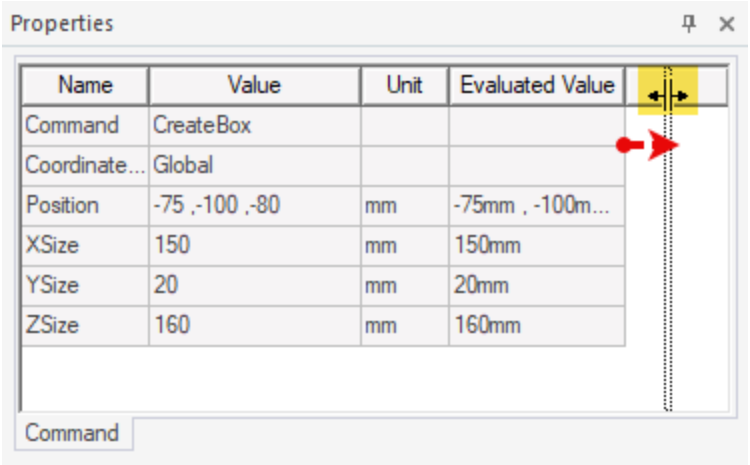
Moving and Resizing Windows

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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: 

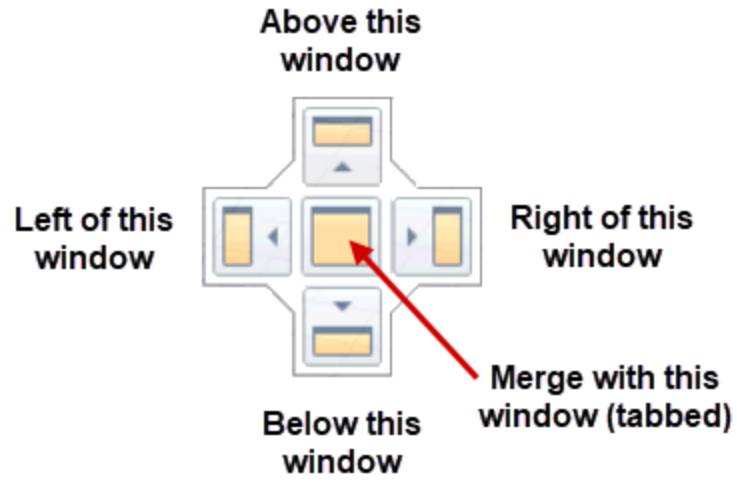
Right edge of user interface: 

Top of user interface: 

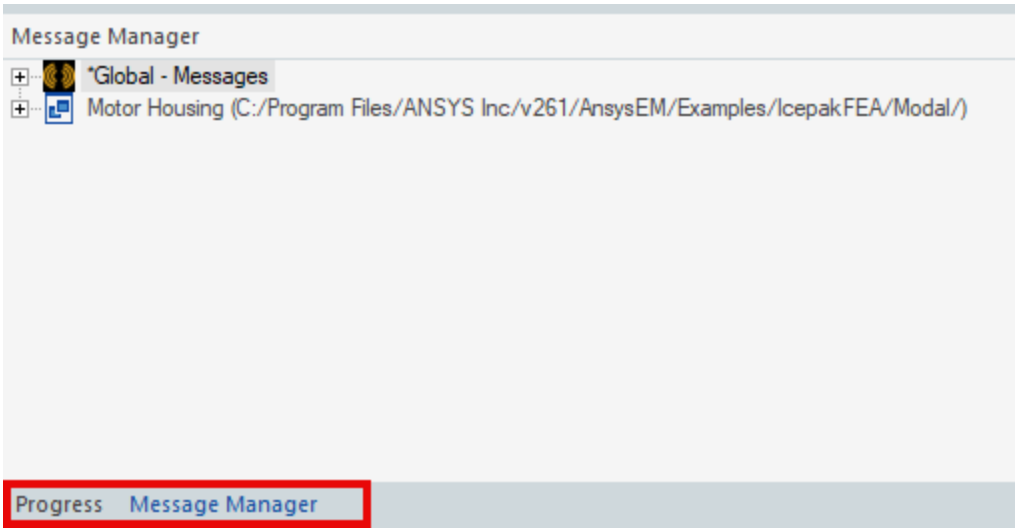
Bottom of user interface: 

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

Ansys Electromagnetics Suite 2026 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 over the middle icon, the two windows are merged, with tabs at the bottom to select which one to view:



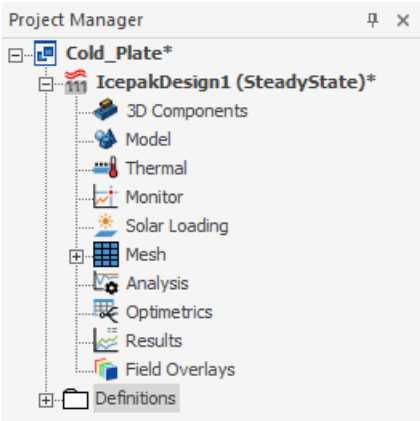
If the middle icon does not appear, it's because tabbed windows are not supported by one of the windows. For example, you cannot merge any window with the Modeler window.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

- For an Icepak design, the project includes a geometric model, its thermal boundary conditions and material assignments, monitor points, mesh regions, analysis setups, optimetrics, solution results, post-processing information, and relevant definitions. Any 3D Components appear under the 3D Components icon.

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.

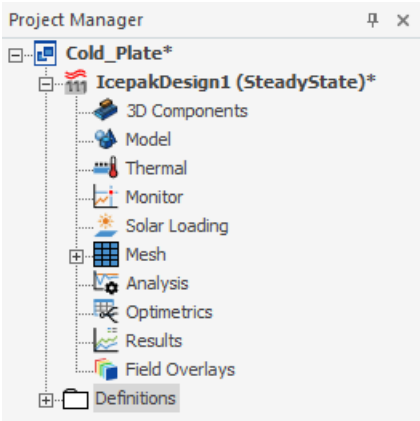
- Right-click in the toolbars area on the desktop, and then click **Project Manager** on the shortcut menu.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

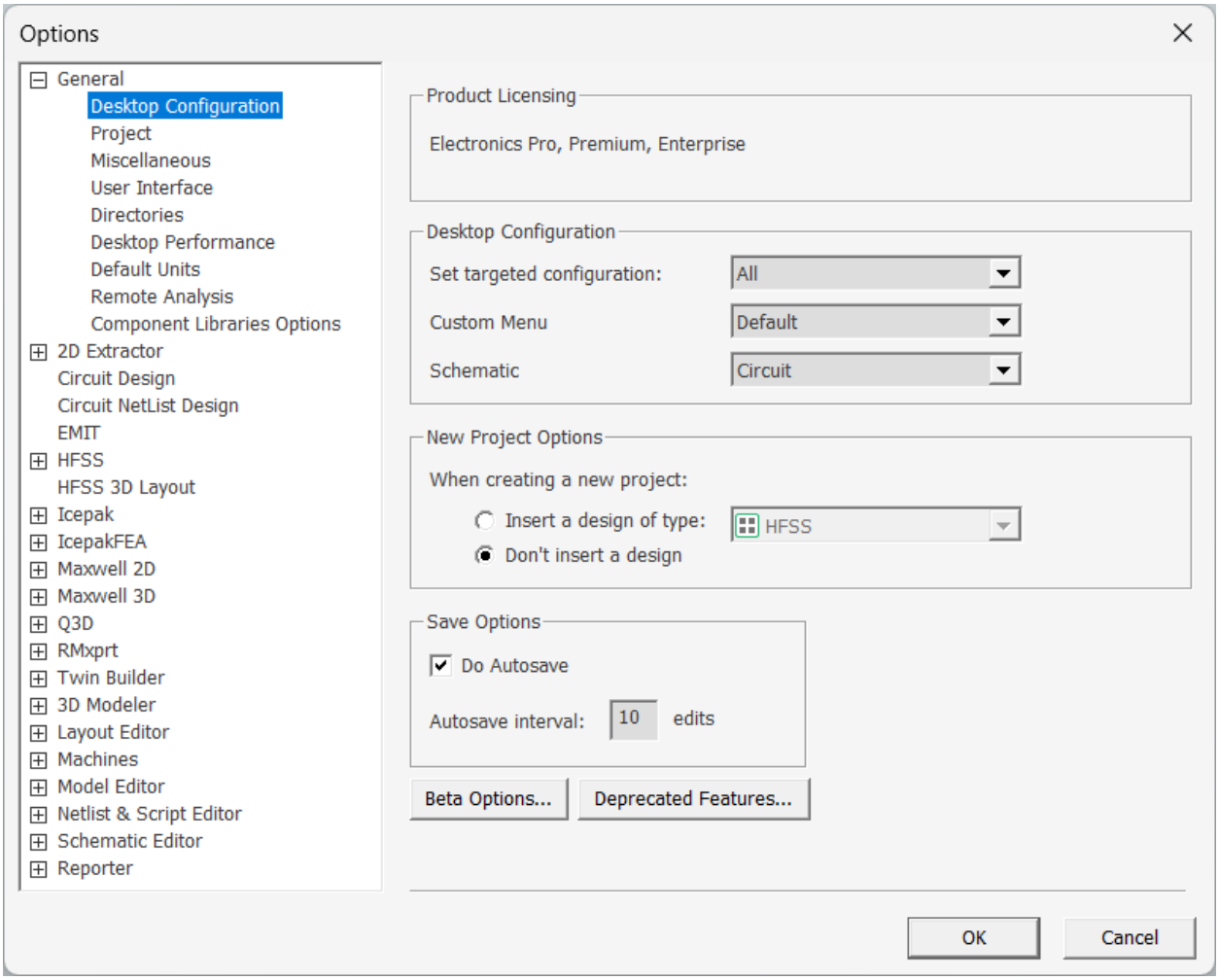
Setting the Project Tree to Expand Automatically

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

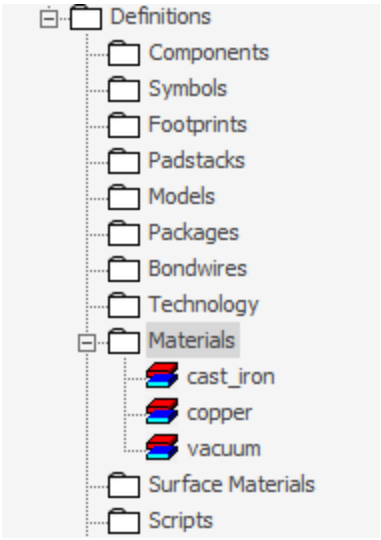


2. Click **User Interface**.
3. In the **Project Tree Visualization Options** area, check the **Expand Project Tree on Insert** check box.
4. Click **OK**.

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 2026 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.
Thermal	Displays the thermal boundary conditions assigned to an Icepak design, which specify the field behavior at the edges of the problem region and object interfaces.
Monitor	Displays the point and face monitors created in an Icepak design.
Solar Loading	Displays the solar loading assignments in an Icepak design.
Initial Temperature	Displays the initial temperature assignments in a transient Icepak design.
Mesh	Displays the mesh regions created in an Icepak design.
Analysis	Displays the solution setups for an Ansys Electronics Desktop design. A solution setup specifies how Ansys Electronics Desktop will compute the solution.
Optimetrics	Displays any Optimetrics setups added to an Ansys Electronics Desktop design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results	Displays any post-processing reports that have been generated.
Field Overlays	Displays fields overlay plots, 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 Icepak, 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. The following icons appear next to a message to indicate information, warnings, errors, or actions:

 Indicates an informative message.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**.

Automatically Expanding the Message Manager Tree

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

Ansys Electromagnetics Suite 2026 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**.

   The **Options** window appears.

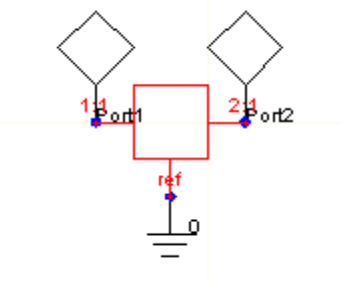
2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Ensure that new messages are visible in the Message Window Tree** check box.
4. Click **OK**.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Tools** > **Options** > **General Options**.
3. Check the **Show message window on new messages** check box.

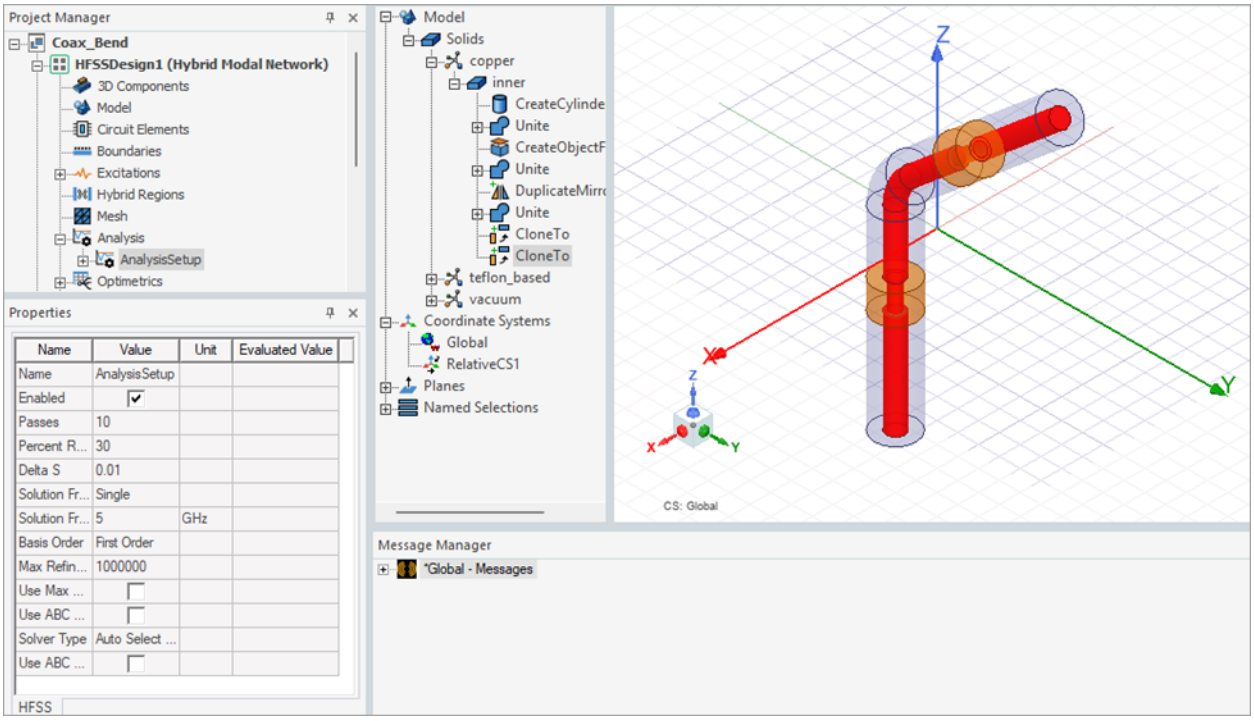
The **Message Manager** window will re-open when Ansys Electronics Desktop reports errors, warnings, or successful completion of any simulation.

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 by clicking on **View** > **Properties**. 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.

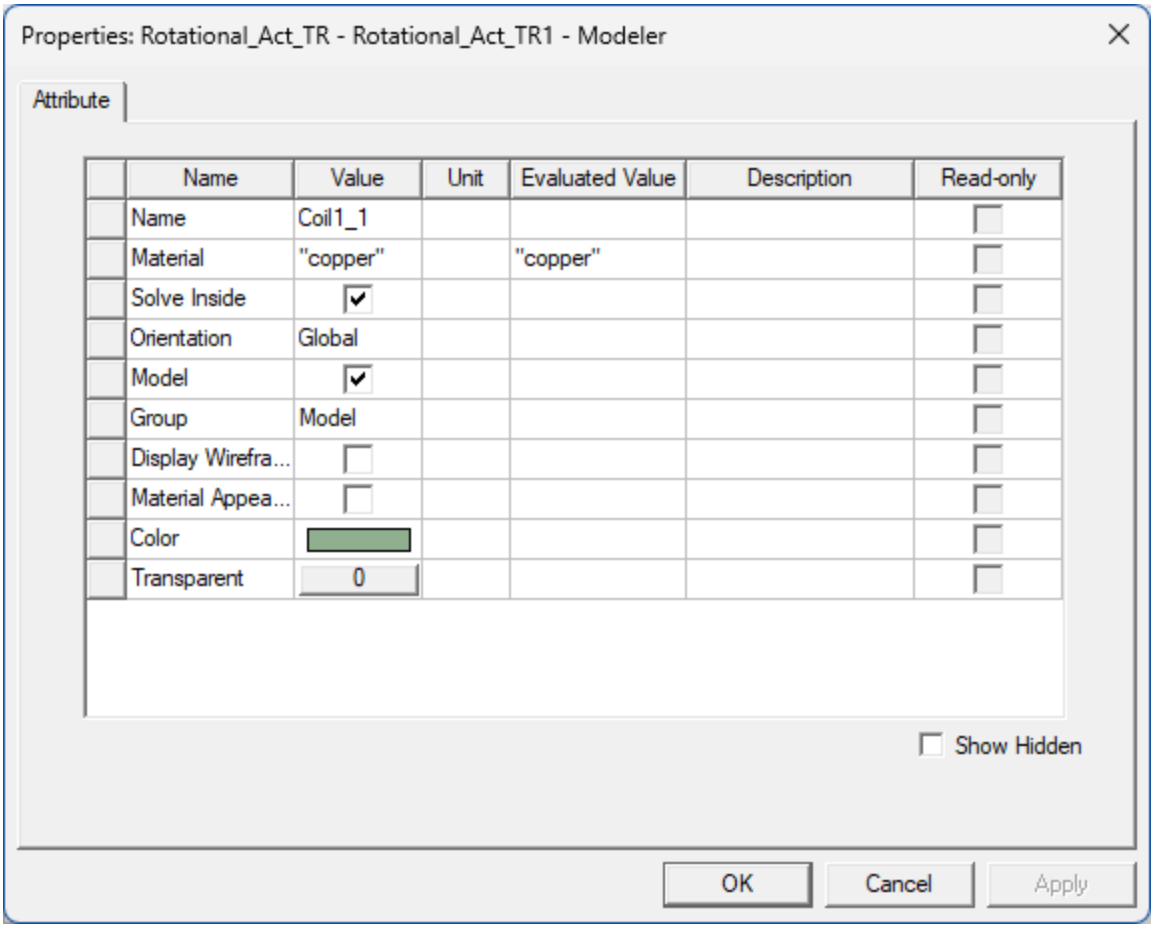


Regardless of whether or not you display a docked **Properties** window on the desktop, you can open an undocked **Properties** dialog box for any item in the project tree or history tree by double-clicking the item or right clicking and selecting **Properties**. Note that when the properties of an object are opened this way, the dialog box may have different tabs, and this window is not

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dockable. To distinguish between these two displays, the documentation will refer to the dockable properties pane to the Properties window, and the free-standing, undockable window as the Properties dialog box.

An example of the Properties dialog box for a solid model object follows. For solid model entities and sheets, the Properties window and the Properties dialog box are nearly identical.



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.

Ansys Electromagnetics Suite 2026 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

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.

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 Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

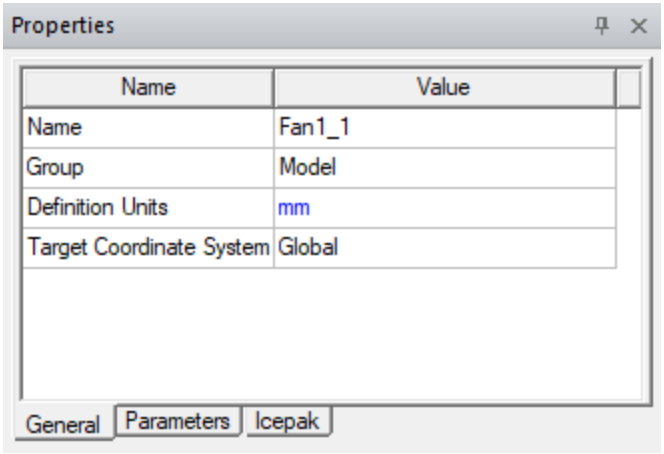
Properties Window for Schematic and Layout Editors

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

- The Param Values Tab
- The General Tab
- The Parameters Tab
- The Icepak Tab

General Tab

The **General** tab in the **Property** window of the Ansys Electronics Desktop lists the selection's name, group, unit definition, target coordinate system, and other data.



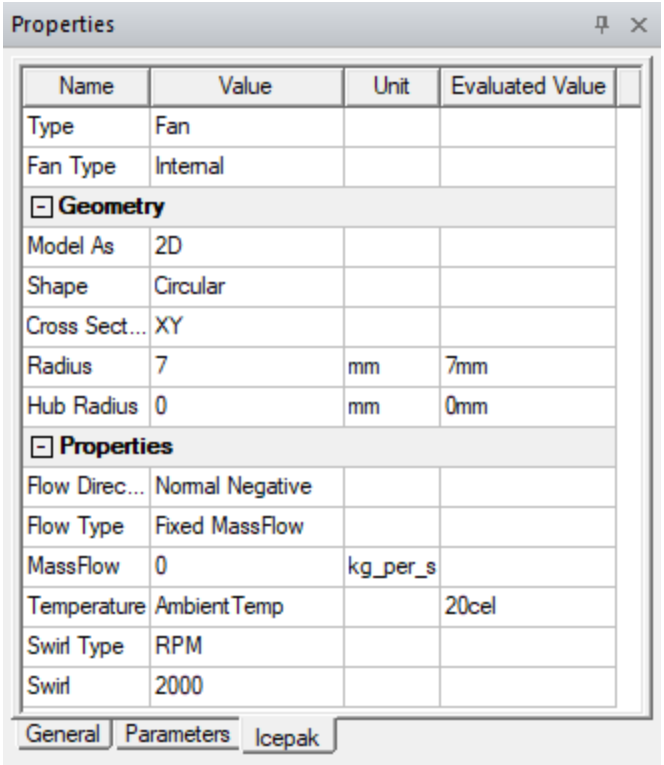
Parameters Tab

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

Icepak Tab

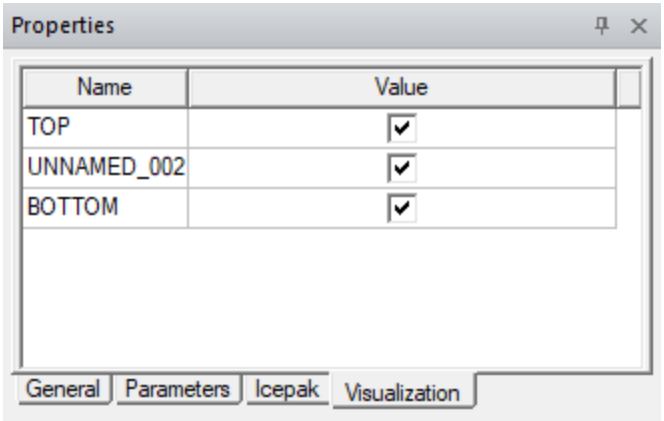
The **Icepak** tab lists a 3D component's **Type** and other properties specific to the component type. For example, the **Icepak** tab for a fan component lists the **Model As** definition, **Shape**, and **Fan Type** as well as geometry and other properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Visualization Tab

The **Visualization**tab displays information about a PCB's ECAD layers. The names are displayed in the **Name** column. Select the check box in the **Value** column to display the ECAD information for the corresponding layer.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Closing the Properties Window

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

For more information, see the Basic View Menu.

Properties Dialog Box

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

- Attributes (if a solid model entity or sheet is selected), which is the same as the Properties window.
- Component (displays if the circuit component definition can be edited)
- Parameters
- Icepak
- Visualization (PCBs)

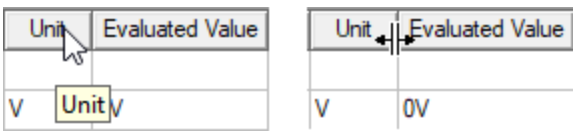
Opening the Properties Dialog Box

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

- Select the object, and select **Edit** > **Properties**.
- To open the **Properties** dialog box for a component, double-click the component.
- Right-click the object and select **Properties**.

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

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



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Properties Dialog Box to Open Automatically

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

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

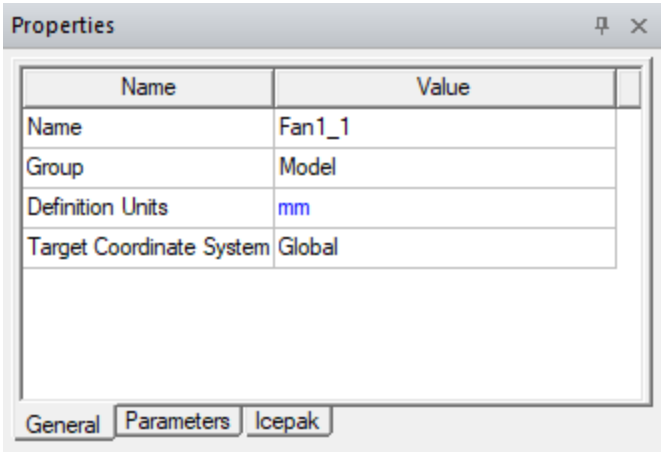
   The **Options** window appears.

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

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

General Tab

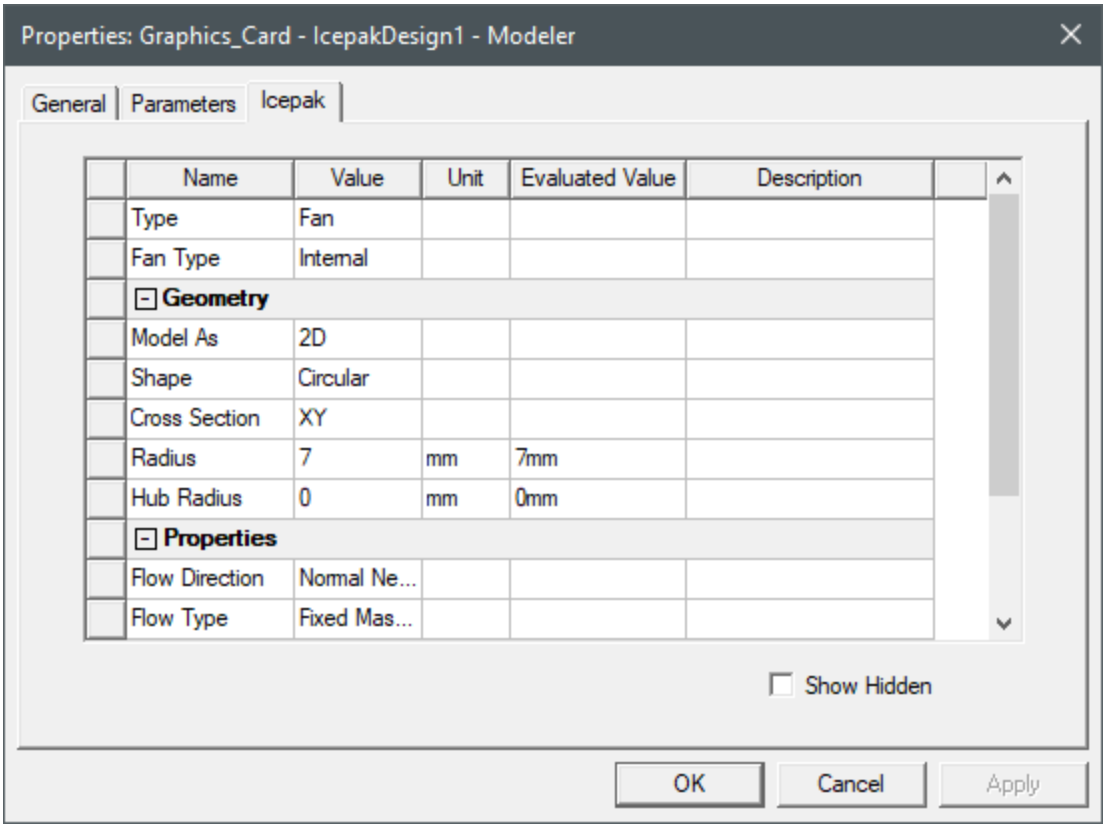
The **General** tab in the **Property** window of the Ansys Electronics Desktop lists the selection's name, group, unit definition, target coordinate system, and other data.



Icepak Tab

The **Icepak** tab lists a 3D component's **Type** and other properties specific to the component type. For example, the **Icepak** tab for a fan component lists the **Model As** definition, **Shape**, and **Fan Type** as well as geometry and other properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



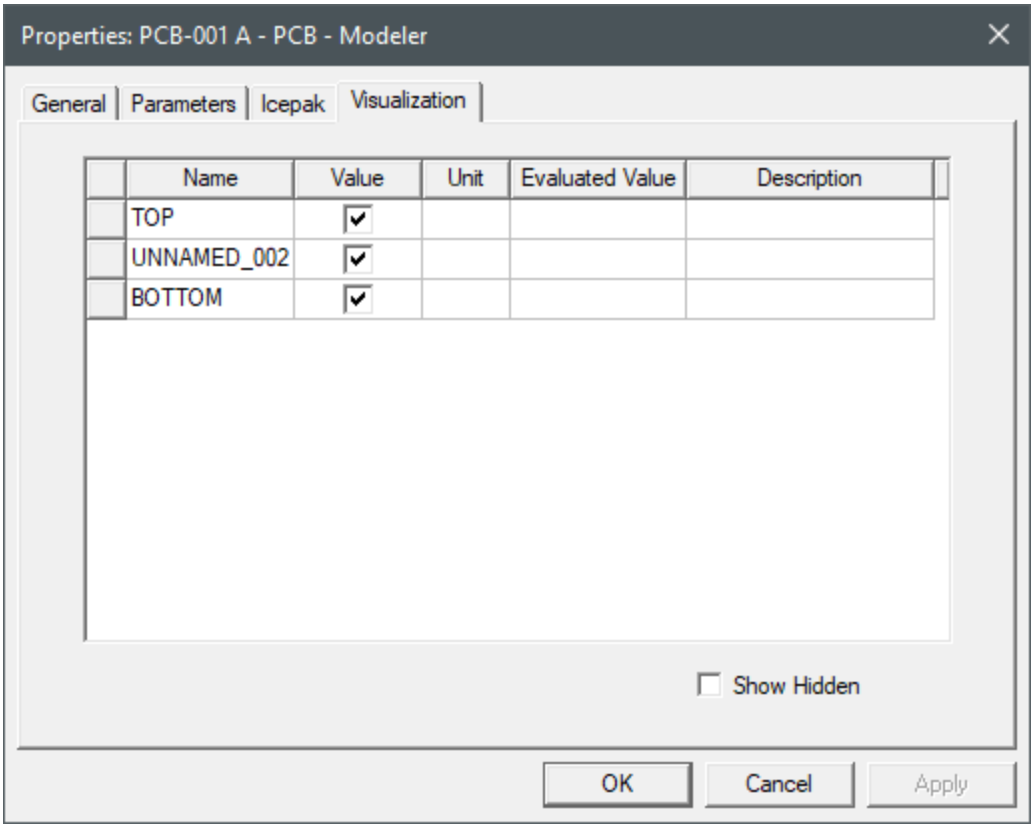
Parameters Tab

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

Visualization Tab

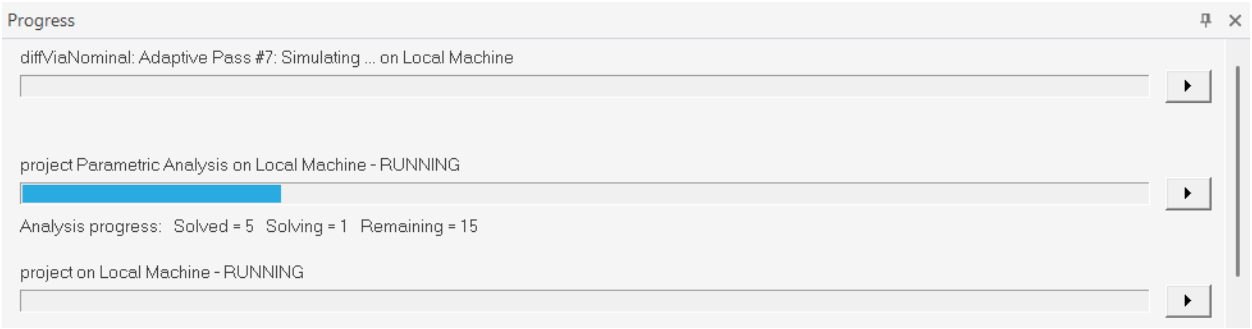
The **Visualization**tab displays information about a PCB's ECAD layers. The names are displayed in the **Name** column. Select the check box in the **Value** column to display the ECAD information for the corresponding layer.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



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

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

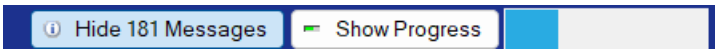


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

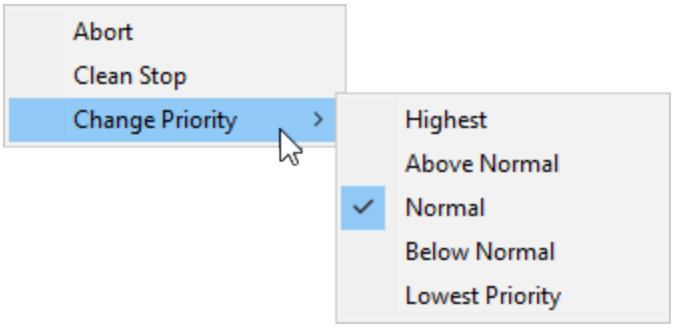
When more than one progress bar is active, the top progress bar is represented on the status bar with a progress indicator.



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.

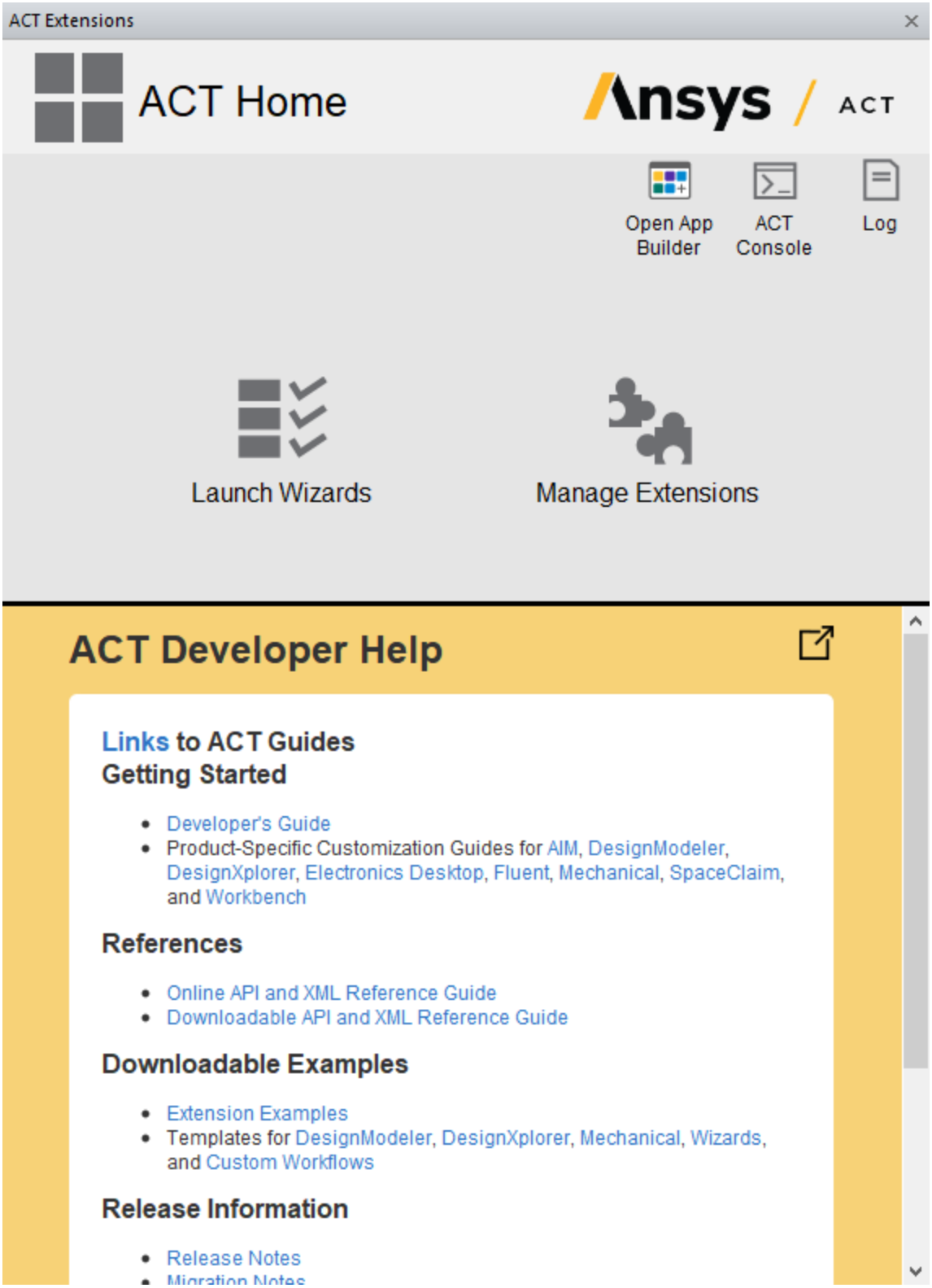


ACT Extensions Window

> **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.

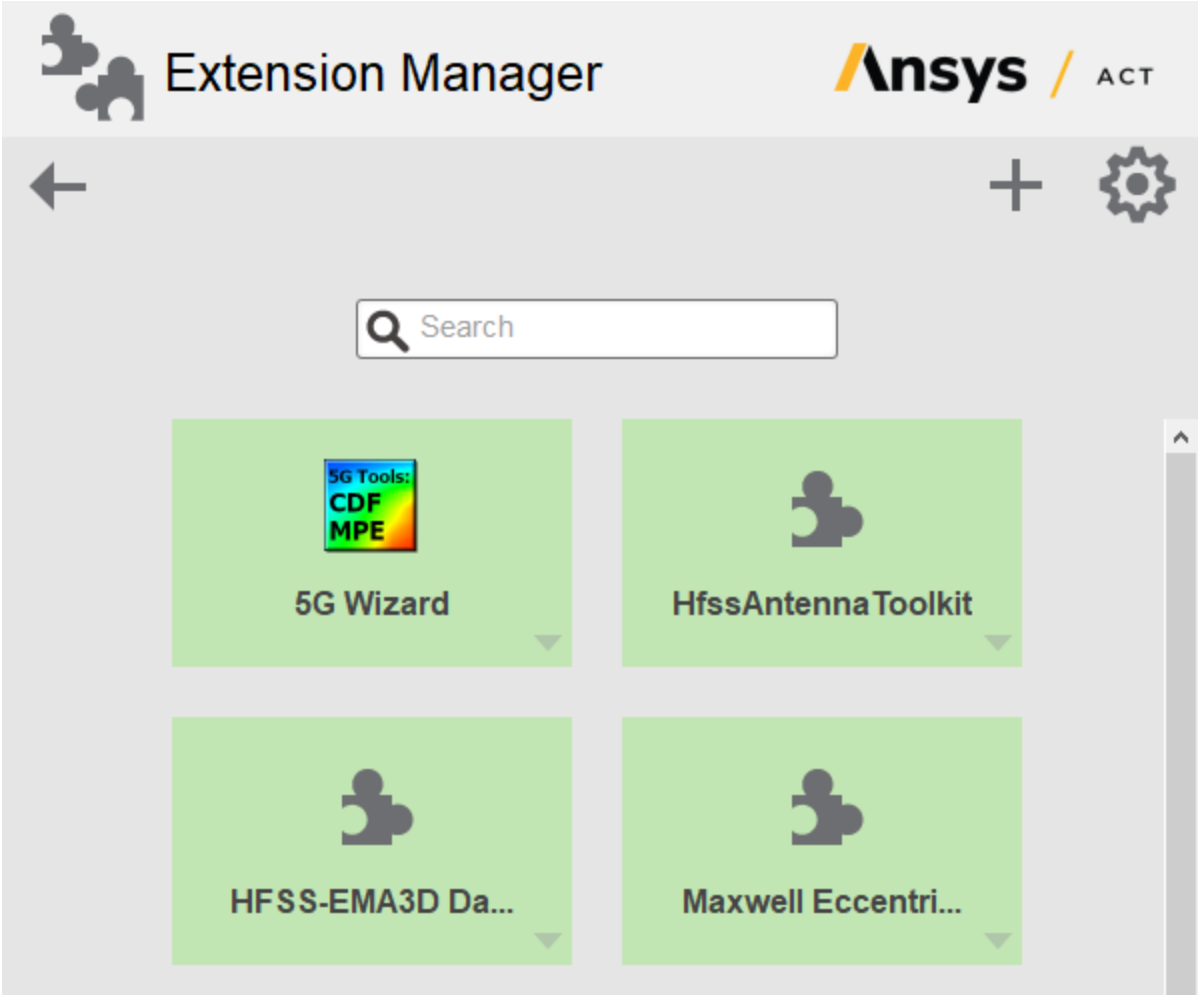
The **View** > **ACT Extensions** command opens the ACT Extensions window for toolkit extensions implemented via Ansys ACT. Depending on how you size and arrange the desktop, the appearance may vary. You may choose to detach and size the window to suit your needs.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



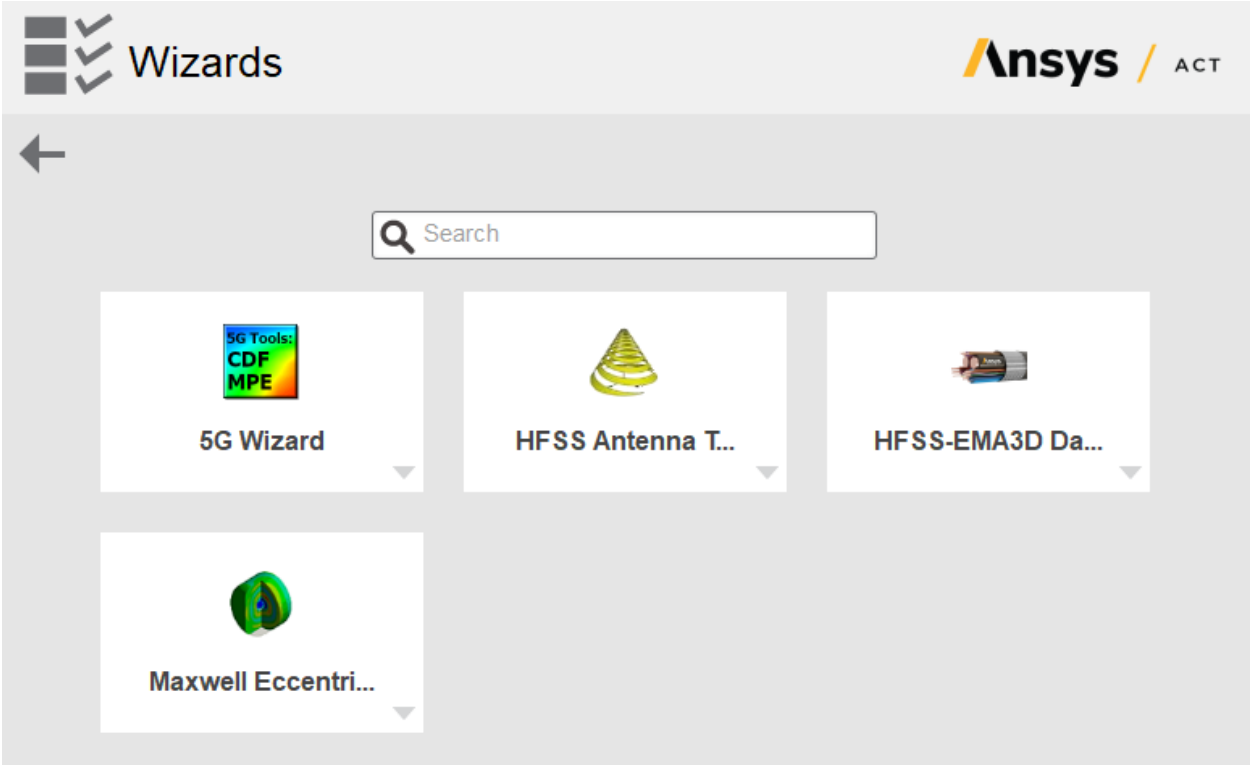
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



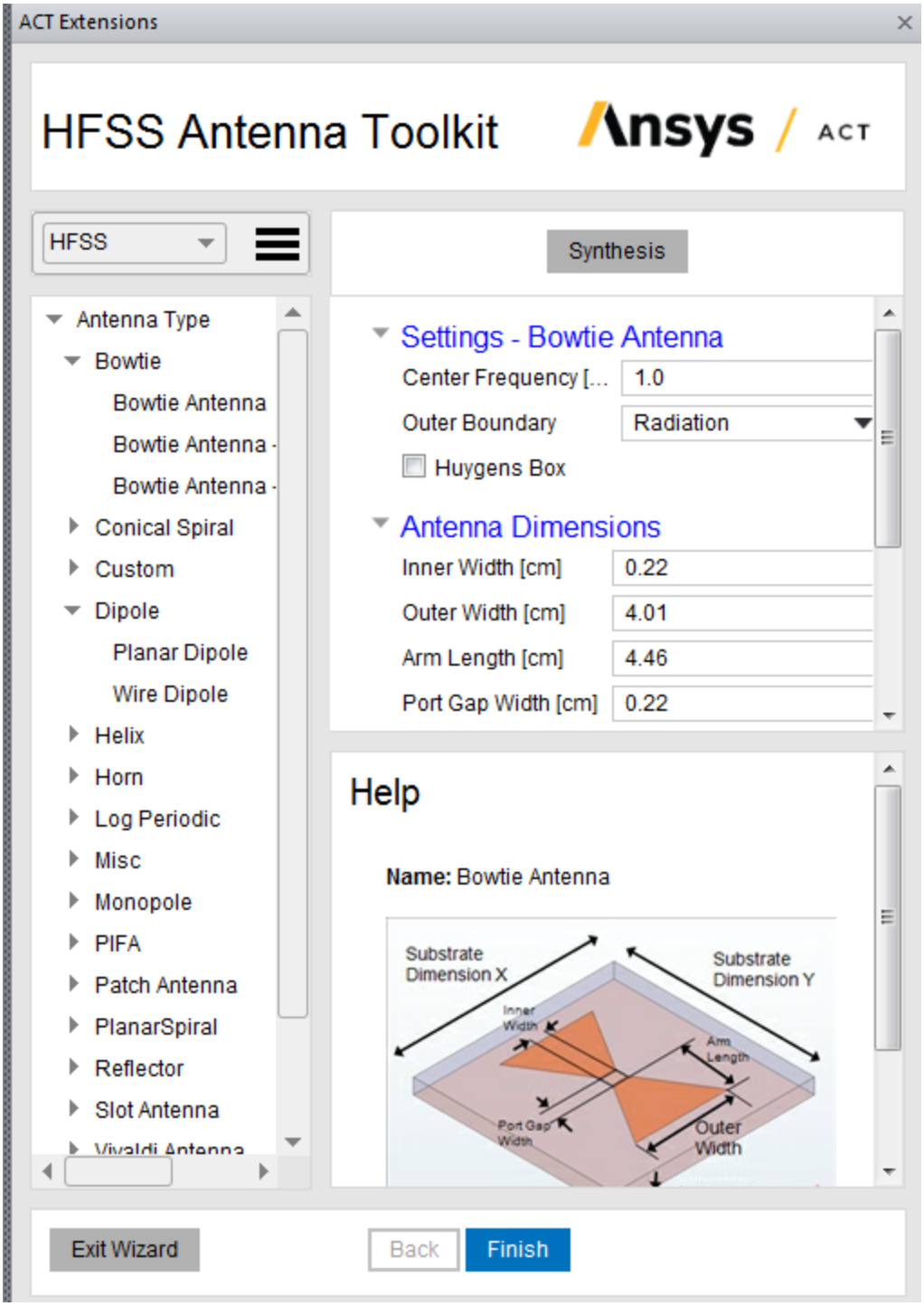
Click the **Launch Wizards** icon to load existing toolkits for activation. If there are many available extensions, you can use the **Search** bar. Search is not case sensitive.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Bar

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

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

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

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

Order of Menus

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

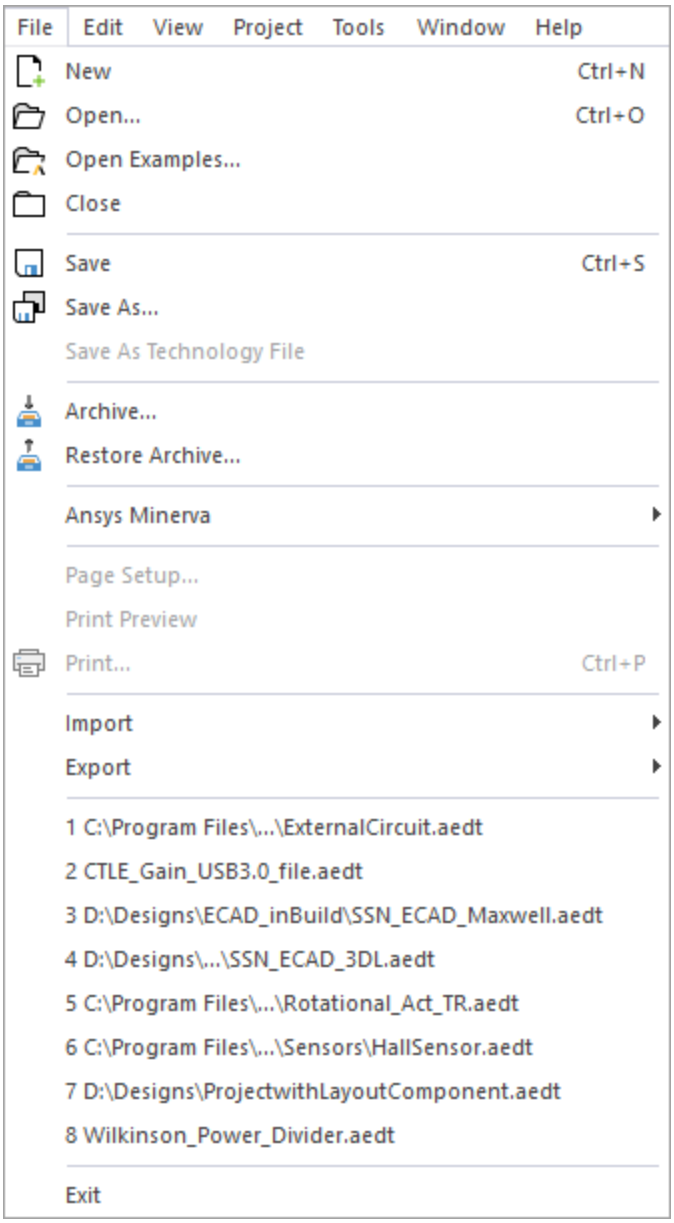
File Edit View Project Draw [Modeler | Layout | Schematic](1) <*Design-Type*>(2) Tools Window Help

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

These menus are described in the following sections.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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.

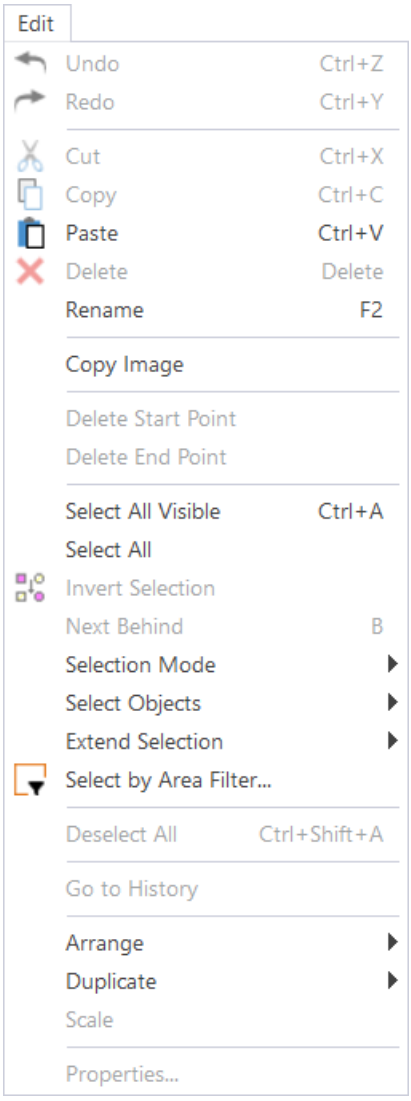
These menus include:

- 3D Modeler

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

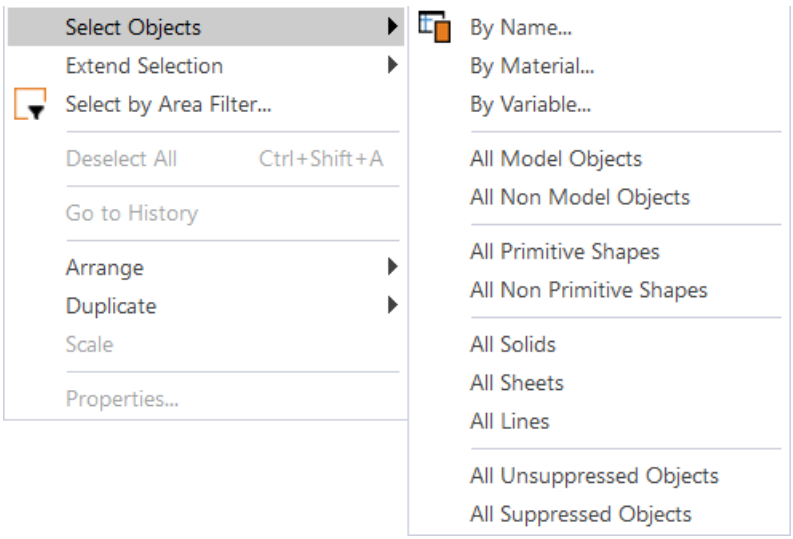


Commands will be available or not available for clicking depending on what is currently selected or on whether any operations have been performed, undone, or redone. The 3D Modeler's Edit menu contains the following options:

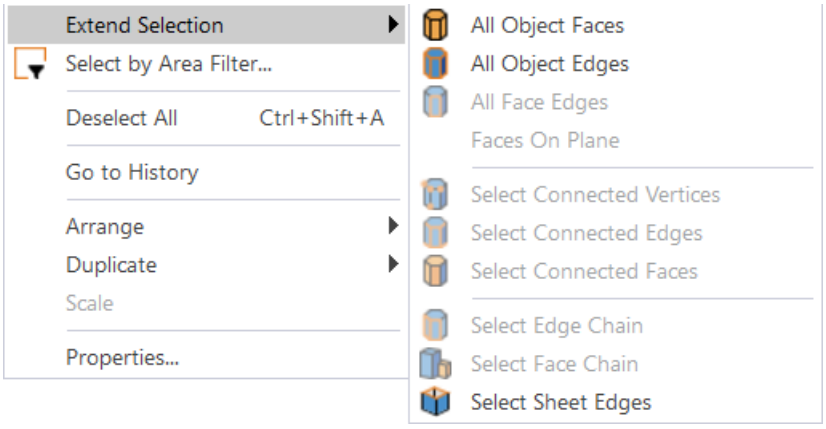
- **Undo** – allows you to undo the last action, which is specified after the word "Undo."
- **Redo** – allows you to redo an undone action, which is specified after the word "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste elements.
- **Delete** – deletes a selected element.
- **Rename**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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.



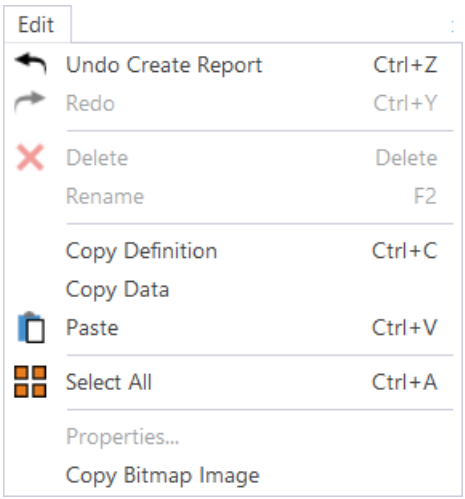
- **Select by Area Filter** – opens the **Select By Area Filters** window.
- **Deselect All** – deselects all selected objects.
- **Go To History** – selects the History Tree entry for the selected object.
- **Arrange** – allows you to move, rotate, or mirror the selected object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

View Menu

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

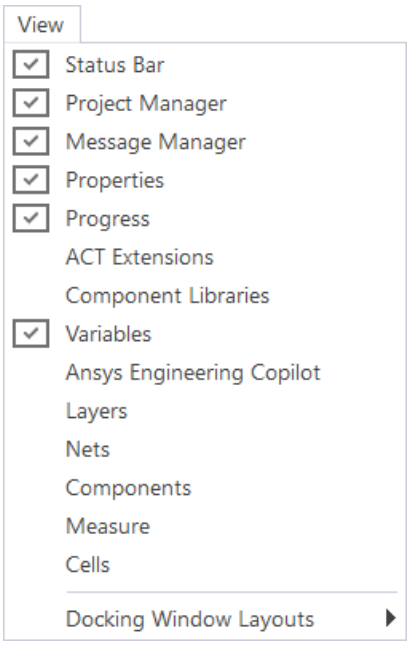
These menus include:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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:

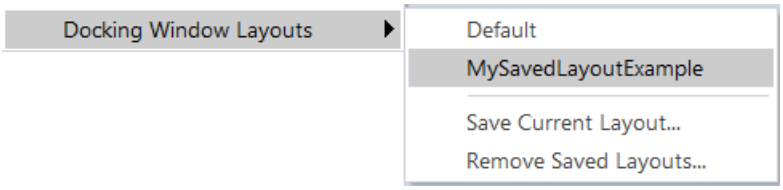


- The check boxes toggle the display of the **Status Bar**, **Message Manager Window**, **Project Manager**, **Properties Window**, and **Progress Window**. You can enable the display of **Component Libraries**, **Layers**, **Nets**, **Components** and **Variables** windows.
- The **ACT Extensions**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

> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

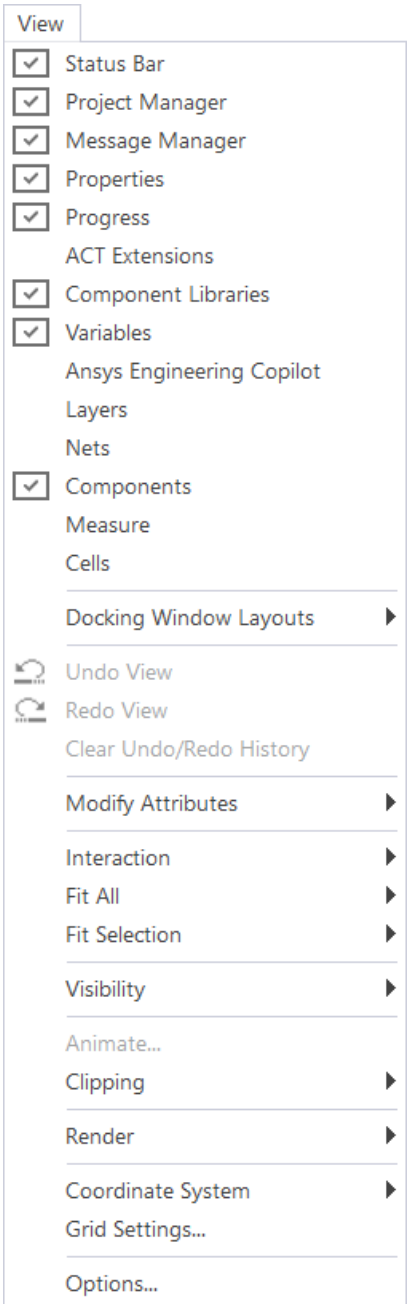


- The **Variables** option opens the **Project** and **Design** **Variables** window, allowing you to set variables.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

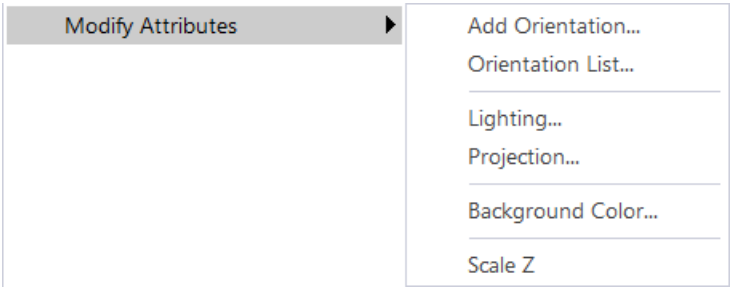


The **View** menu includes the options available on the Basic View Menu. The following additional options are available when a *Modeler* window is active. Note that a number of these options also appear in the View branch of the shortcut menu that appears when you right-click in the Modeler window.

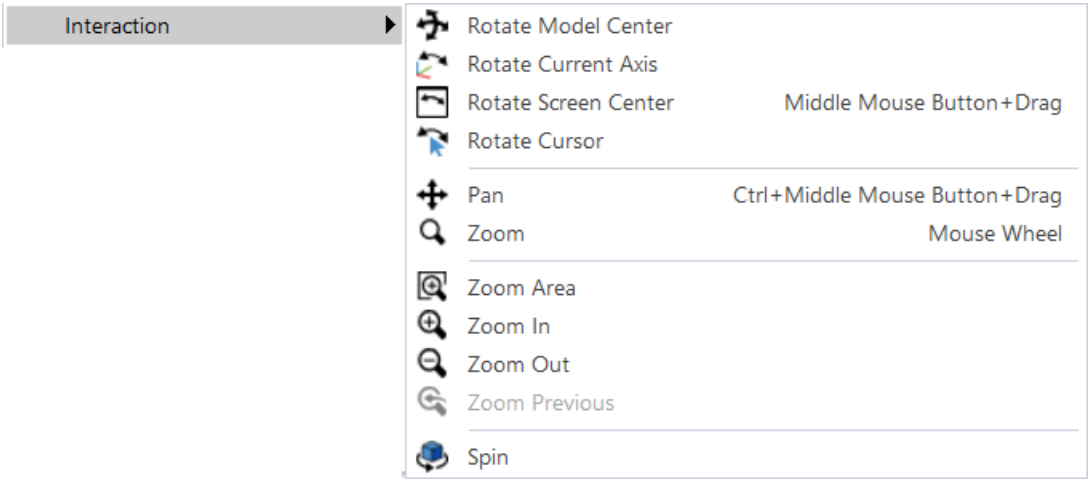
- **Undo View** / **Redo View** / **Clear Undo/Redo History** – allow you to change the view in the design area, based on view history.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

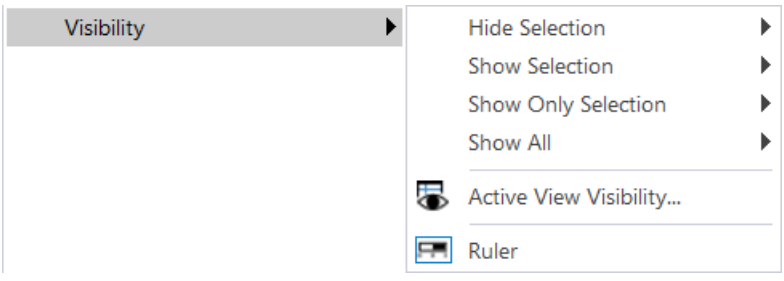
- **Modify Attributes** – offers options to **Add Orientation**; view the **Orientation List**; change **Lighting**, **Projection**, or **Background Color**; and **Scale Z**.



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



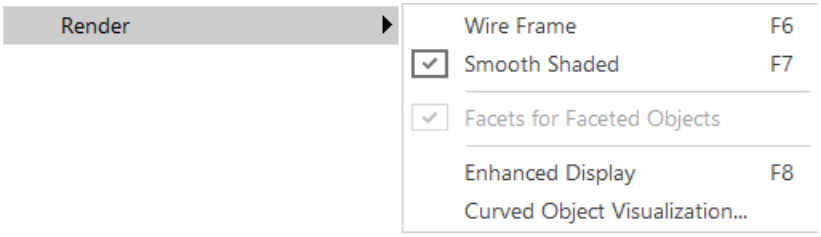
- **Fit All** – allows you to fit all objects to the view window.
- **Fit Selection** – allows you to fit selected objects to the view window.
- **Visibility** – offers options to show and hide selections, objects, shapes, and the ruler.



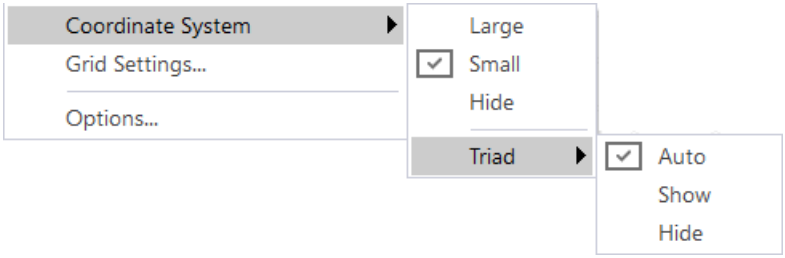
- **Animate** – opens the **Create Animation Setup** window, so that you can begin creating animations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

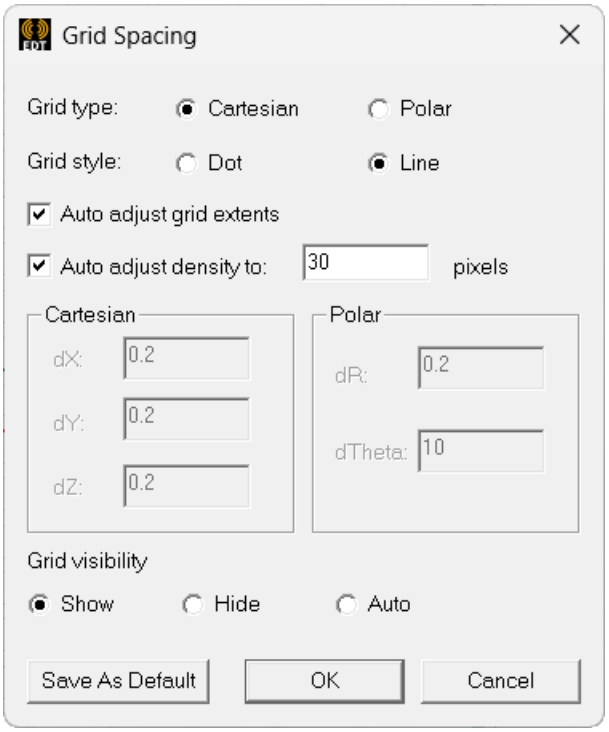


- **Coordinate System** – offers options to change how the coordinate system displays in the design area.



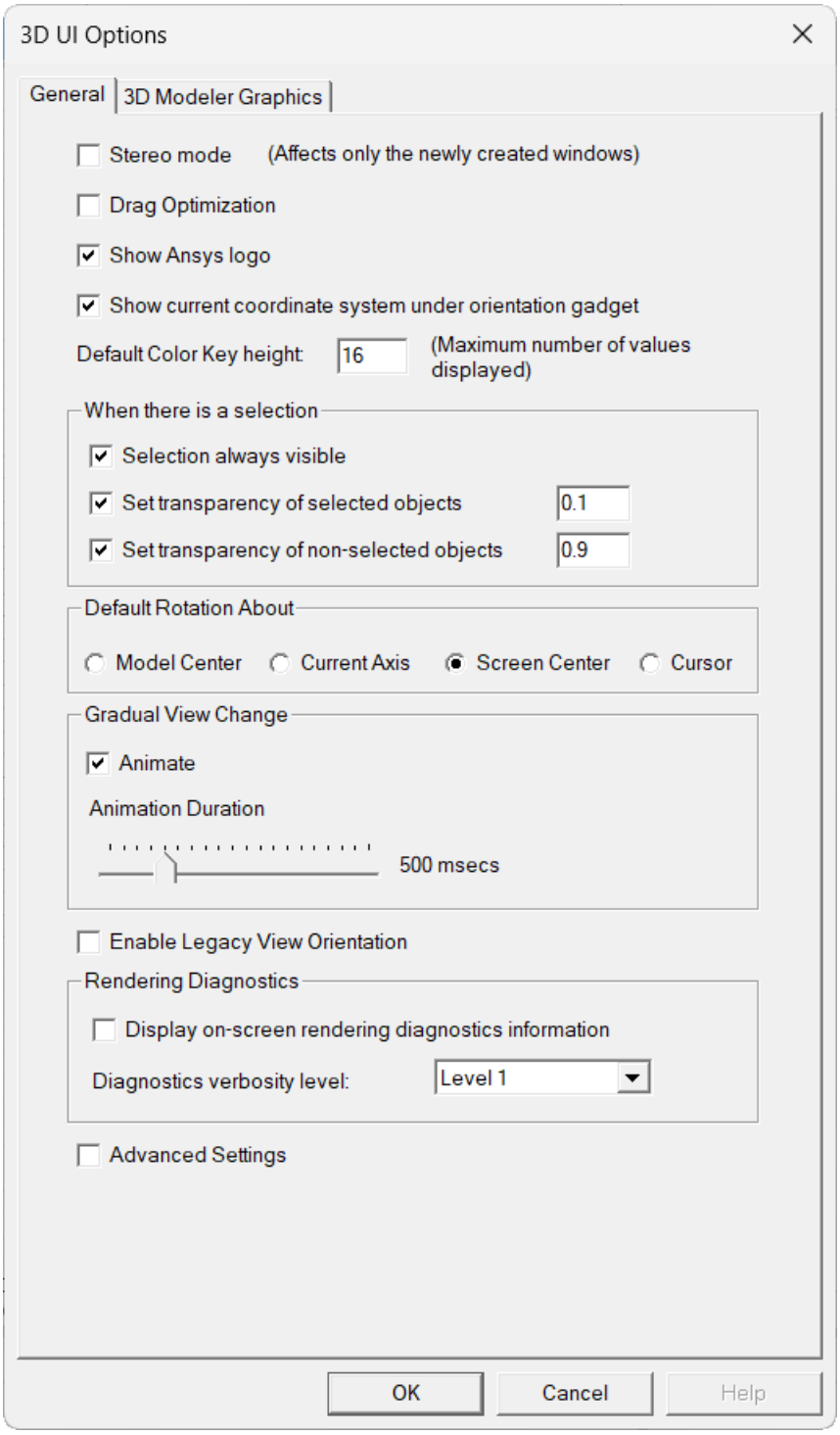
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

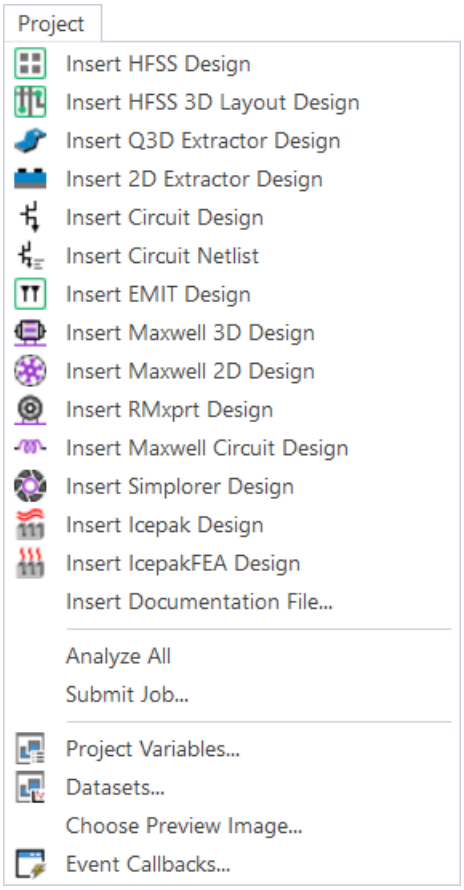
- **Options** – opens the **3D UI Options** window, where you can change additional settings.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project Menu

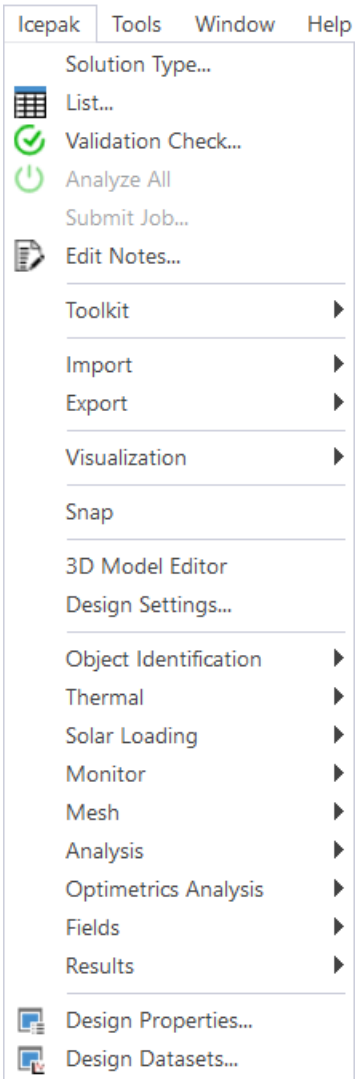
To create (insert) a new design, you must first open a new project folder by selecting **New** from the **File** menu. Then select a project to insert from the Project menu. Or you can click the icon corresponding to the type of design you wish to create.



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.

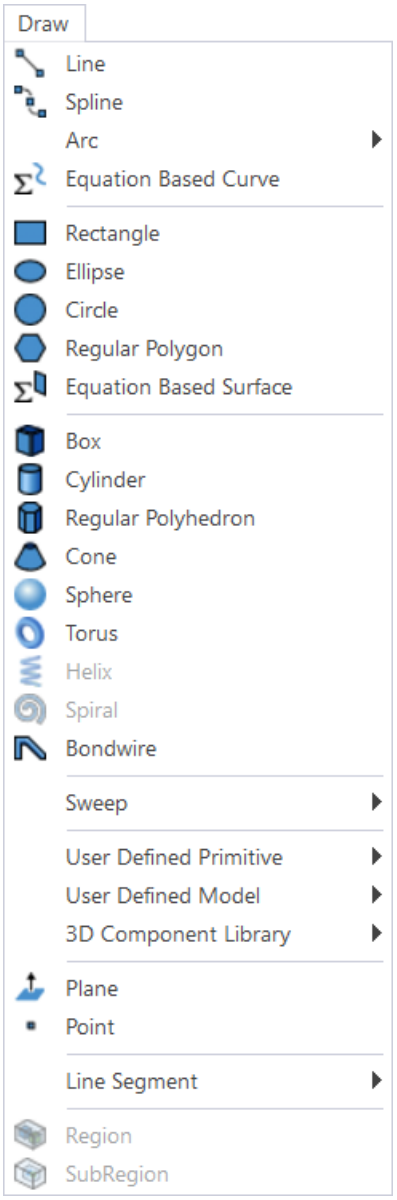
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Layout Editor Draw Menu

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

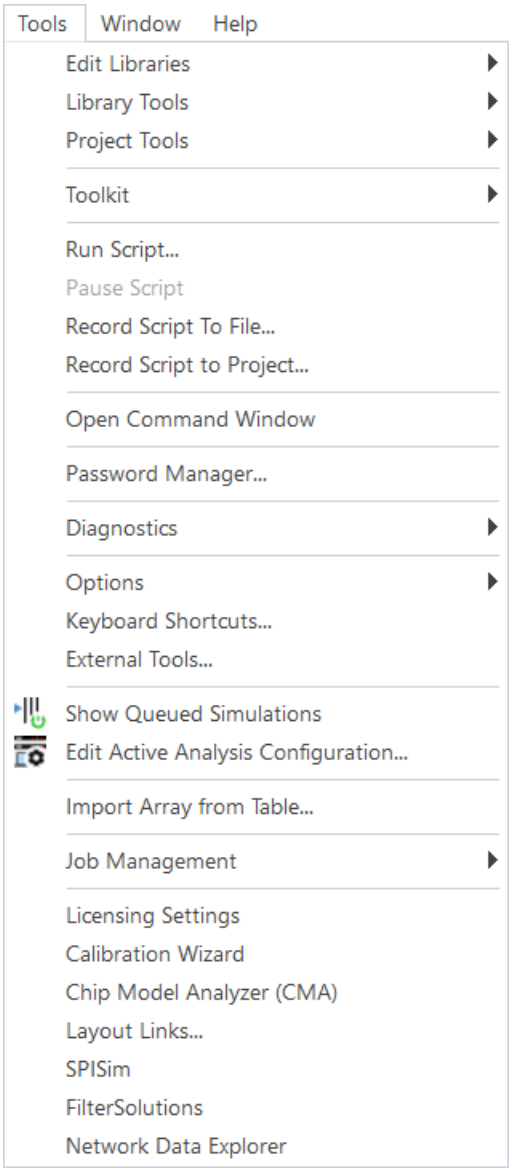
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tools Menu

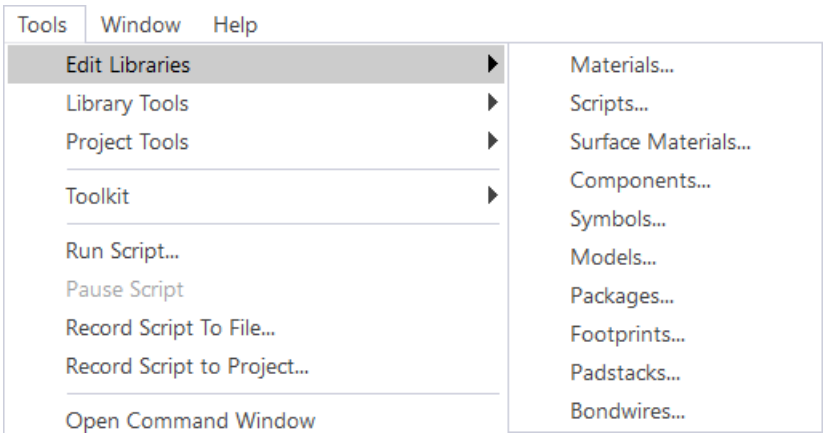
The **Tools** menu contains operations that are common to all design types:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

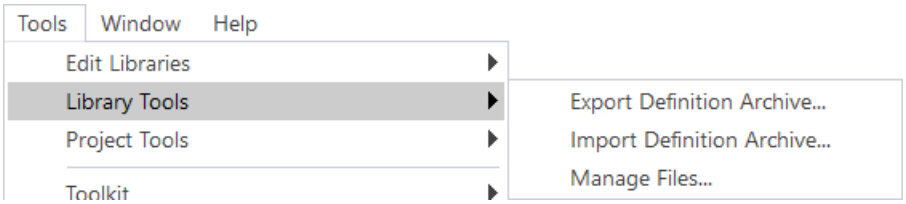


- Click **Tools** > **Edit Libraries** to view commands allowing access to the libraries available for the various Ansys Electronic Desktop solves, such as Materials, Scripts, and Components, Symbols and Models for Simplorer, Icepak, and Circuit.

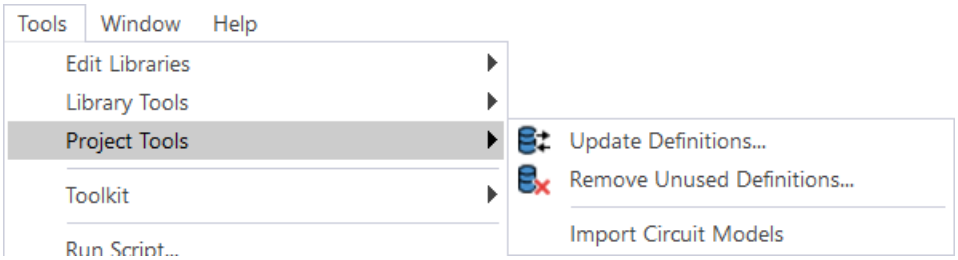
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Click **Tools** > **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.

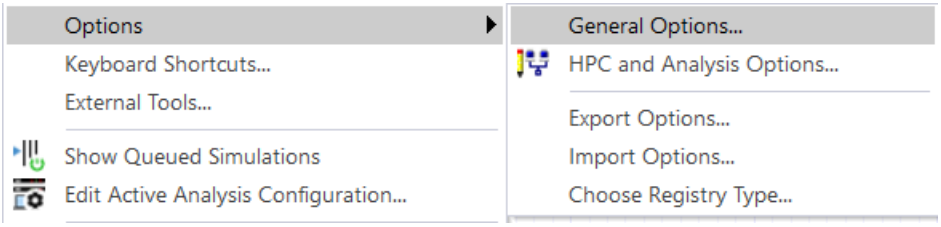


- Click **Tools** > **Toolkit** to install PyAEDT or update product menus to show any added Python-based toolkits.
- Click **Tools** > **Runscript...** to run scripts that you have recorded. A Pause command is available. You can also **Record Script to File...** or **Record Script to Project...**. See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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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



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

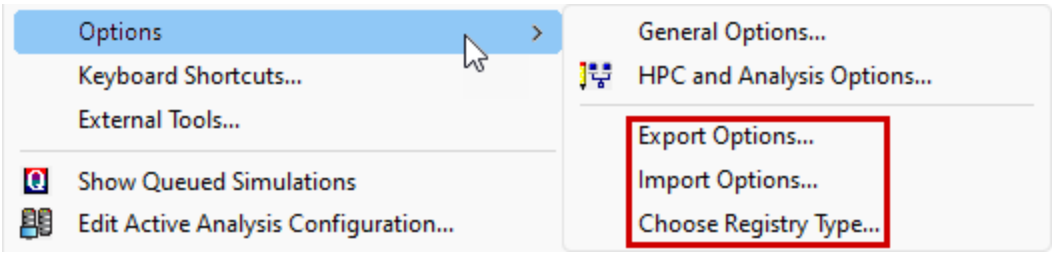
- Click **Tools** > **Keyboard Shortcuts...** to manage your keyboard shortcut behavior.
- Click **Tools** > **External Tools...** to manage menu access to external tools.
- Click **Tools** > **Show Queued Simulations** to view a dialog listing simulations in a queue and providing tools to manage the 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.

Understanding Registry Tools

In the context of this topic, when we say *"Registry"* we are referring to the files where the various Ansys Electronics Desktop application options are stored. We are not referring to the Windows registry. Specifically, this topic discusses the **Export Options**, **Import Options**, and **Choose**

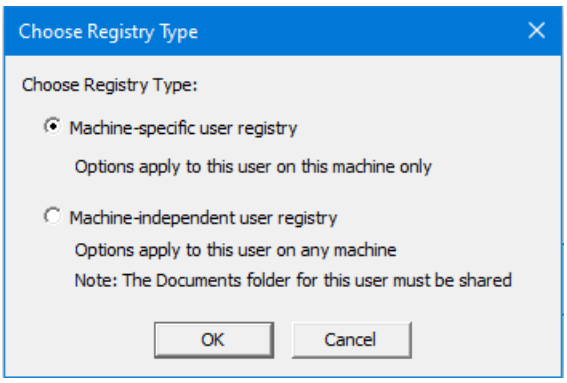
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Type commands that are listed in the fly-out menu that appears when you click **Tools** > **Options** from the menu bar.



Choose Registry Type:

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



The choices are summarized as follows:

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

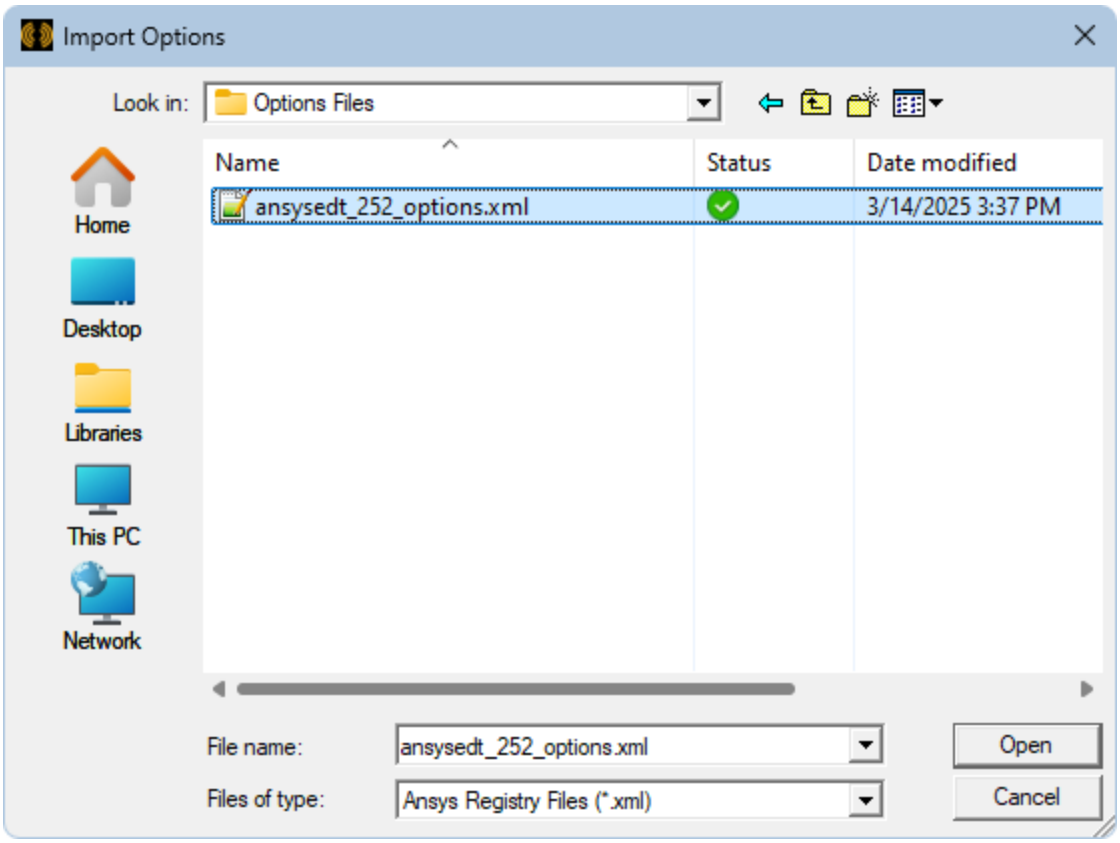
> **Note:**
>
> For a machine-independent user registry to work, the *Documents* folder must be shared by all machines on which you run the software, and you must log into the same account on each machine. The user registry file is stored in the path, ...Documents\Ansoft\ElectronicsDesktop20*yy.m*, where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

Exporting and Importing Options:

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

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

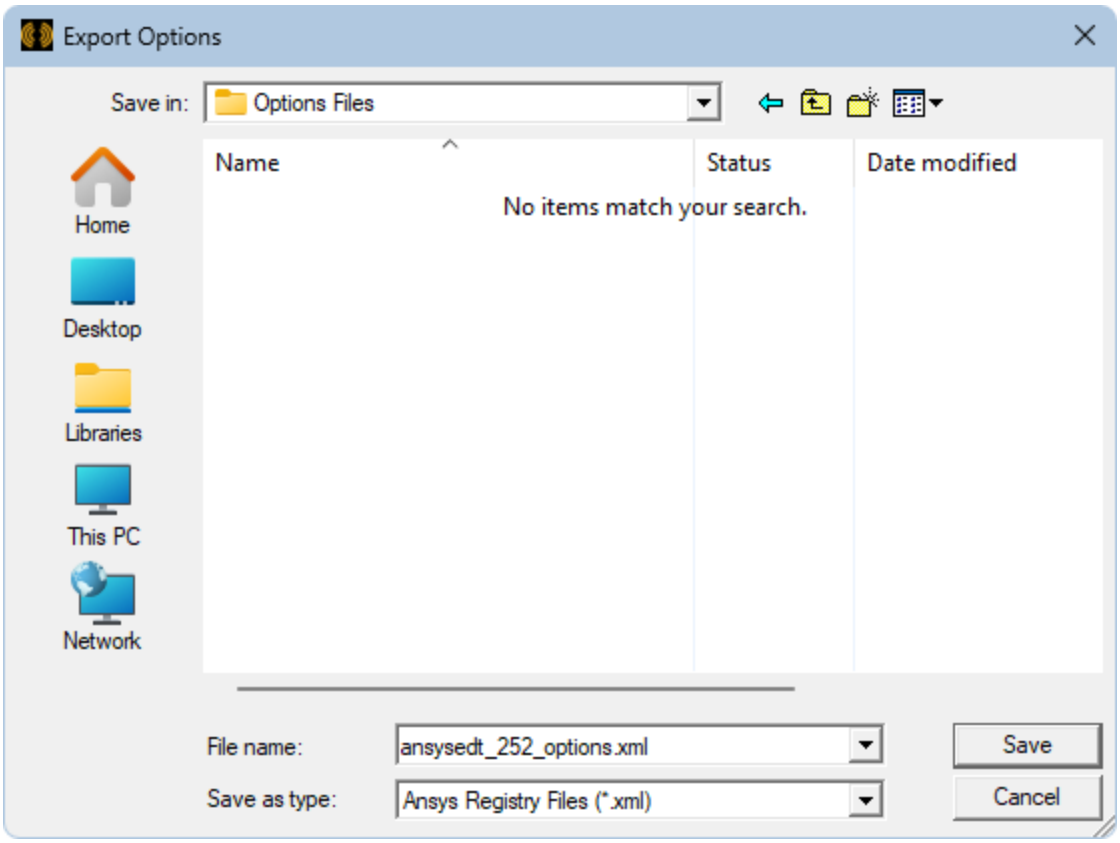
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

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

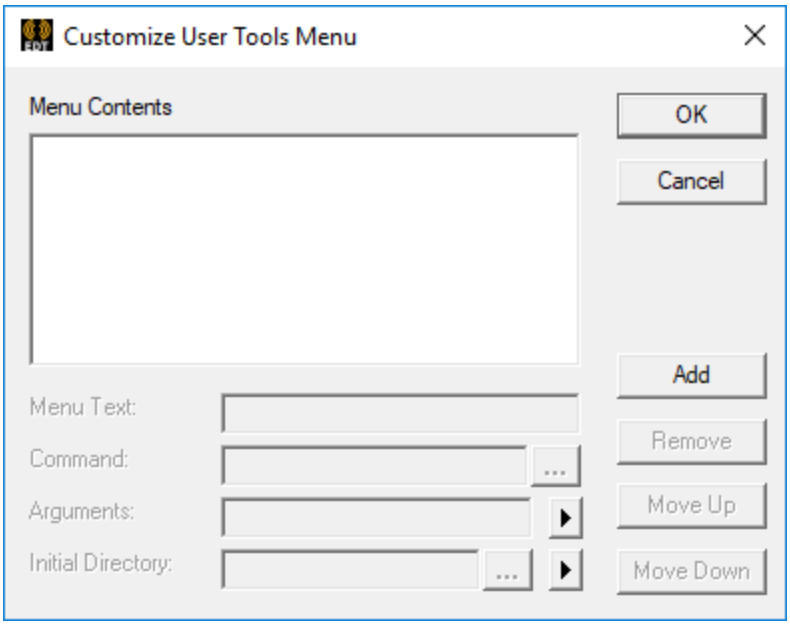
Adding External Tools to the Tools Menu

To add an executable to the Tools menu:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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.

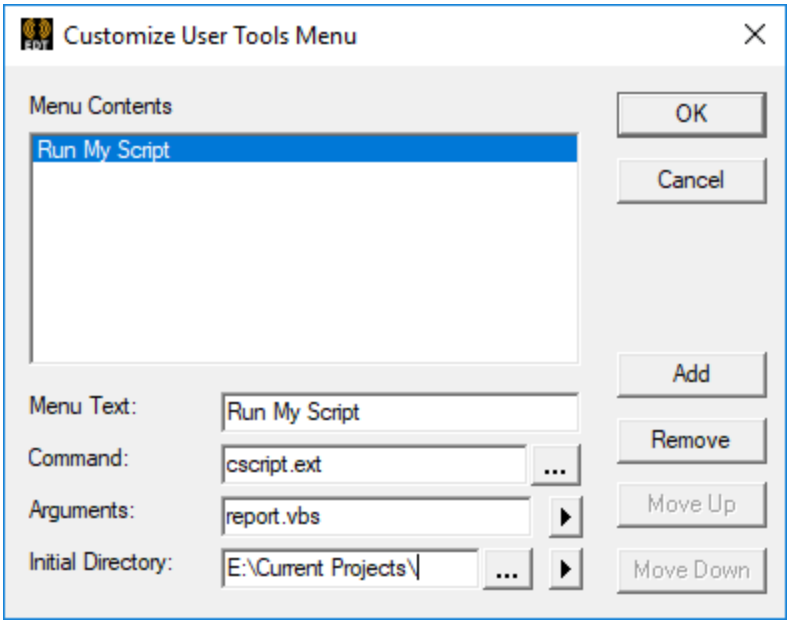
   Arguments field – accepts command arguments from the **>** button menu selections for File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

   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 Icepak 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.

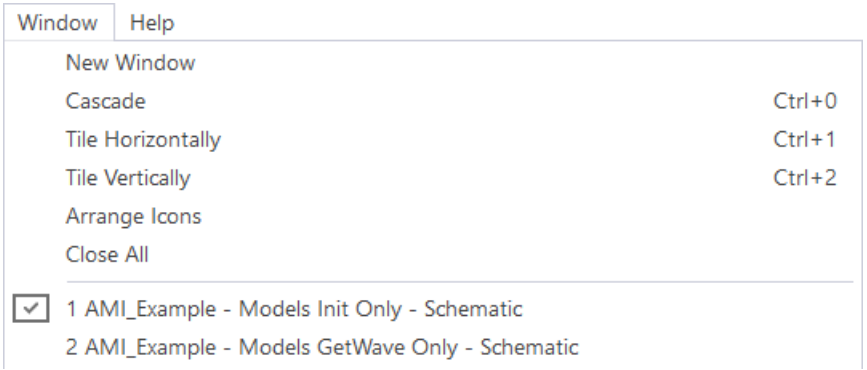
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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).

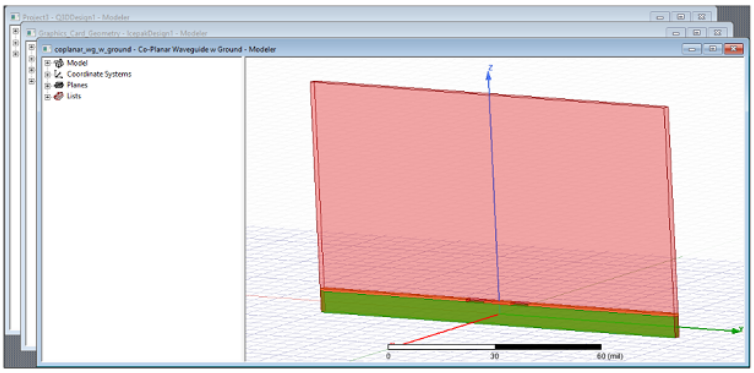
- Click **Tile Horizontally** (or type **h**) to arrange the open design windows in a top-to-bottom sequence:
- Click **Tile Vertically** (or type **v**) to arrange the open design windows in a side-to-side sequence:

> **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.

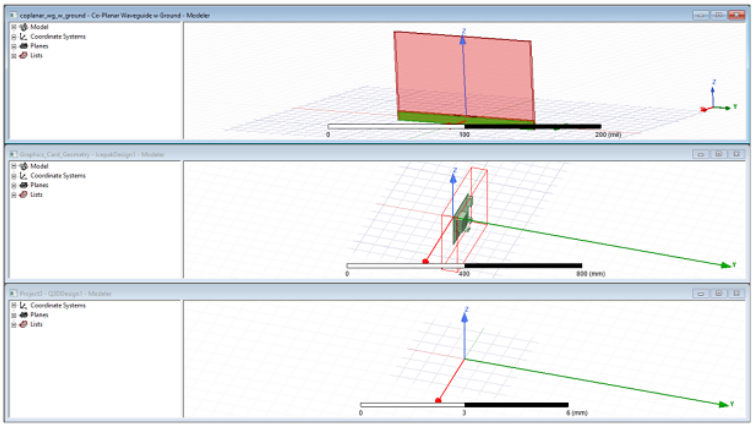
- Click **Arrange Icons** (or type **a**) to restore iconized windows to a neat row at the bottom of the Design Area, after the icons have been repositioned manually.
- Click **Close All** (or type **l**) to close all the editor windows in the Design Area.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

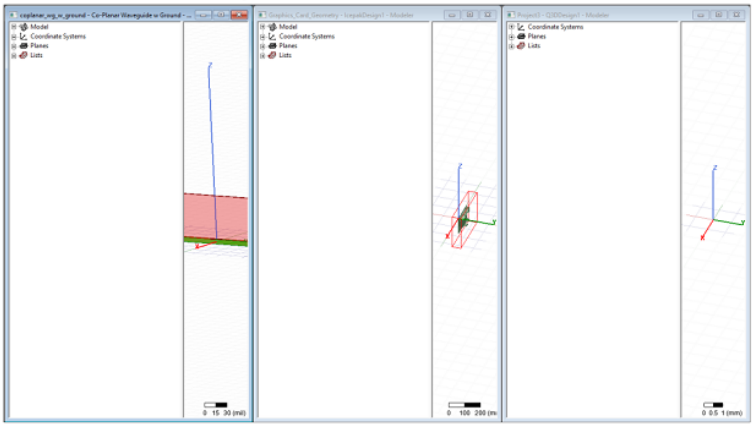
Cascade



Tile Horizontally



Tile Vertically



Ansys Electromagnetics Suite 2026 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 Icepak are:

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

- **Icepak Help** – opens the Icepak help within the Electronics help system. You can also access PDF versions from within the help system.
- **Icepak Scripting Help** – opens the Icepak scripting help.
- **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 2026 R1.
- **Ansys Product Improvement Program** – opens a window describing the Product Improvement Program option.
- **License Settings** – opens help for the License Settings tool.
- **Ansys Engineering Copilot** – opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant.
    - **Home** – includes a search function that will search the courses, knowledge base, and learning forum for information relevant to the search term you have entered (searches the entire Ansys Innovation Space (AIS)). The Search AIS field also provides filters to narrow your search in a particular information category; however, this feature does not automatically filter based on physics type or the product you are using.
    - **Ansys Innovation Courses** – the Search function can be used to access a wide range of Ansys Electronics Engineering courses that include on-demand, self-paced video training and quizzes.
    - **Ansys Knowledge** – provides access to Ansys expert knowledge articles. The search filter is preset based on the application you are using, but you can change the filter as desired.
    - **Ansys Learning Forum** – provides access to discussions and presentations from Ansys experts, partners and customers. Users can also post questions to the forum, which is moderated by Ansys employees.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **AnsysGPT** – is a multilingual Ansys Artificial Intelligence (AI) Virtual Assistant. The AnsysGPT™ assistant takes your query and produces answers based on internal and external expert knowledge of the product. It draws upon information from Ansys Knowledge articles, Ansys Innovation Courses, Ansys Learning Hub courses, the Ansys Learning Forum, Ansys documentation, Ansys.com, and the official Ansys YouTube channels, as well as other Ansys information sources.

  To use the Copilot tool:

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

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

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

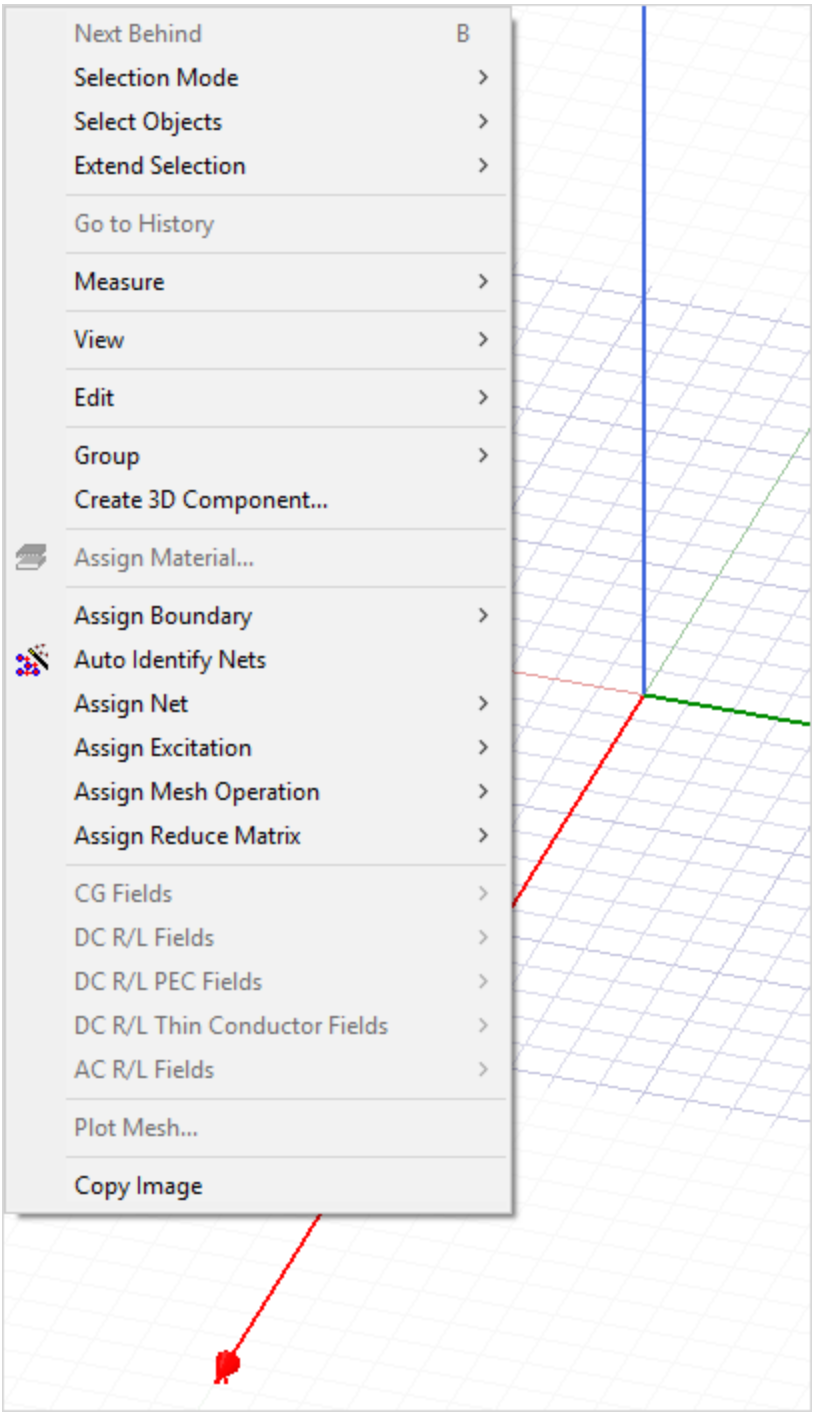
> **Note:**
>
> All commands available on shortcut menus are also available from the menu bar.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

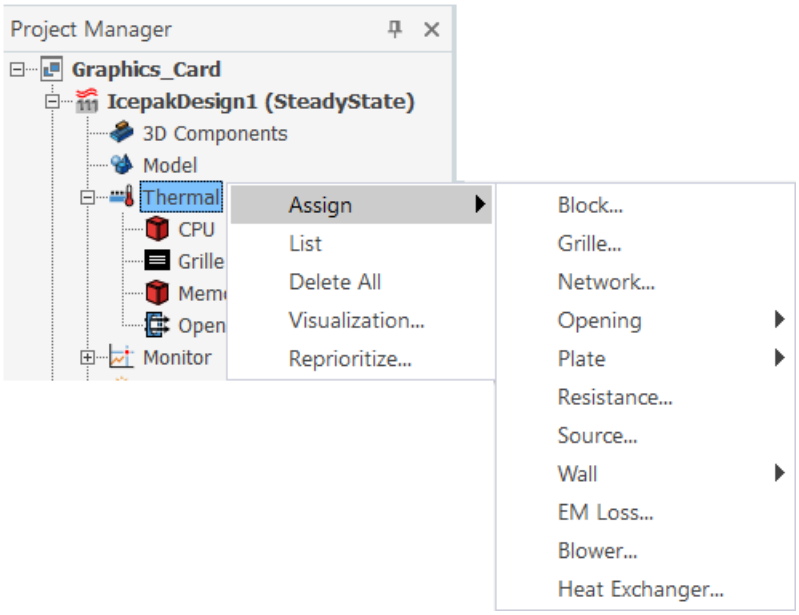


Shortcut Menus in the Project Manager Window

Each node, or item, in the project tree has a shortcut menu. For example, from the shortcut menu for the **Thermal** icon, you can assign thermal boundary conditions to selected objects;

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

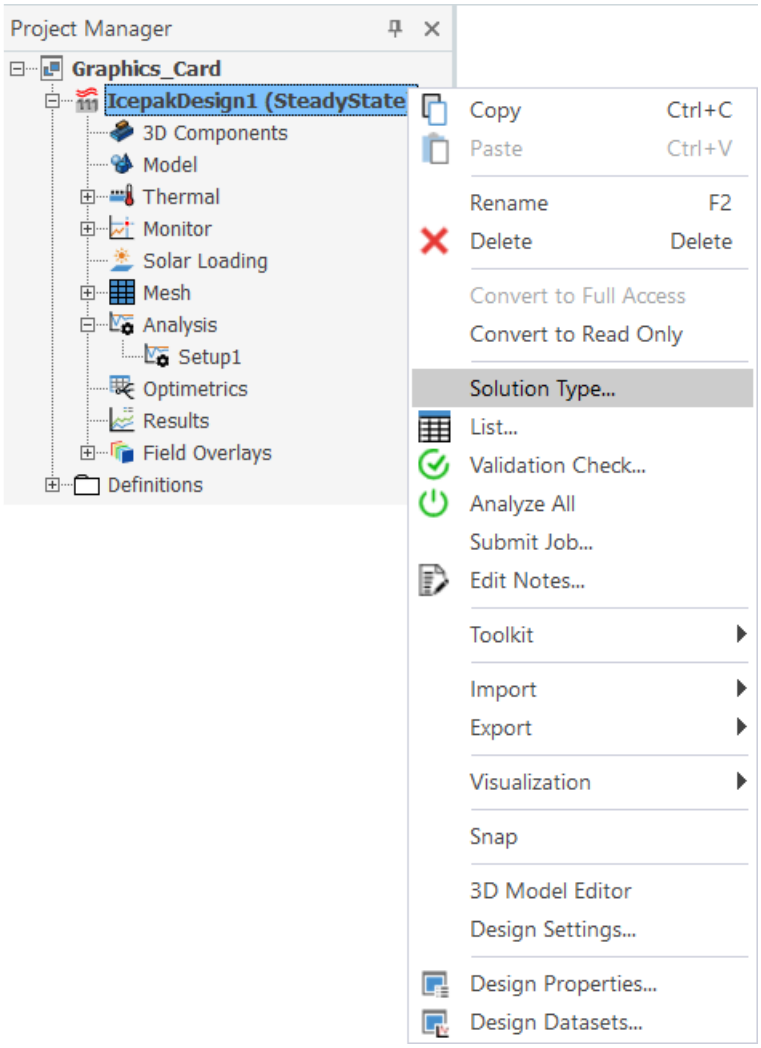
review information for all the boundary assignments for the active design; delete all boundary condition assignments; show or hide a boundary's geometry, name, or vectors; and change the priority of a previously assigned boundary.



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 2026 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 Menu Bar. You can modify the 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 shortcut 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 the Ansys Electronics Desktop application are:

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

     > **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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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)

  > **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>

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

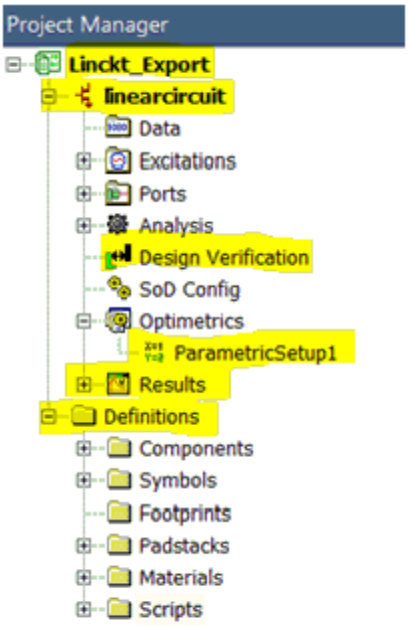
- For “DesignerMenu” and “Context”, only no action setting or “useDefault” is valid.
- The “DesignerMenu” action setting is significant because it is how you specify whether you want to construct your own menus or modify existing default menus. If this is set to “useDefault”, default menu settings are 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 2026 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 2026 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**

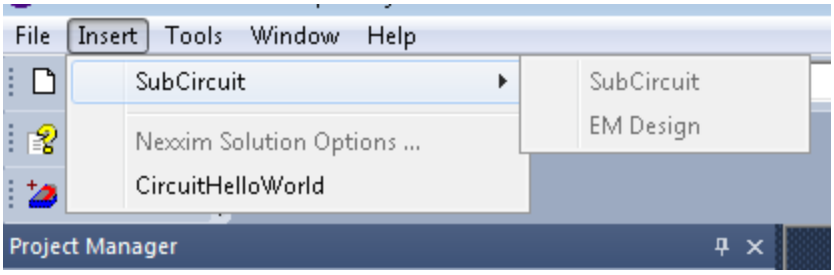
```
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">
   <Context name="Design">
      <TopMenu>
         <MenuName>Insert</MenuName>
         <pop-upMenu>
            <MenuName>S&amp;ubCircuit</MenuName>
            <LeafMenu>
               <MenuName>&amp;SubCircuit</MenuName>
               <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>

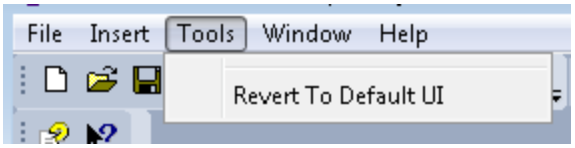
            <MenuID>38460</MenuID>
            <ShowBitMap>No</ShowBitMap>
            <Accelerator></Accelerator>
         </LeafMenu>
         <CustomMenu>
            <MenuName>CircuitHelloWorld</MenuName>
            <ScriptPath>$USERLIB/HelloWorld.vbs</ScriptPath>
            <ShowBitMap>No</ShowBitMap>
  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


Ansys Electromagnetics Suite 2026 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





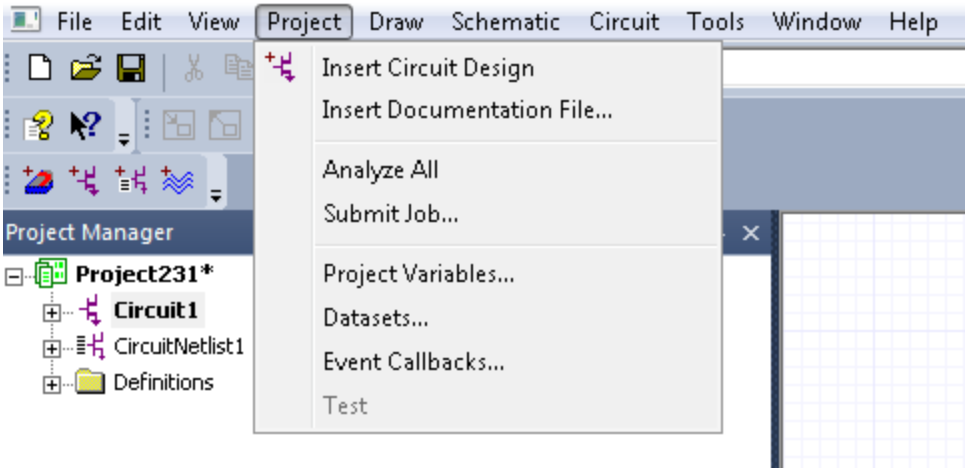
2. **Circuit Design.xml Sample**

```
<?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;Project</MenuName>
           <LeafMenu action="delete">
                        <MenuName>Insert HFSS Design</MenuName>
           </LeafMenu>
           <LeafMenu action="delete">
                        <MenuName>Insert HFSS 3D &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>
```

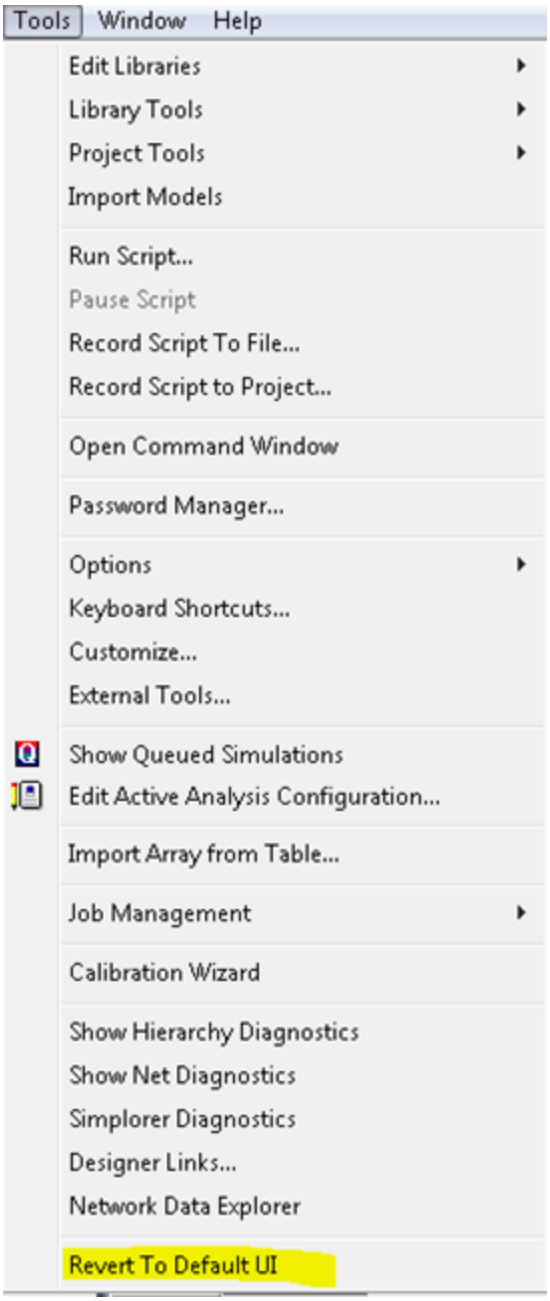

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            <LeafMenu action="delete">
                        <MenuName>Insert 2D Extractor
        Design</MenuName>
            </LeafMenu>
            <LeafMenu action="delete">
                        <MenuName>Insert Circuit
        &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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



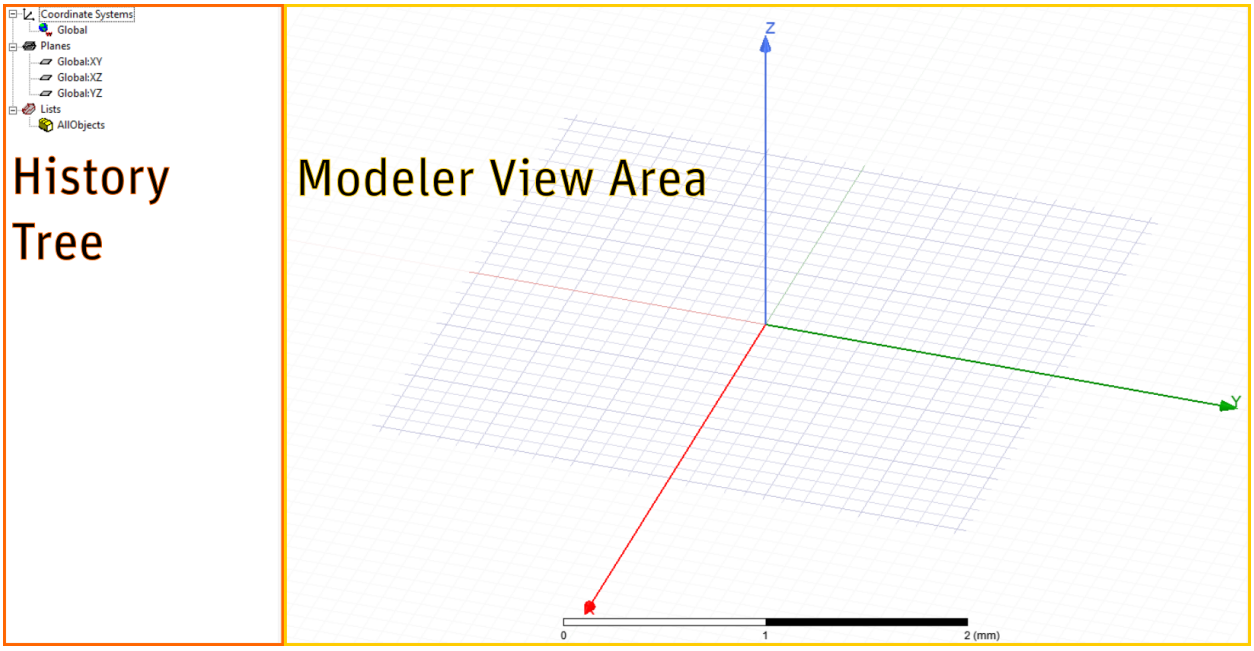
Ansys Electronics Desktop Design Area

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 2026 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.

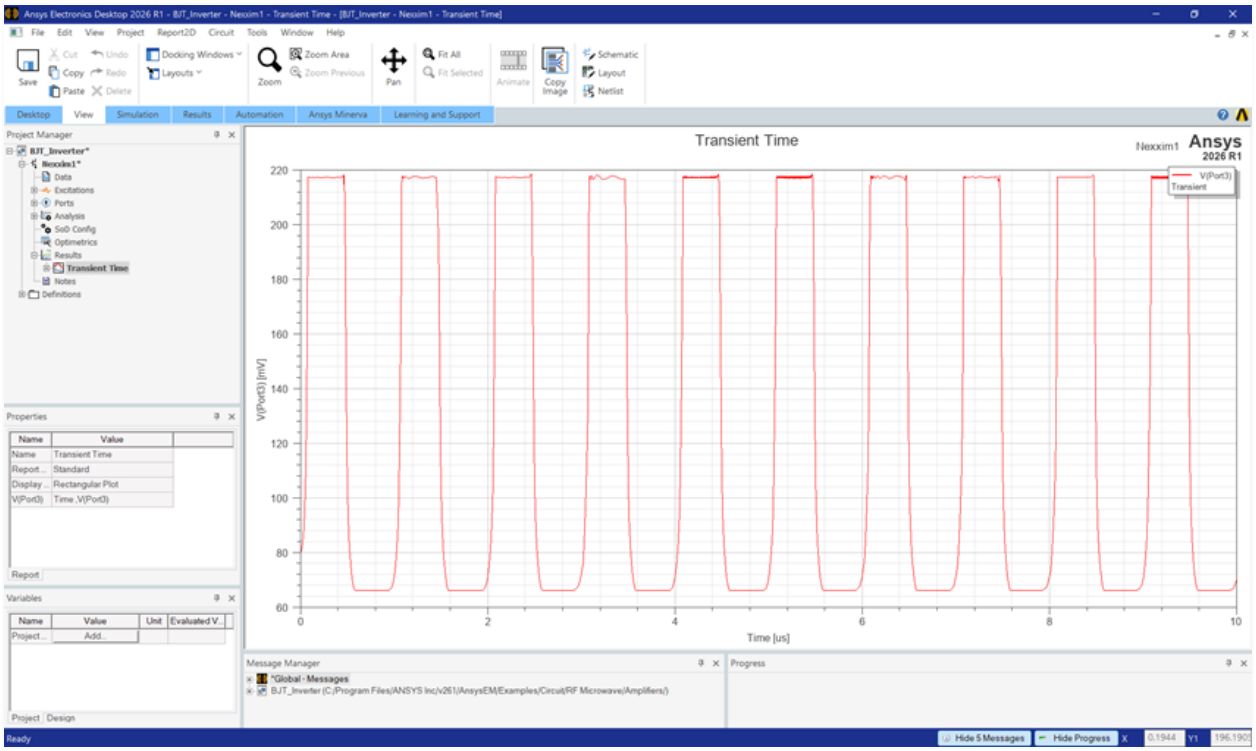
You can modify the view of objects in the **3D Modeler** window without changing their actual dimensions or positions. Besides menu and icon commands, you can also use **Alt+click**, mouse buttons, and keyboard shortcuts. See: Modifying the Model View.

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 2026 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:

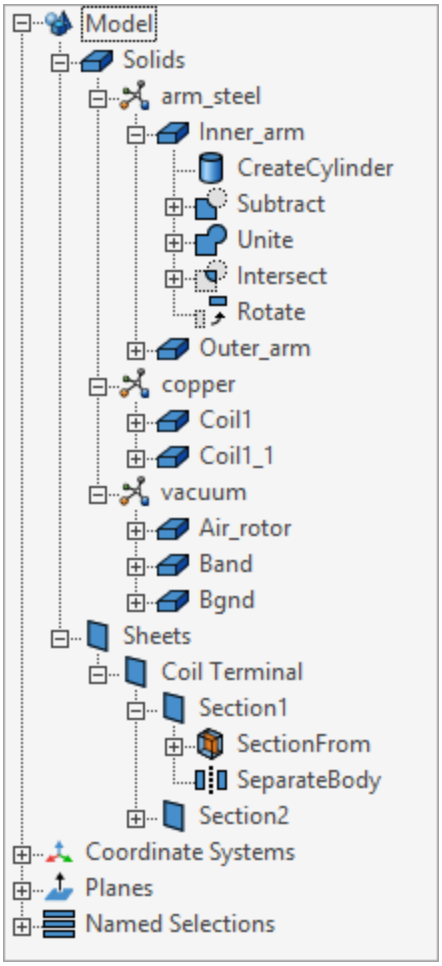


For more information, see Postprocessing and Generating Reports.

Ansys Electronics Desktop History Tree

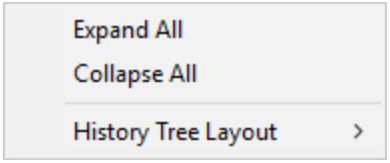
The history tree in the **3D Modeler** window lists the active model's structure and grid details.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

- 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.
Named Selections	Lists the object or face selections for the active model. By default, a selection called *AllObjects* appears. You can add a specific group of objects or surfaces for field plot or for calculation. See: Creating an Object Selection and Creating a Face Selection.

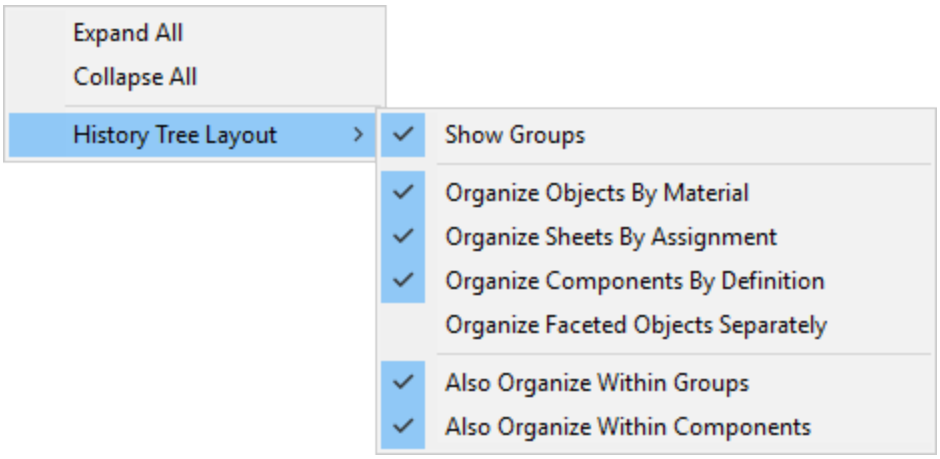
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

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

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 (IcepakFEA - 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 IcepakFEA – 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **HFSS and Maxwell:** Priority is specified by the user.
    - **Icepak, IcepakFEA, 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**

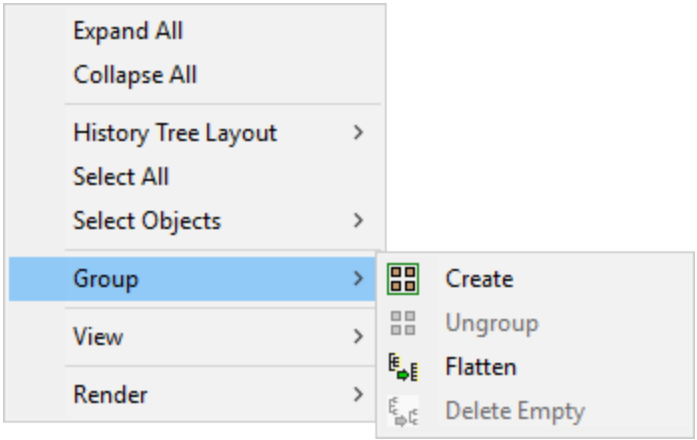
Controlling the View of Objects in the History Tree

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

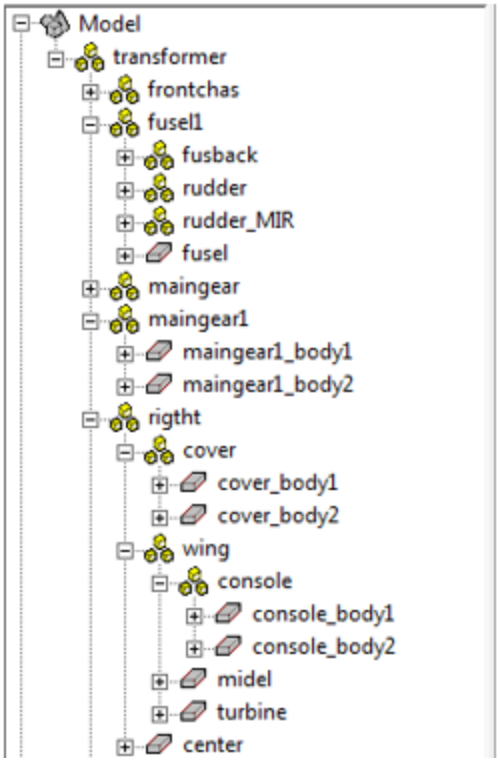
The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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-groups 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.



- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

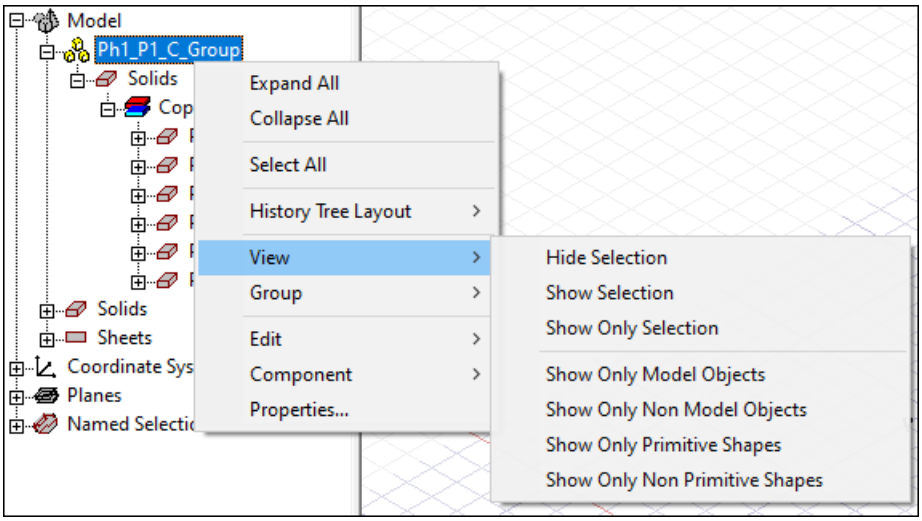
Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operations on Groups

View Options

For groups, you can hide or show the entire group. You can also opt to show only particular types of objects or shapes in that group. The right-click view options are shown below:



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select a 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.

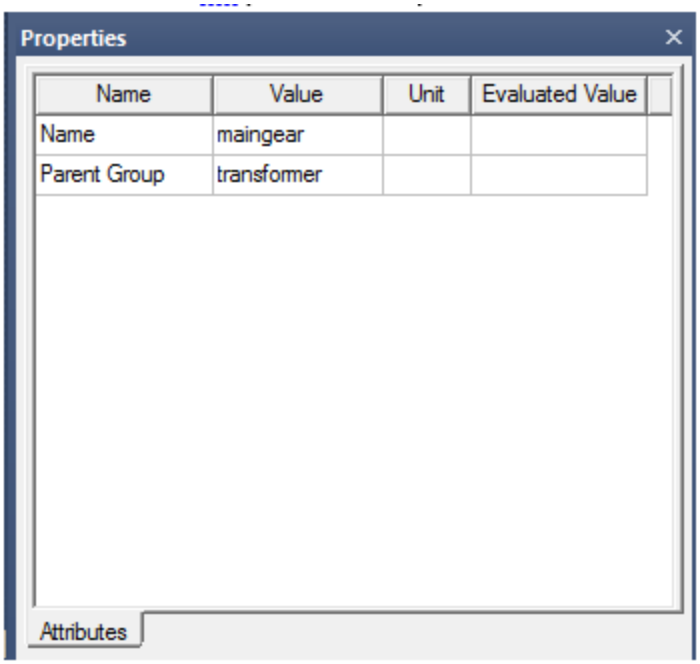
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate are available for groups. 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.

If the group has hierarchy, you can also flatten it.

Group Properties

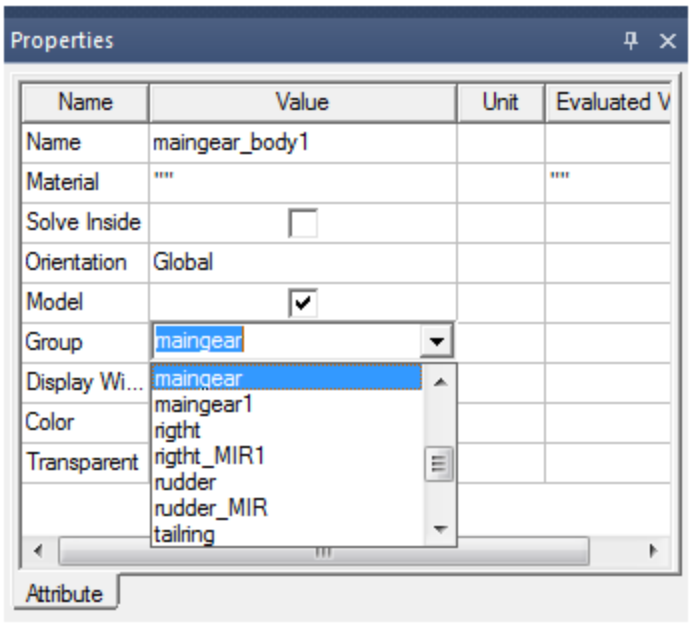
A Group's Property window is shown when you select a group in the history tree. The Group's properties include Name and parent Group.



Group Property Window

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

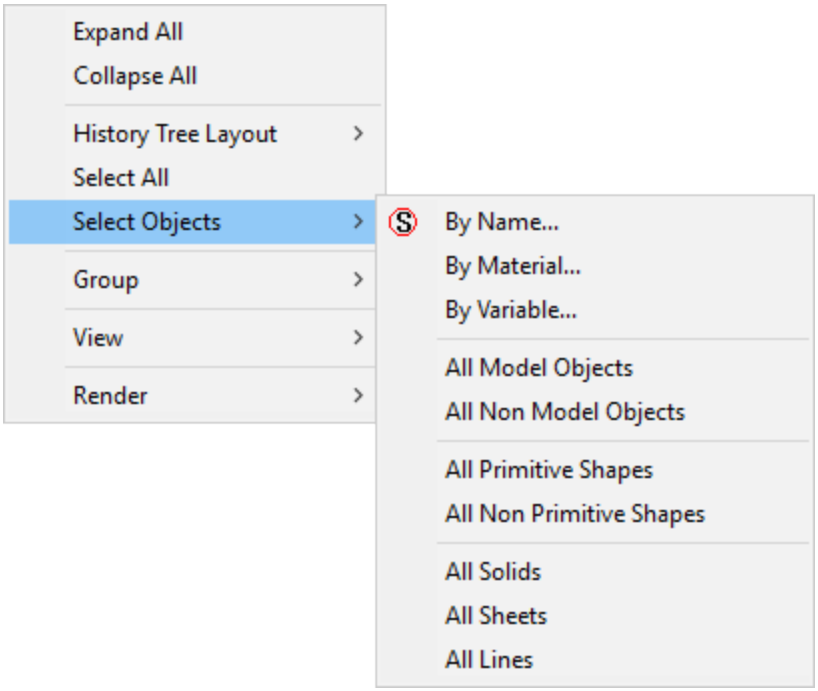
Selecting Objects in the History Tree

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:

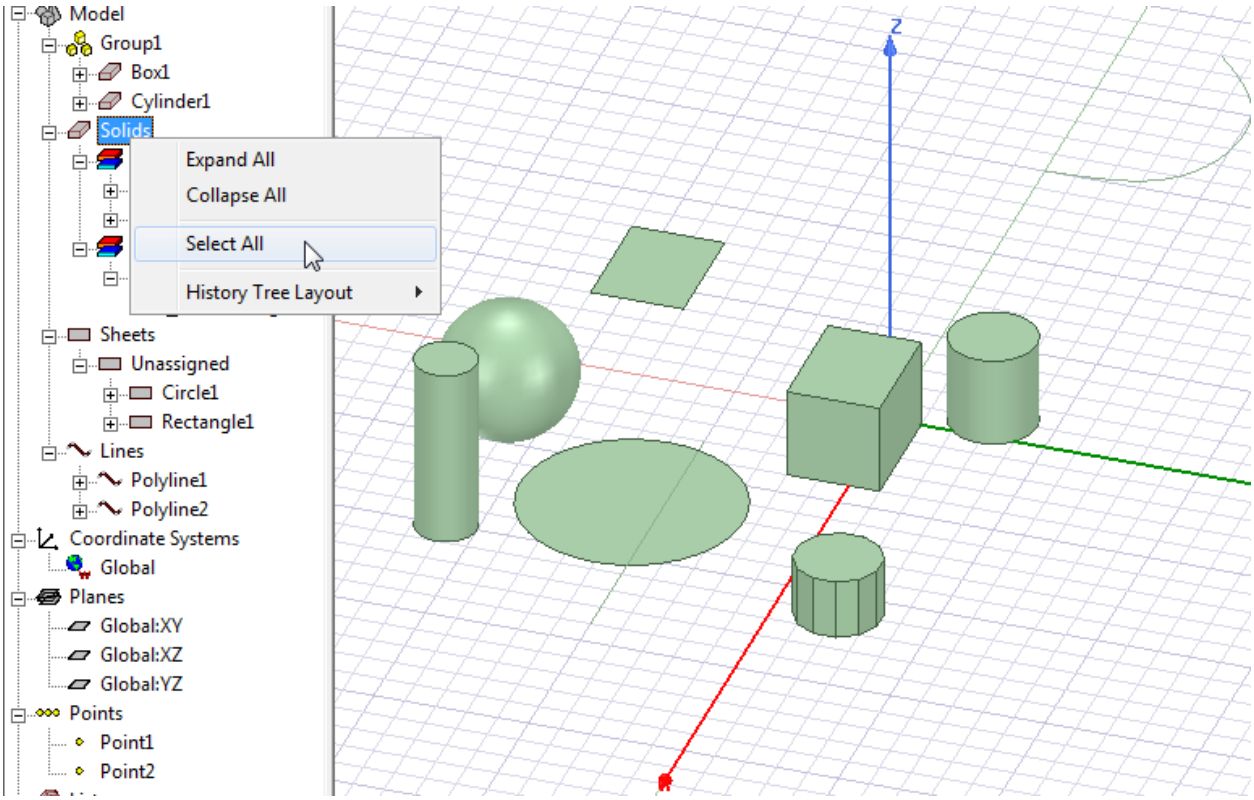
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**:

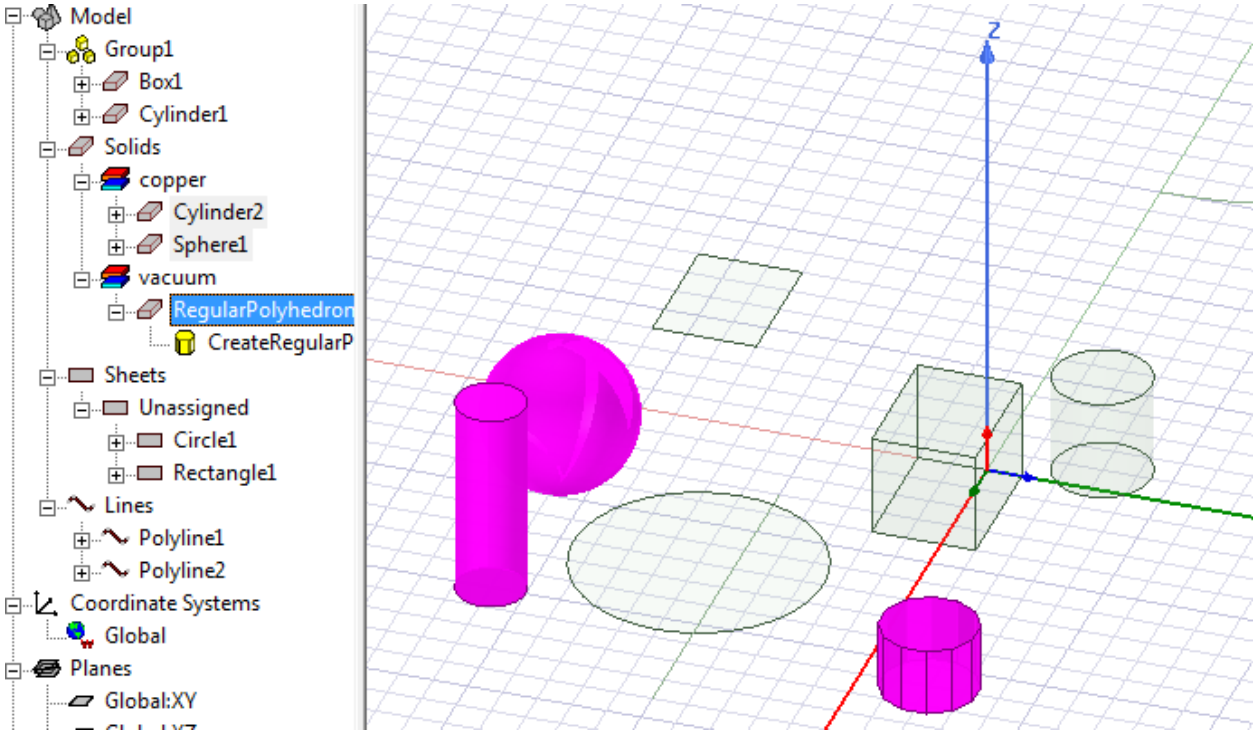
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



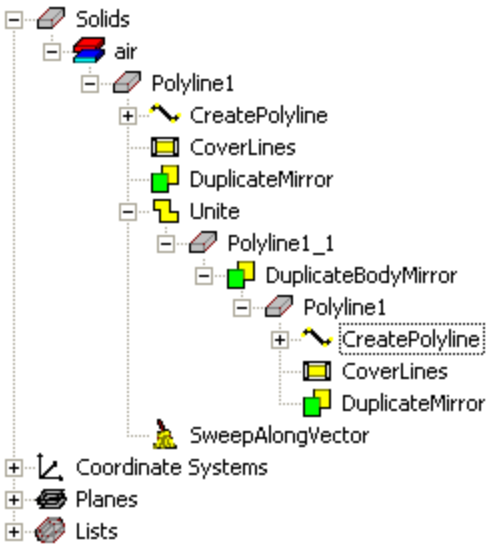
The right-click for Model and **Select All** does select the members of Group1 because they are part of the model.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

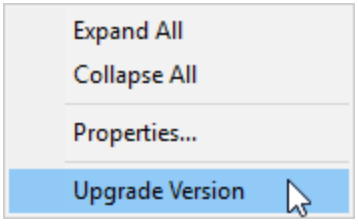


Selecting a command in the history highlights the object in the Modeler window and shows that object's properties (if available) in the docked **Properties** window. You can look at the fields in the **Properties** window to see any editable fields for that command, such as coordinate system, line type,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



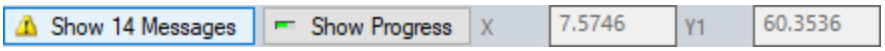
By default, the Modeler opens legacy projects using the modeling kernel version under which the project was saved, so that users don't see side effects from slight topology changes between versions. The **Upgrade Version** operation forces Electronics Desktop to use the latest modeling

Ansys Electromagnetics Suite 2026 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.

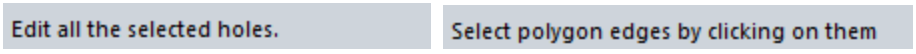


When more than one progress bar is active, the top progress bar is represented on the status bar with a progress indicator:



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

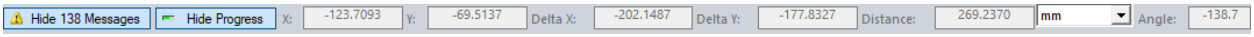
Directions for inputs appear on the left side of the status bar:



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 2026 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 Icepak 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + 2	Tile windows vertically
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Navigation:	
Ctrl + D	**Fit** view
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 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

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

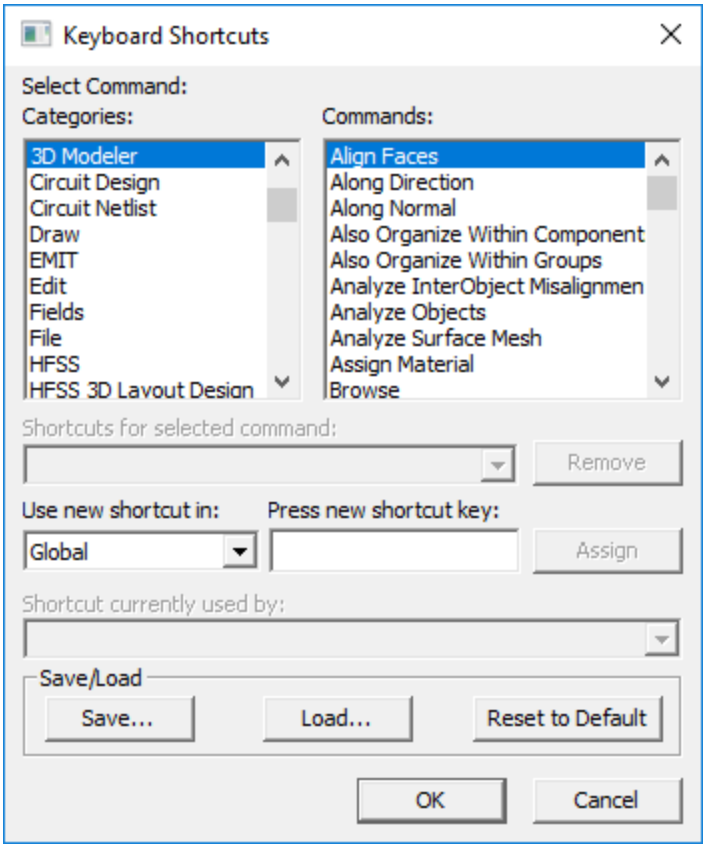
Function	Key / Mouse Button Assignment	
	Current	**Legacy**
Pan (all contexts)	Ctrl + MMB + drag or Ctrl + *arrow keys*	Shift + LMB + drag or Ctrl + *arrow keys*
Rotate (3D contexts)	MMB + drag or Alt + *arrow keys*	MMB + drag or Alt + LMB + drag or Alt + *arrow keys*
Zoom (all contexts)	Wheel or Shift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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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:

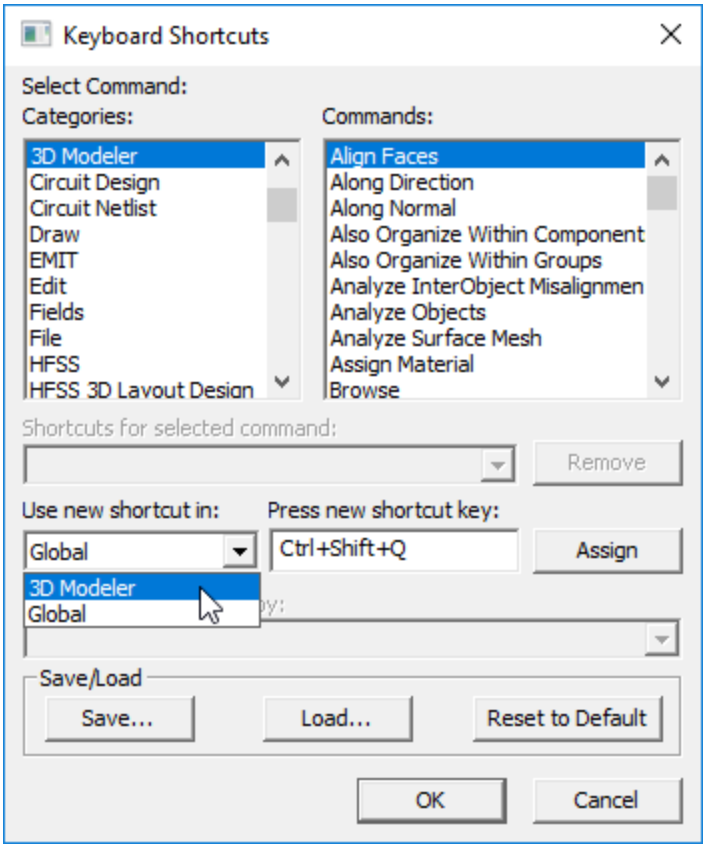
1. Select the **Category** and **Command**.
2. If you want to disable a current assignment for the selected command, click **Remove**.
3. To assign a keyboard shortcut, place the cursor in the **Press new shortcut key** field.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

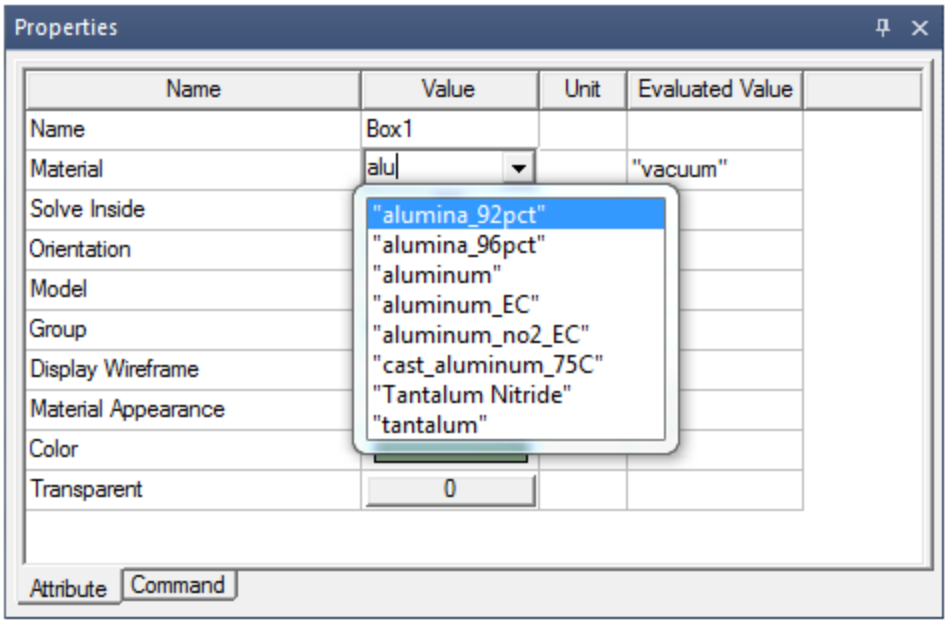
6. Click **OK** to save the current settings, or **Reset to Default**.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time by selecting a predetermined match rather than typing out the entire name. The following figure shows an example of auto-complete for material names:



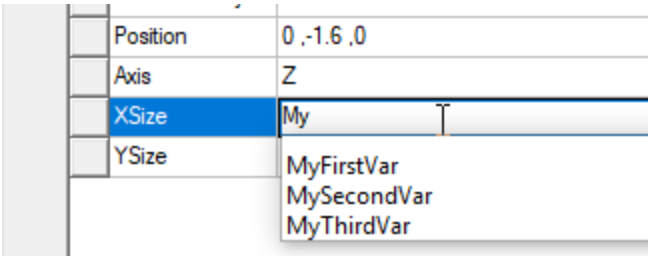
Using Auto-Complete

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.

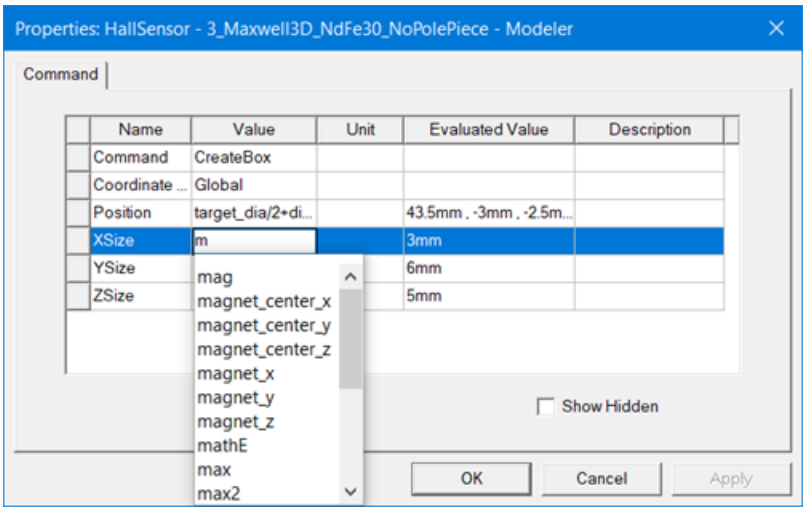
Hitting **Tab** or **Enter** accepts the current selection for auto-complete, which replaces the text typed with the full auto-complete match and hides the list. Otherwise, hitting the escape key **Esc** hides the auto-complete list. Typing more letters causes auto-complete matching to resume.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

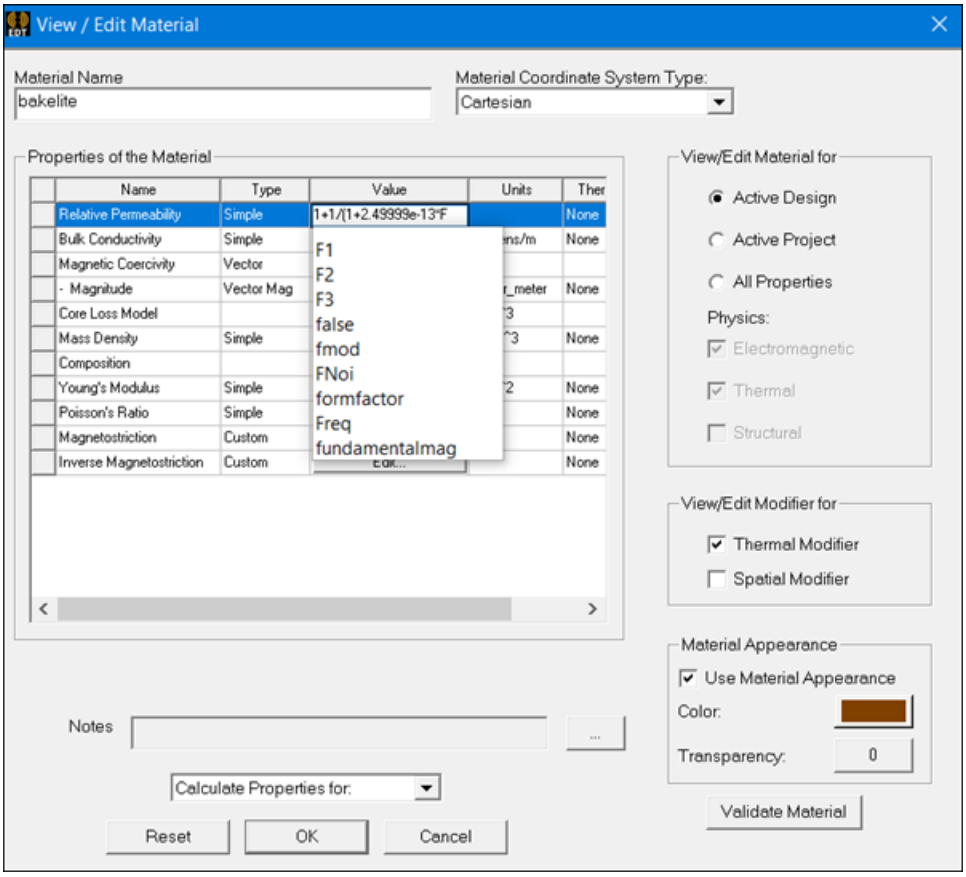


Properties of other items may also have auto-complete configured to work with value expressions (for example, properties of a CreateBox command or of a circuit component). The following shows a design property with auto-complete matching:



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

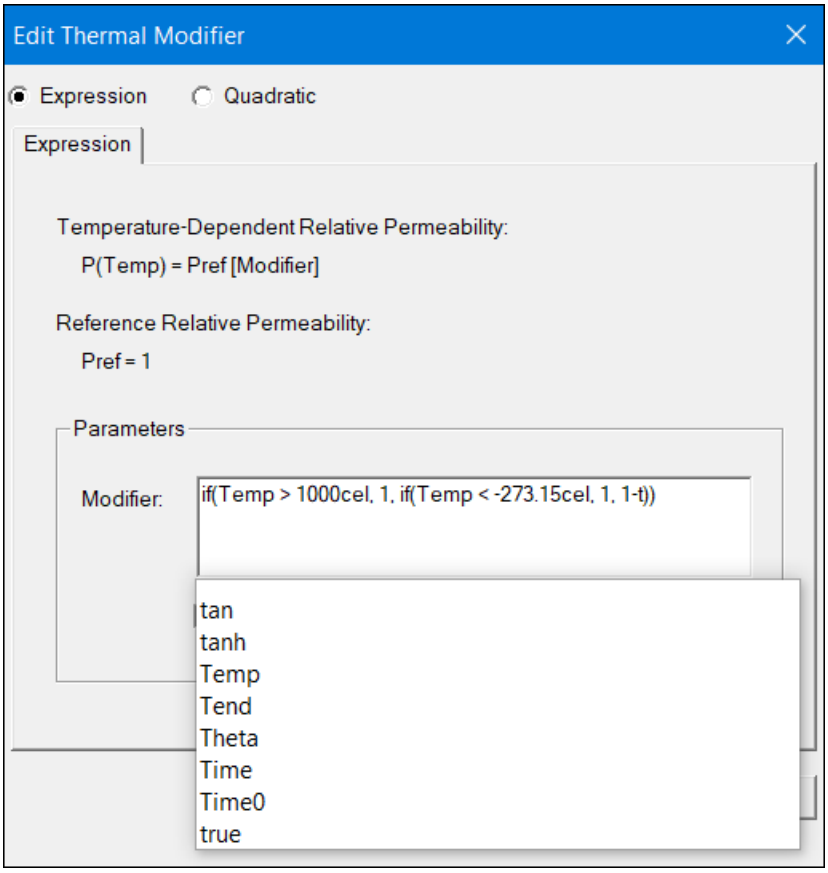
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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 2026 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.

> **Note:**
>
> Command-line functionality for Large Scale DSO is handled through the desktopjob interface; see Large Scale DSO Command Line Interface for more details.

Command-Line Syntax

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


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 that has spaces.

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 are 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 for `-BatchSave`:

  `norecurse`: only process files in the top level of the 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>`**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

The `-generatearchive` option can be used with the `-BatchSolve` command to create an archive that includes the project file, batchinfo directory, any external files, and the results directory after the batchsolve completes. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc.*, will be tried. If you want to run a batchsolve on an existing archive, use the -archiveoptions repackageresults command.

If you wish to specify which setups `-BatchSolve` completes, you can use additional parameters:

- `[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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`<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"
```

> **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",
  ```


Ansys Electromagnetics Suite 2026 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 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:

  ```
  -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:

  ```
  "excludelypes=Frequencies,Mesh Assembly"
  ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `maxlevels= <1 | 2>` – the maximum number of levels of job distribution (the current maximum is 2). See Selecting Optimal Configurations for Distributed Analysis.
- `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:

- `-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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `ExportProfile`
- `ExportConvergence`
- `ExportMeshStats`
- `ExportNetworkData`
- `ExportNMFData`
- `ExportEigenmodes`
- `ExportTransientData`
- `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.
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
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.
```

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\v261\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\v261\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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
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.

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, these `-archiveoptions` commands are available:

- `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 subdirectory 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The `-AutoExtract` option is only valid when used with `-BatchSolve`.
- The `-AutoExtract` option is automatically added for all Ansys Cloud Burst 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.
- `-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. For the ansysedtng beta option, see Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta.
- `-WaitForLicense` – directs Electronics Desktop to wait for unavailable licenses.
- `-Scriptargs` – used in conjunction with `-RunScript` or with `-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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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.

> **Note:**
>
> `-Batchoptions` is only valid for batch jobs. It is ignored if you have not specified `-BatchSolve`, `-BatchSave`, or `-BatchExtract`.

-Batchoptions Examples

The `-Batchoptions CreateStartingMesh` is available for the following products:

HFSS, HFSS-3DLayout, Icepak, Q3D, Q2D, Maxwell3D, Maxwell2D, and IcepakFEA.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

Ansys Electromagnetics Suite 2026 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 IcepakFEA 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 2026 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 (\\ ) because 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 2026 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 2026 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 **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 parameters 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 2026 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

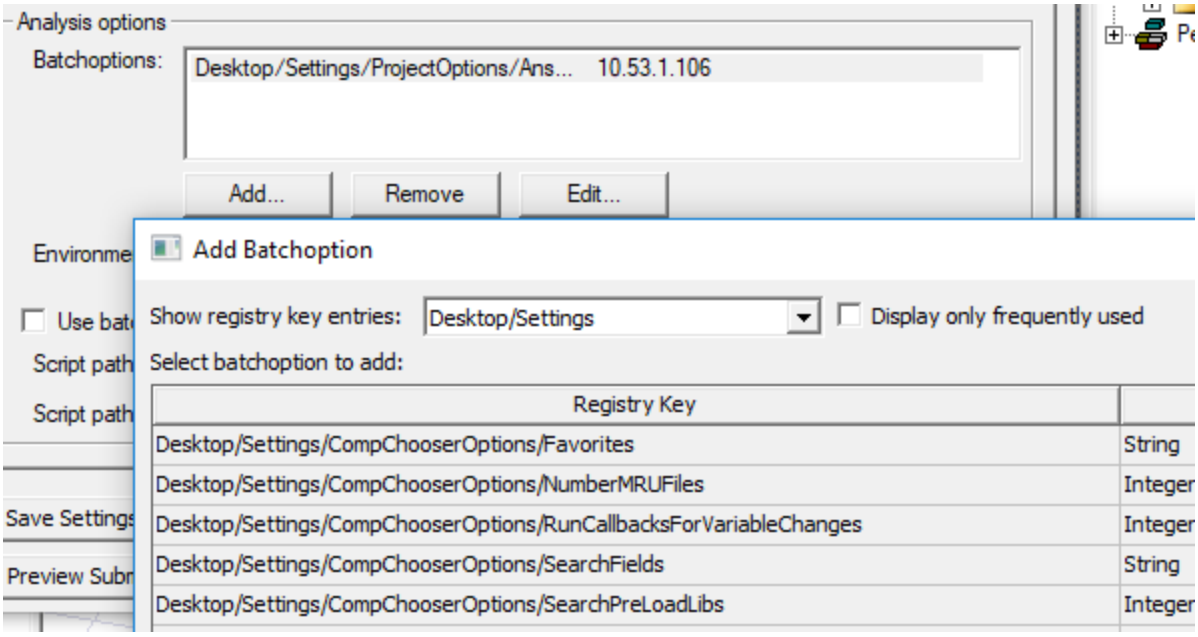
For -batchoptions Use: Various Desktop Settings

Note that most of these options only affect the software interface. 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 2026 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 2026 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 2026 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.

Batchoptions: Override Registry Entry

Any registry entry can be overridden via a command-line option.

Example: batchoptions specified on a command line

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
ansysedt -batchsolve -batchoptions
"'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4 'HFSS 3D
Layout Design/NumCoresPerDistributedTask'=2" projectname.aedt
```

Example: batchoptions specified in a file

```
ansysedt -batchsolve -batchoptions numprocfile projectname.aedt
```

where numprocfile contains

```
$begin 'Config'
  'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4
  'HFSS 3D Layout/NumCoresPerDistributedTask'=2
$end 'Config'
```

Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta

> **Note:**
>
> Running Electronics Desktop using the ansysedtng command is currently a beta feature. For information on enabling beta options, see General Options: Desktop Configuration.

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.

> **Note:**
>
> Linux users whose installations lack a MainWin home directory receive an error when running the ansysedt command. To run Electronics Desktop from a command line successfully on such Linux machines, use the ansysedtng command and options described in this section.

Command-line Syntax

```
ansysedtng <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 2026 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. BatchSolve 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 **-Help** option. The commands are further described below:

- **-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
- 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:**
  >
  > - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.
- **-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 **-Logfile** 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:

  - **[*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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **[*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"
    ```

    > **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

  ```
  -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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
- **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:

- **-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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

- **-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.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedtng -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedtng -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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Burst Computing 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.
- **-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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-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.
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

> **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.

> **Note:**
>
> **-batchoptions** is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, Icepak, Q3D, Q2D, Maxwell 3D, Maxwell 2D, and IcepakFEA.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

Ansys Electromagnetics Suite 2026 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 IcepakFEA 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:

```
ansysedtng -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 2026 R1 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta

> **Note:**
>
> Running Electronics Desktop using the ansysedtng command is currently a beta feature. For information on enabling beta options, see General Options: Desktop Configuration.

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.

> **Note:**
>
> Linux users whose installations lack a MainWin home directory receive an error when running the ansysedt command. To run Electronics Desktop from a command line successfully on such Linux machines, use the ansysedtng command and options described in this section.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command-line Syntax

```
ansysedtng <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.

Run Commands

The following command line run commands are available for Ansys Electronics Desktop. BatchSolve 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 **-Help** option. The commands are further described below:

- **-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
- 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:**
  >
  > - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.
- **-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 **-Logfile** and **-WaitForLicense** options. If parallel solve is possible, you can use the **-Distribute** option.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you wish to specify which setups **-BatchSolve** completes, you can use additional parameters:

- **[*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 2026 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 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:

  ```
  -MachineList num=4
  ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
- **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:

- **-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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

- **-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.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedtng -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedtng -LogFile "H:\Logs\mylog.log" -BatchSolve
myDesign:Nominal "H:\Projects\MyProject.aedt"
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying Project Files

Specifying a project file opens that project when Electronics Desktop launches. If **-BatchSolve** is set, the project will also be solved.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The -autoextract option is only valid when used with -BatchSolve
- The -autoextract option is automatically added for all Ansys Cloud Burst Computing 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.
- **-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.
- **-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.
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **Note:**
>
> **-batchoptions** is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, Icepak, Q3D, Q2D, Maxwell 3D, Maxwell 2D, and IcepakFEA.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

> **Note:**
>
> No adaptive meshing occurs for any Icepak or IcepakFEA 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:

```
ansysedtng -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 2026 R1 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Running Ansys Electronics Desktop with a JSON File

Ansys Electronics Desktop includes line arguments that can be included in a JSON file and executed by entering the -json command line option. The -json option takes a JSON file as argument, and runs AEDT with the options specified within the file. All options in the -help text is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

supported. Options in the JSON file can be used together with options specified in the command line. If the same option is specified both in the command line and the JSON file, the option in the command line overrides the option in the JSON file.

> **Note:**
>
> The -json option functions with both the ansysedt and ansysedtng commands.

Enter the command as follows to execute the JSON file

```
ansysedt -json <filepath to the JSON file>
```

JSON File Format

Take the following into consideration when creating a JSON file.

- The JSON file must contain an object. Each option is a key value pair where the key is the option name (no dash) and the value is the argument.
- For options that do not take any arguments, such as -monitor, the value must be a bool - true or false to turn on or off the option, respectively.
- For options that take optional arguments, such as -logfile, the value can either be a bool or some other type (in this case, a string specifying the log file) depending on the option.
- Project/archive file and folder should be specified directly on the command line when using -json.
- See the example below for a JSON file containing all options in the -help text.

> **Note:**
>
> The example does not contain a valid combination of options but is solely used to show what type of argument each option takes.

```
{
    "archiveoptions":
    {
        "overwritefiles": true,
        "path": "\\\\dataserver\\home\\user\\OptimTee.aedt",
        "repackageresults": true,
        "winpath": "\\\\dataserver\\home\\user\\OptimTee.aedt"
    },
    "auto":
    {
        "numDistributedVariations": 2
    },
    "autoextract": "reports, fieldplots",
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    "batchextract": "\\\\dataserver\\home\\user\\batchextract_
script.py",
    "batchoptionhelp": false,
    "batchoptions":
    {
        "HFSS/CreateStartingMesh": 1,
        "Desktop/ProjectDirectory": "\\\\dataserver\\home\\user"
    },
    "batchsave":
    {
        "autoheal": false,
        "cleanupmodel": true,
        "forcearchive": true,
        "norecurse": true,
        "outputfolder": "\\\\dataserver\\home\\user\\batchsave_
folder",
        "preservehistory": false,
        "previewonly": false
    },
    "batchsolve":
    {
        "designName": "TeeModel",
        "setupType": "Nominal",
        "setupName": "OptimTee_Analysis"
    },
    "distributed":
    {
        "excludeTypes":
        [
            "Frequencies",
            "Mesh Assembly"
        ],
        "includeTypes": [],
        "maxLevels": 2,
        "numLevel1": 2
    },
    "help": false,
    "iconic": false,
    "local": false,
    "logfile": "\\\\dataserver\\home\\user\\logfile.log",
    "machinelist":
    {
        "list":
        [
            {
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                "machine": "sjohpcw2k16fe",
                "numTasks": -1,
                "numCores": 1,
                "RAMPercent": 90,
                "numGPUs": 1
            },
            {
                "machine": "dcuww10core01",
                "RAMPercent": 80
            }
        ]
    },
    "monitor": true,
    "ng": true,
    "runscript": "\\\\dataserver\\home\\user\\script.py",
    "runscriptandexit": "\\\\dataserver\\home\\user\\script.py",
    "scriptargs": "Design1 Setup1",
    "showmonitorjob": false,
    "showselectscheduler": false,
    "showsubmitjob": false,
    "waitforlicense": false
}
```

Running Ansys Electronics Desktop from a Windows Remote Terminal

When running Icepak 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

- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Platform Support webpage and click the links to the following topics:

- *Ansys 2026 R1 - Job Schedulers and Queuing Systems Support*
- *Ansys 2026 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.

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*.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Installing PyAEDT (Beta)

PyAEDT is a Python library that interacts with the Electronics Desktop API to make scripting simpler for the end user.

It supports:

- All 3D products (HFSS, Icepak, Maxwell 3D, and Q3D Extractor)
- 2D tools
- IcepakFEA
- EMIT
- Circuit Nexxim
- Twin Builder
- Layout tools (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 Python code while reusing information as much as possible across the various APIs.

Documentation for PyAEDT, including installation instructions, extensions, and example scripts 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:

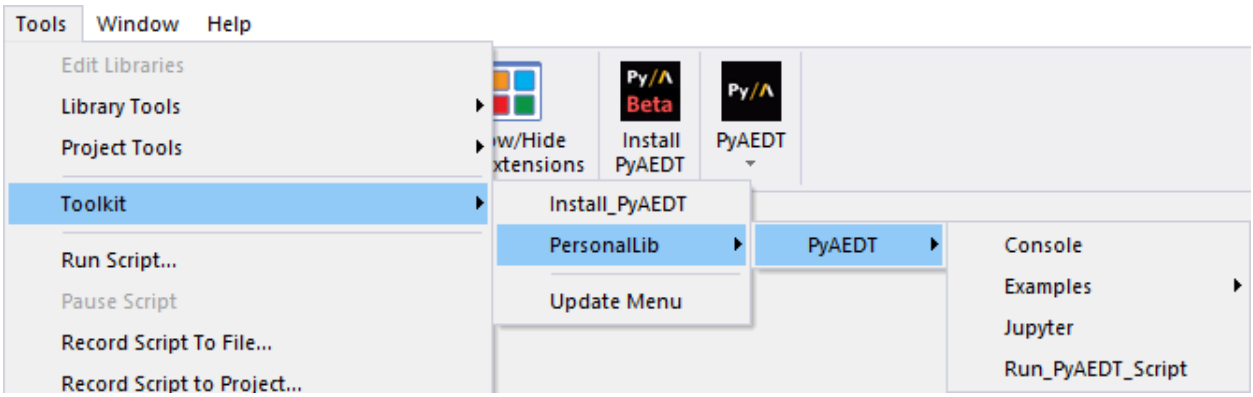
- From the **Automation** tab, click **Install PyAEDT**.

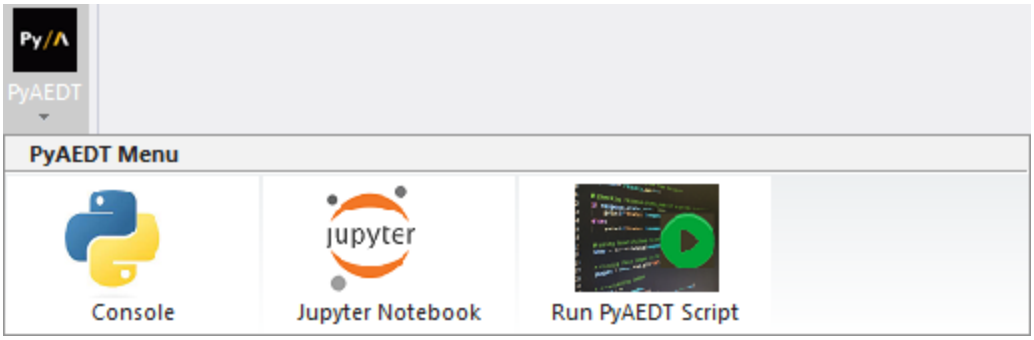


A web browser launches, and takes you to detailed installation instructions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When installation is complete, the **Tools** > **Toolkit** > **PersonalLib** menu and the **Automation** tab update to display PyAEDT menu options:





Debug Logging

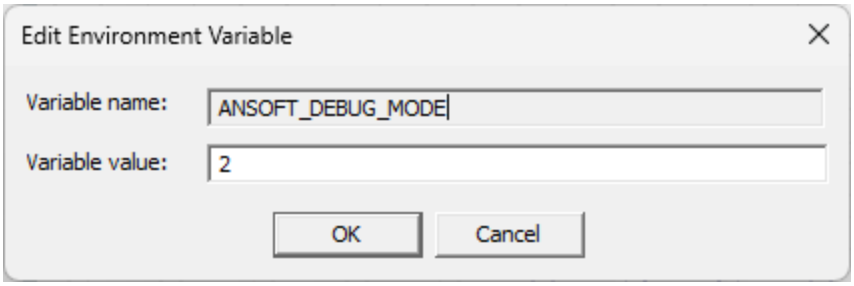
Electronics Desktop provides error logging capabilities.

To begin logging errors:

1. Click **Tools** > **Diagnostics** > **Debug Logging**.

   The **Edit Environment Variable** window appears, showing the ANSOFT_DEBUG_ MODE variable.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

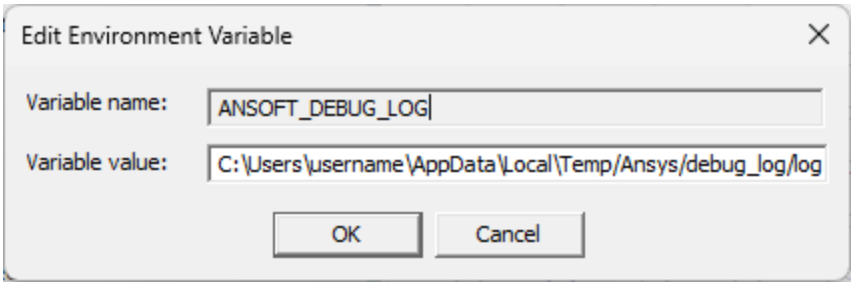


2. Enter a **Variable value**.

   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** > **Diagnostics** > **Stop Debug Logging**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 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.

Project Files

Managing Projects

Undo and Redo

Validating Projects

Exporting Models and Data

Importing Models and Data

Documentation and Printing

Setting Options

Variables

Ansys Workbench Integration

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.

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.
- The Ansys Electronics Desktop 2025R1 version cannot open projects created with 2022R2 or earlier that were using ACIS modeler.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

Getting Started with Icepak

Icepak™ is an interactive software package for calculating the thermal and fluid behavior of an electronic component. The software includes post-processing commands for analyzing this behavior in detail.

Using Icepak, you can compute:

- Thermal solution data, such as temperature, heat flux, and heat flow
- Fluid flow solution data, such as flow rate and speed

You are expected to draw or import the structure, specify material characteristics for each object, and identify ports and special surface characteristics.

Setting up an Icepak Design

To set up an Icepak design, follow this general procedure. After you insert a design, you do not need to perform the intermediate steps sequentially, but you must complete them before you generate a solution.

Insert an Icepak design into a project.

1. Select the solution type.
2. Draw the model geometry, create native components, and assign material characteristics to objects.
3. Assign thermal boundary conditions, which specify the field behavior at the edges of the problem region and object interfaces.
4. Define global mesh settings, assign mesh regions, and generate a mesh.
5. Specify how Icepak will compute the solution.
6. (Optional) Set up any Optimetrics you want to run.
7. Run the simulation.
8. View solution results, post-process results, and view reports.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

After a period of idleness of 10 minutes, Icepak gives up its license. A renewal of activity automatically requests a license. Such idle notifications do not occur during solves.

Inserting an Icepak Design

Click **Project**>**Insert Icepak Design** .

The new design is listed in the project tree. It is named IcepakDesign*n* by default, where *n* is the order in which the design was added to the project.

The **3D Modeler** window appears to the right of the Project Manager. You can now create the model geometry.

Note:

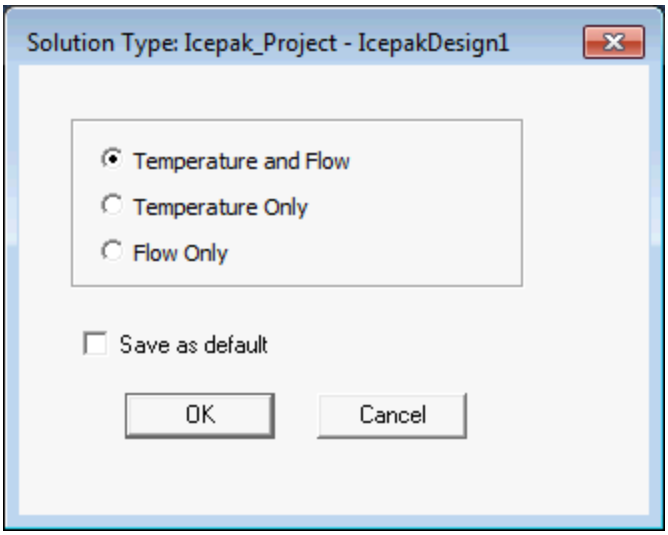
Click the plus sign to the left of the design icon in the project tree to expand the project tree and view specific data about the model, such as its boundary conditions.

Selecting the Solution Type for Icepak

You can specify the Icepak design's solution type to solve for temperature and/or flow.

1. Click **Icepak**>**Solution Type Dialog...**.

   The **Solution Type** dialog box appears.

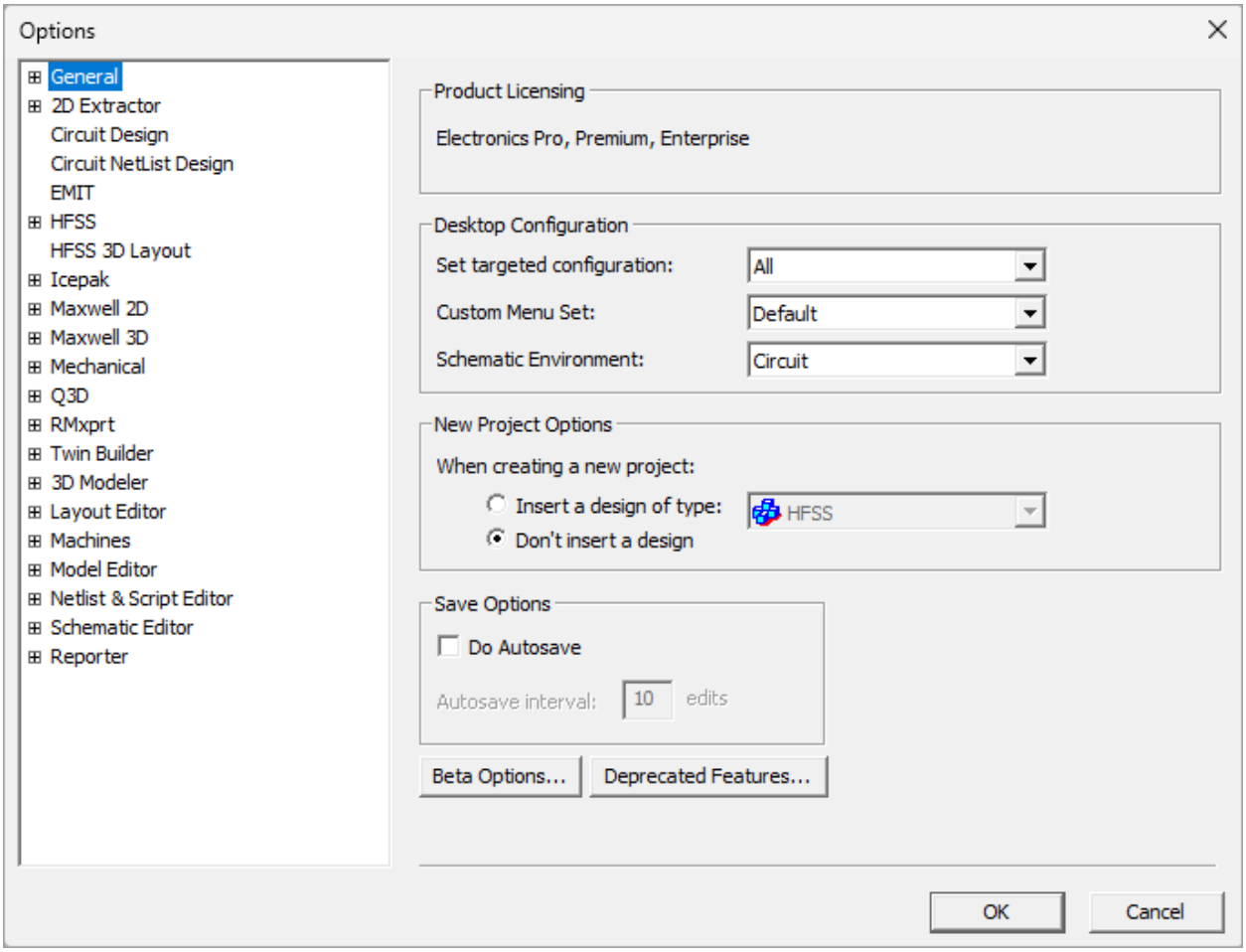


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select one of the following:
    - **Temperature and Flow**: For solving the energy equation for the temperature distribution together with the flow equations.
    - **Temperature Only**: For solving the energy equation for the temperature distribution only.
    - **Flow Only**: For solving the problem for flow only.

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 2026 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
- IcepakFEA
- Maxwell 2D
- Maxwell 3D
- Q3D
- 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.

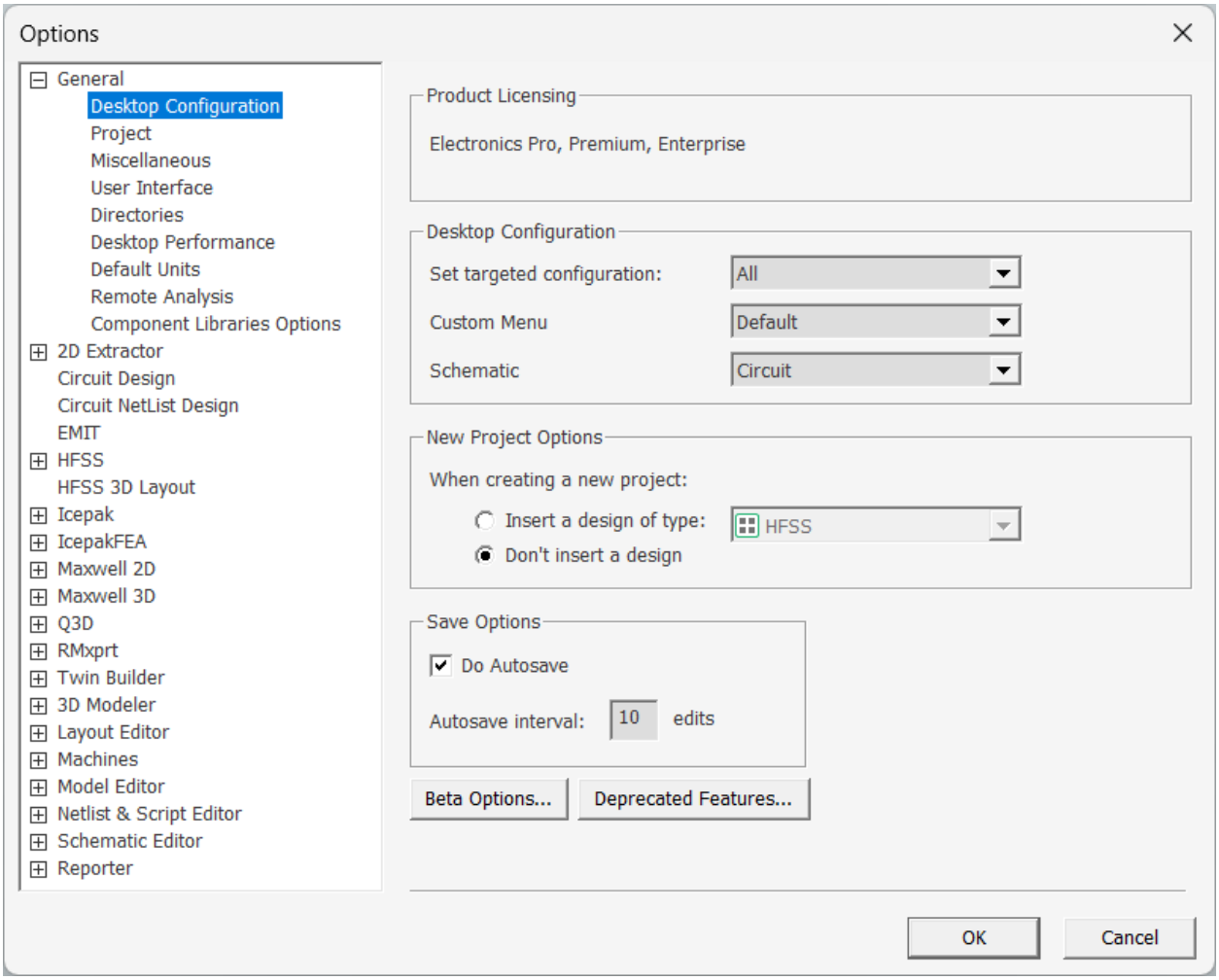
Setting General Options

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 2026 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.

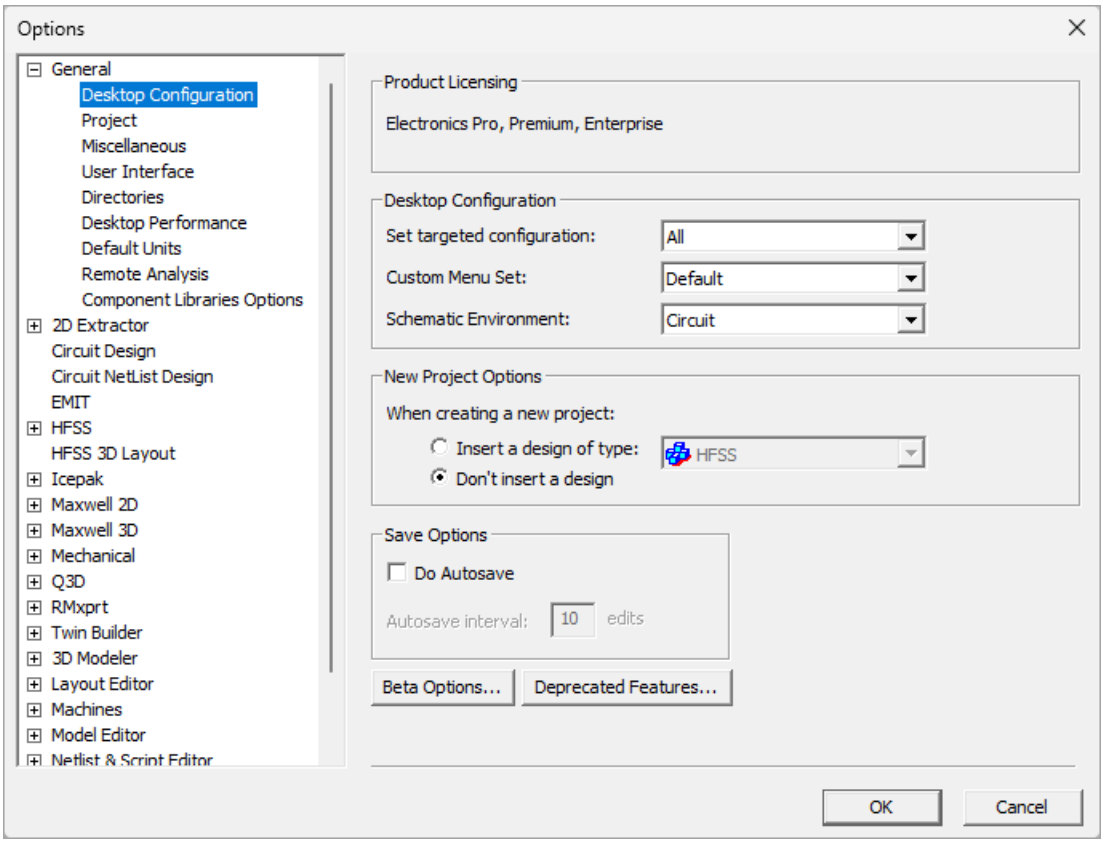
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

For more details, see Setting HPC and Analysis Options.

In the **Desktop Configuration** area, options include:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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:

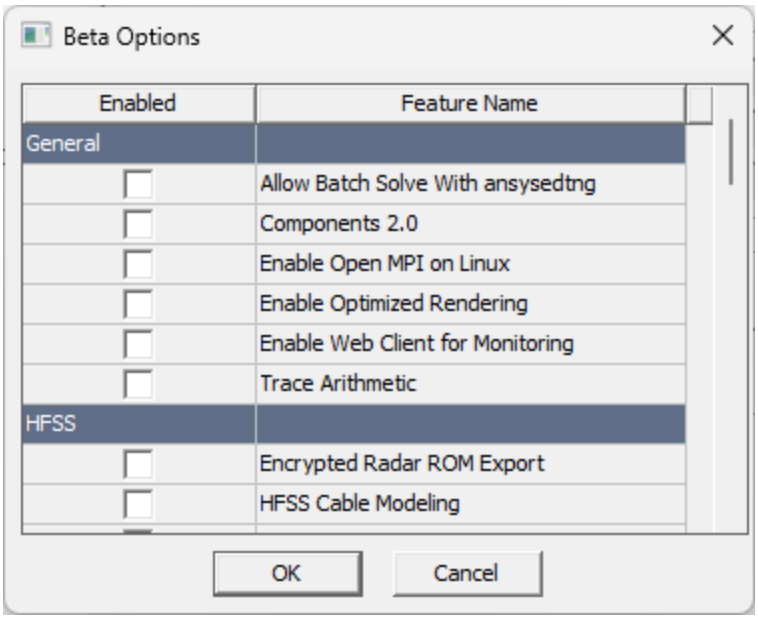
- **When Creating a New Project** – Select whether to insert a design when creating a new project. If you choose to insert a design by default, use the drop-down menu to select the default design type.

In the **Save Options** area, options include:

- **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



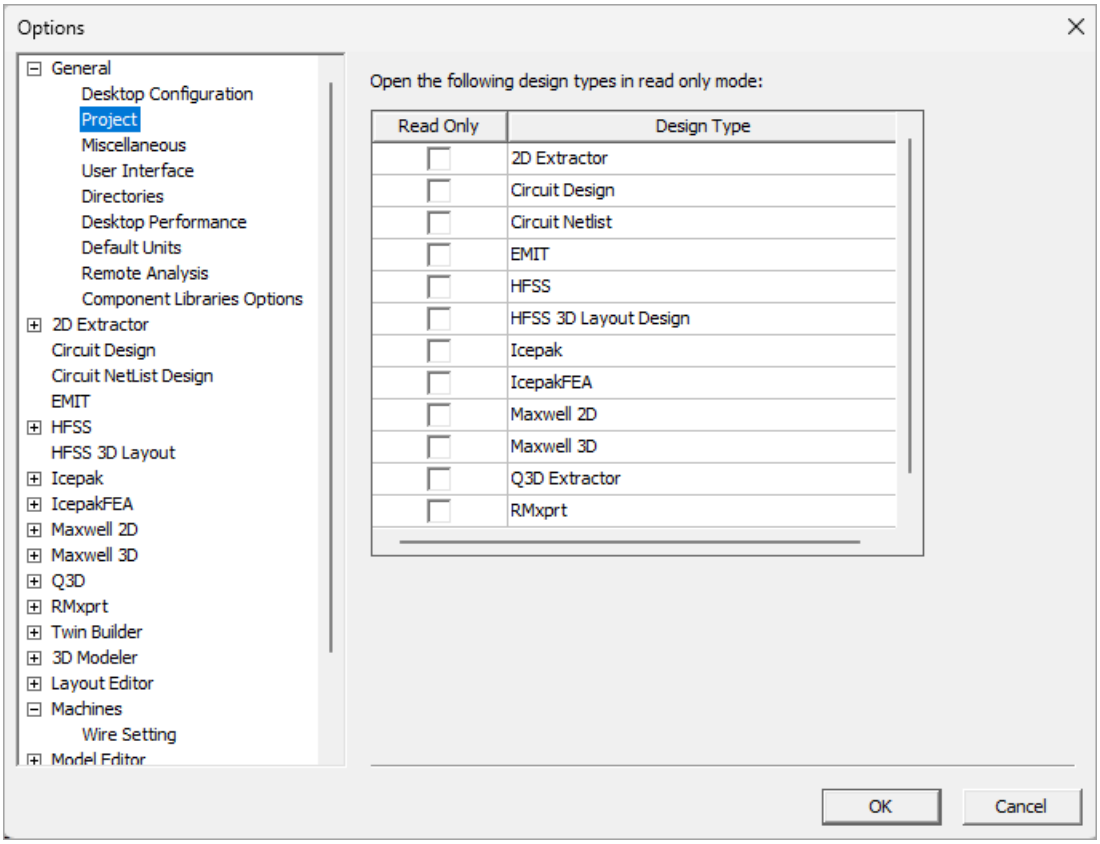
From there, enable or disable options, and click **OK**. You will need to restart to enable your selections.

Deprecated Features opens a window for activating features that have been deactivated, but not removed from the software.

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.

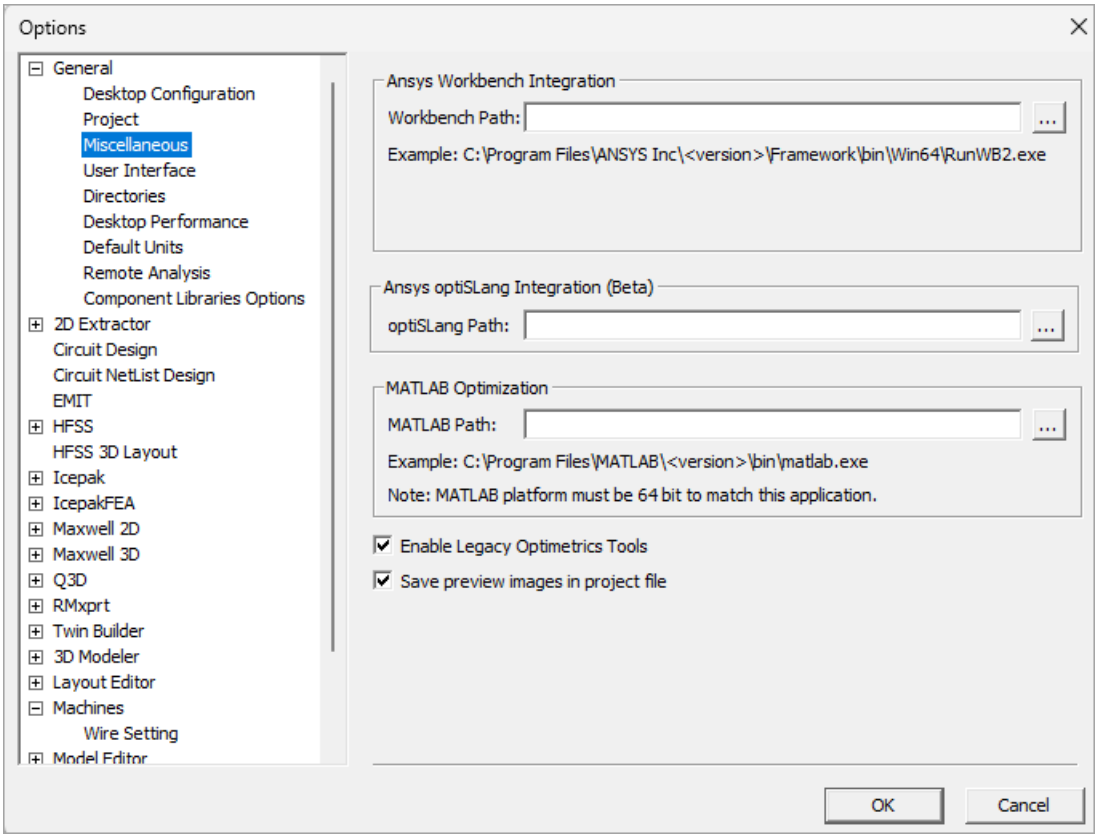
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

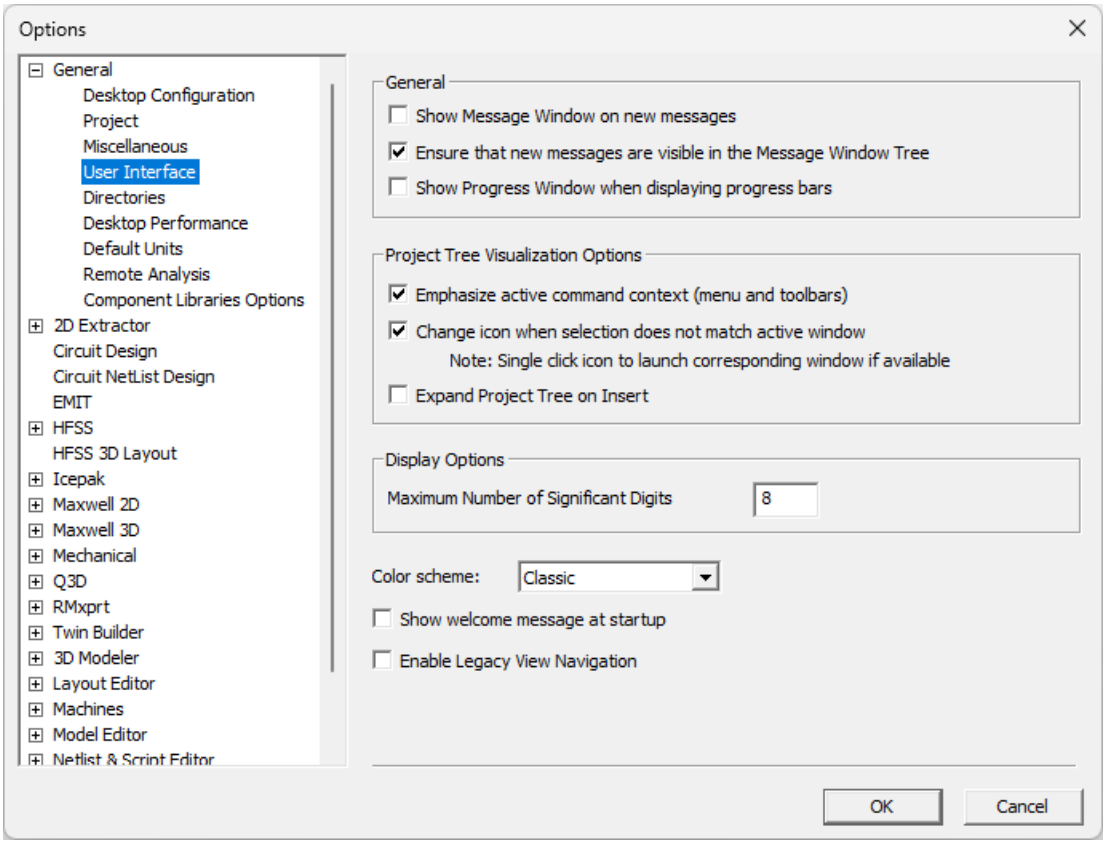
MATLAB Optimization allows you to specify the path to your installation of MATLAB. MATLAB can be used as an Optimizer.

> **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 2026 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:

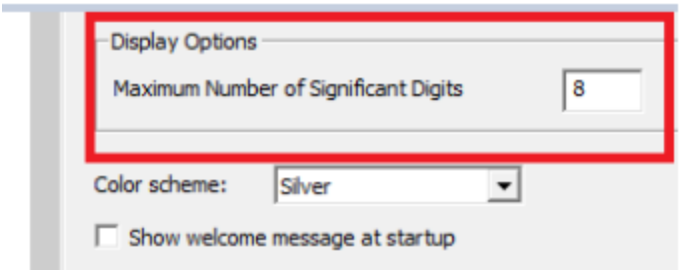
- **Show Message Window on new messages** – when selected, the message window automatically opens if a message arrives.
- **Ensure that new messages are visible in the Message Window Tree** – when selected, the message window expands as needed to display messages.
- **Show Progress Window when displaying progress bars** – when selected, the progress window automatically opens while simulations are in progress.

In the **Project Tree Visualization Options** area, options include:

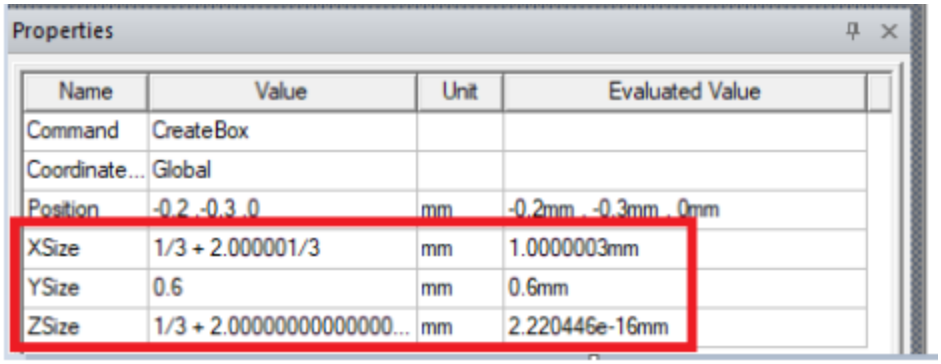
- **Emphasize active command context** – when selected, active elements in the Project Tree display in bold text.
- **Change icon when selection does not match active window** – when selected, a small, window-shaped overlay icon displays in the corner of the selected Project Tree element. This icon changes when the data in the active window is unrelated to the 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

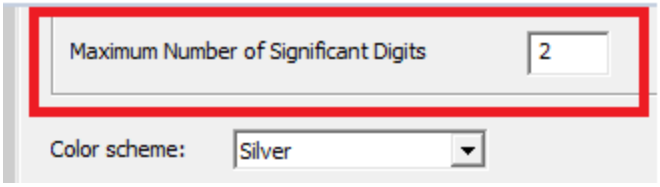
In the **Display Options** area, specify the **Maximum Number of Significant Digits** to display. The default is 8 and the maximum is 20. This affects the digits displayed in the **Solutions** dialog box, evaluated variable values, Animation dialog boxes, Optimetrics, Reports, and so forth. For example, here is the default.



So some evaluated variable values with significant digits appear as follows.

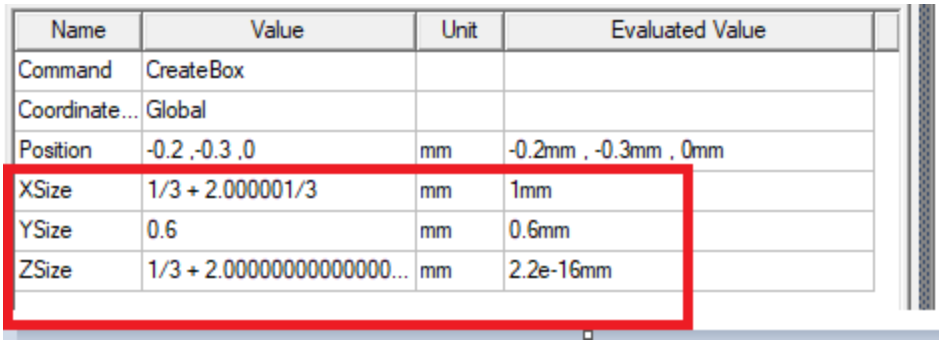


Change Precision = 2. This lets you display near-zero values in the Properties window as 0.



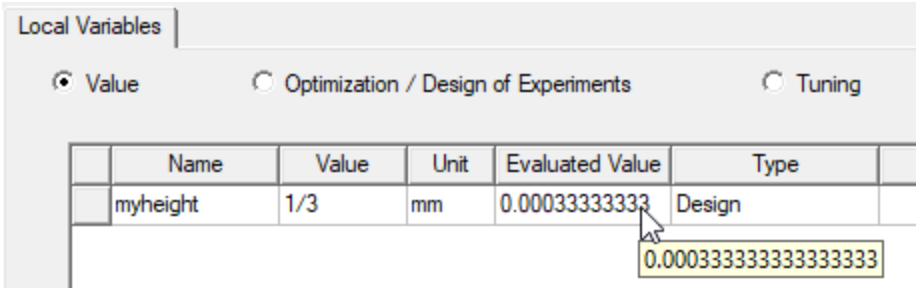
You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

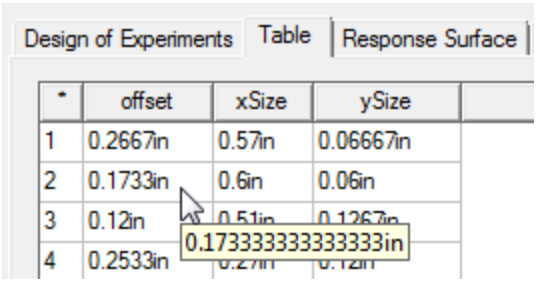


Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.2 ,-0.3 ,0	mm	-0.2mm , -0.3mm , 0mm
XSize	1/3 + 2.000001/3	mm	1mm
YSize	0.6	mm	0.6mm
ZSize	1/3 + 2.00000000000000...	mm	2.2e-16mm

You can still see the full precision values in tooltips by holding a cursor over the displayed value.



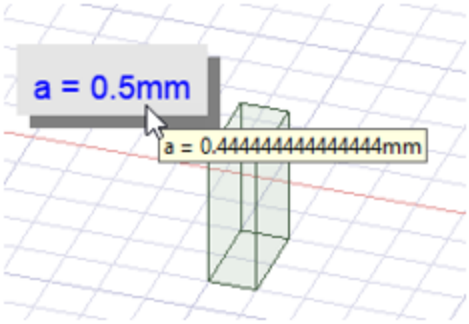
In the case of variable values, if you have assigned more significant digits, you will see these when editing the variable value. In the case of table displays of values, the tooltip display shows all available digits when the mouse pointer is over a result:

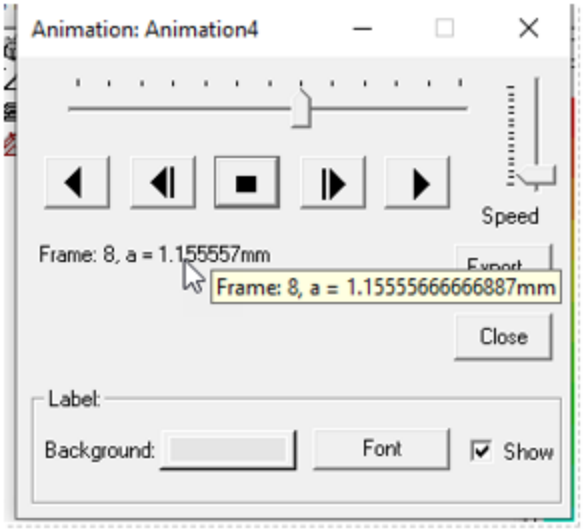


If you set the **Maximum Number of Significant Digits** to a lower value, Change Precision = 2. You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

The tooltip functions to show internal digits in throughout the Ansys Electronics Desktop interface.

Ansys Electromagnetics Suite 2026 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.

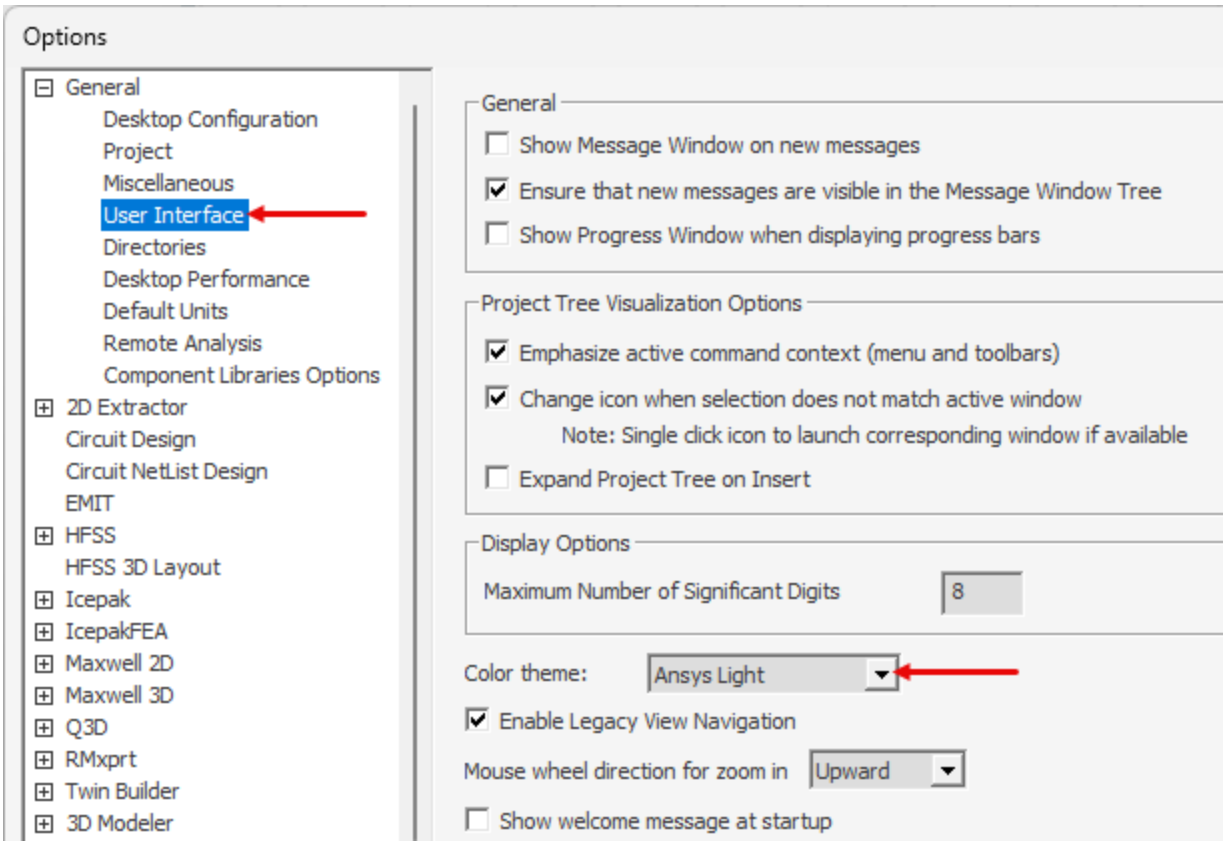
Choosing a Color Scheme

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

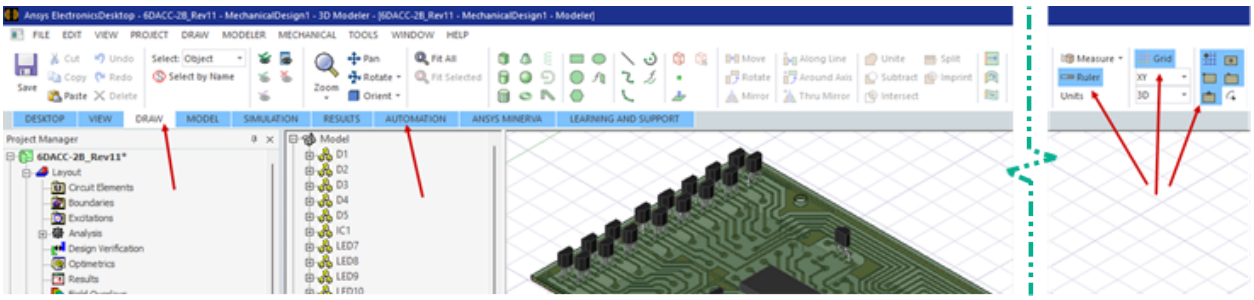
1. Access the **Options** dialog box using one of the following two methods:
   - On the **Desktop** ribbon tab, click **General Options**.
   - From the menu bar, click **Tools** > **Options** > **General Options**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the tree at the left side of the dialog box, expand **Desktop Configuration** and select **User Interface**:

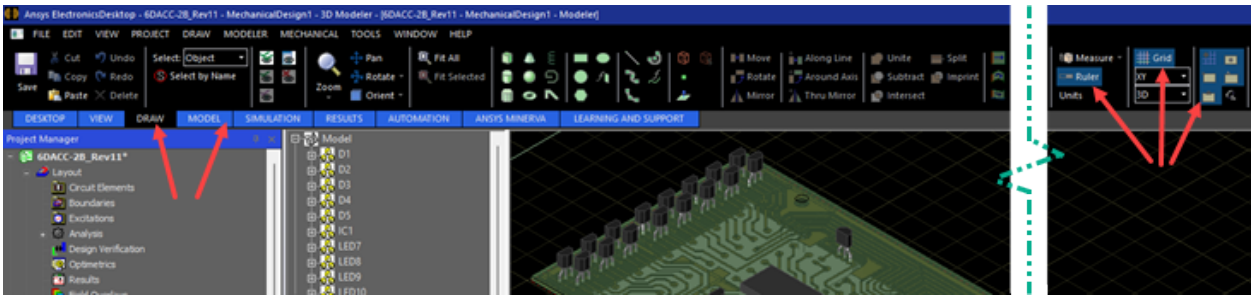


3. From the **Color scheme** drop-down menu, choose one of the following three options:
   - **Ansys Light:** A light gray and white scheme with light blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a white background (including the tab):

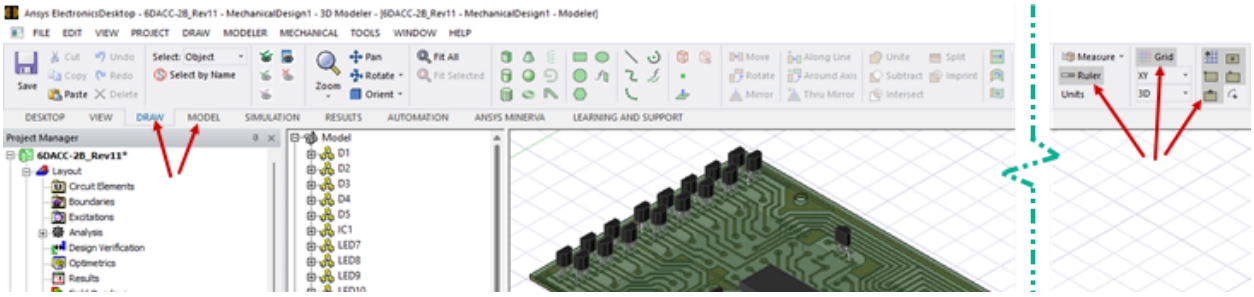


   - **Ansys Dark (beta):** A dark gray and black scheme with medium blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a black background (including the tab):

Ansys Electromagnetics Suite 2026 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:

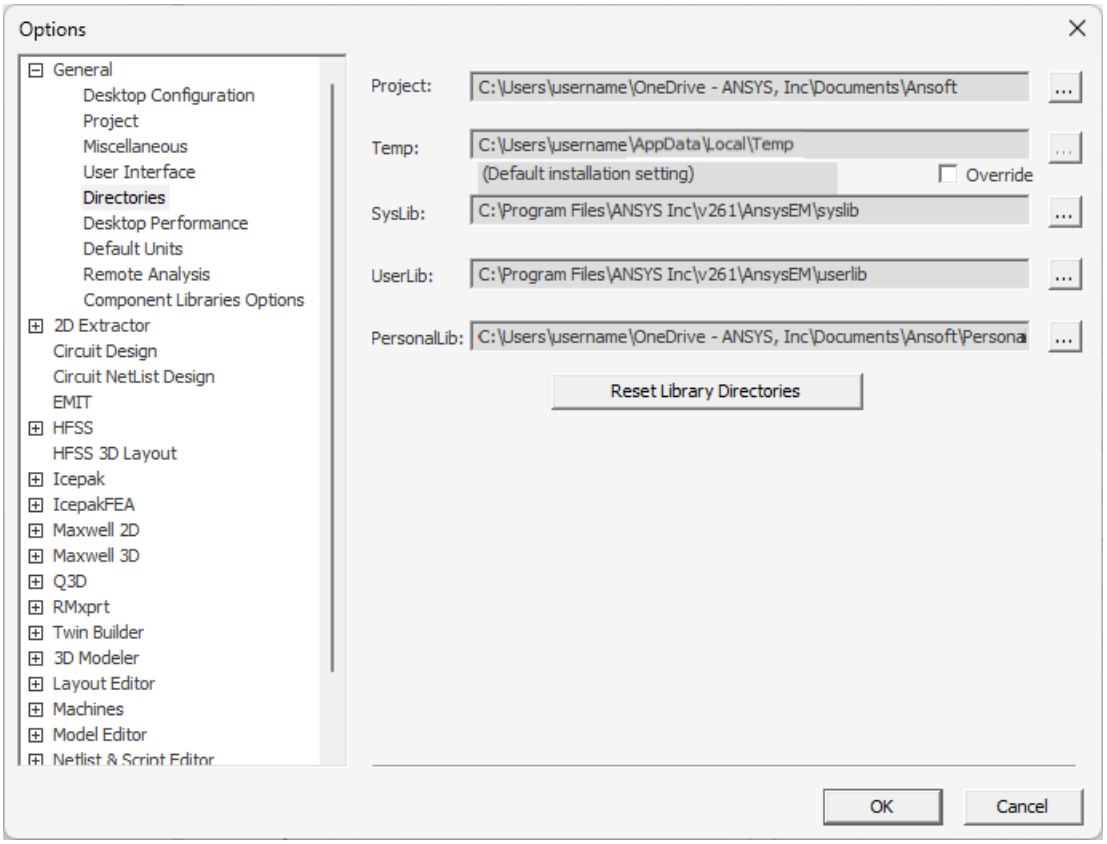


4. Click **OK** to close the **Options** dialog box.

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 2026 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.

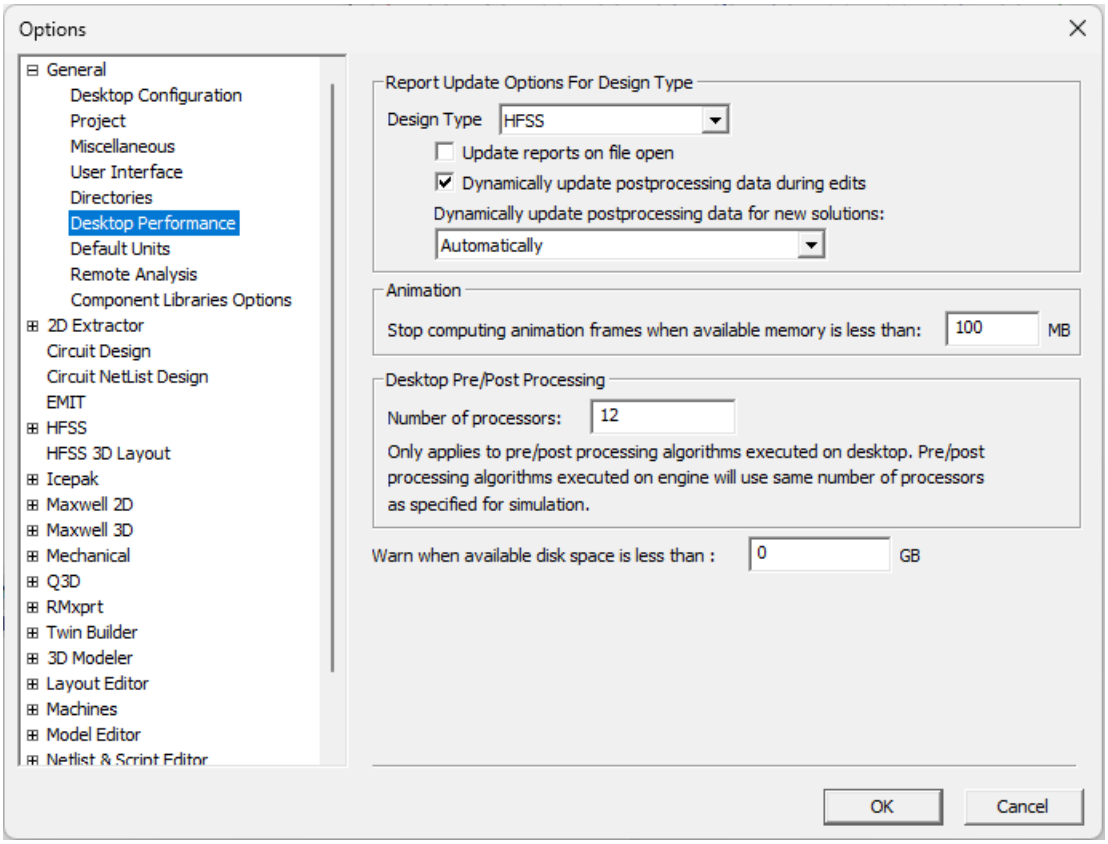
If you need to restore defaults, click **Reset Library Directories**.

> **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/postprocessing, disk space warnings, and updating reports.

Ansys Electromagnetics Suite 2026 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 2026 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 postprocessing (not solve or simulation). Preprocessing 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 2026 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

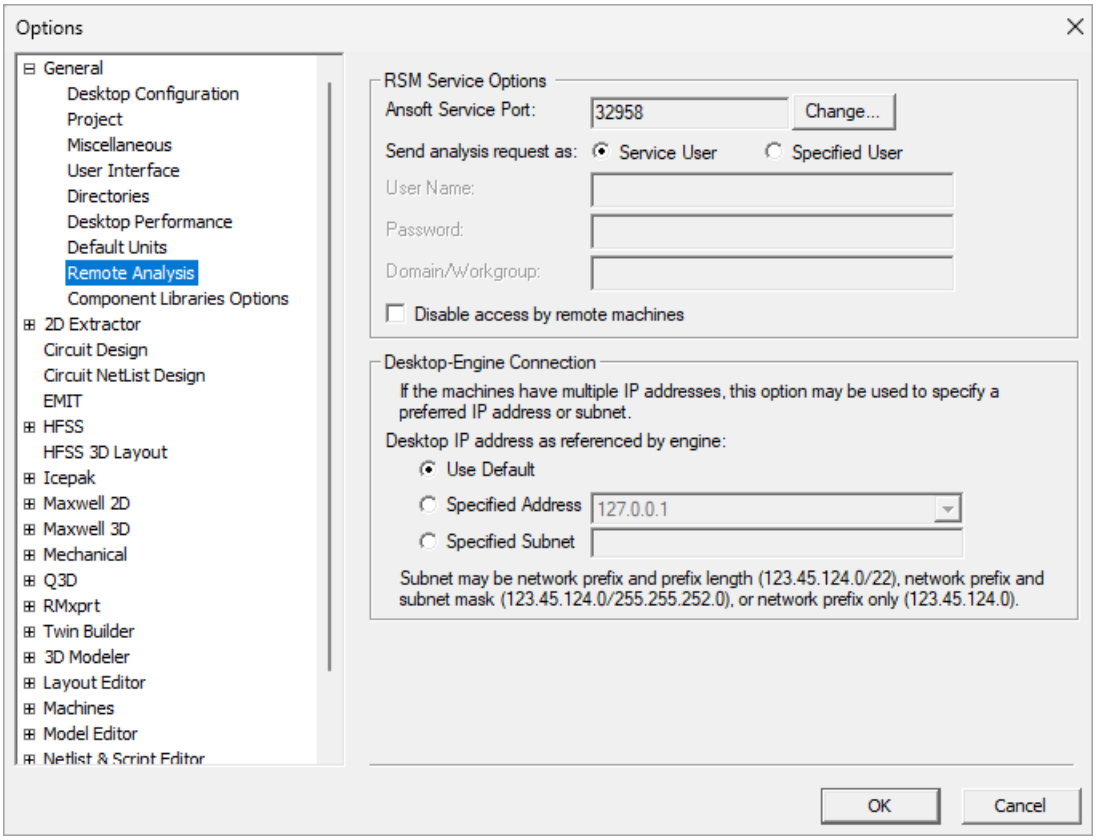
Under General Options, **Default Units** options allow you to set default units for the following:

- 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 2026 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.

- **Disable Access By Remote Machines** – If desired, select to disable access for remote machines.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

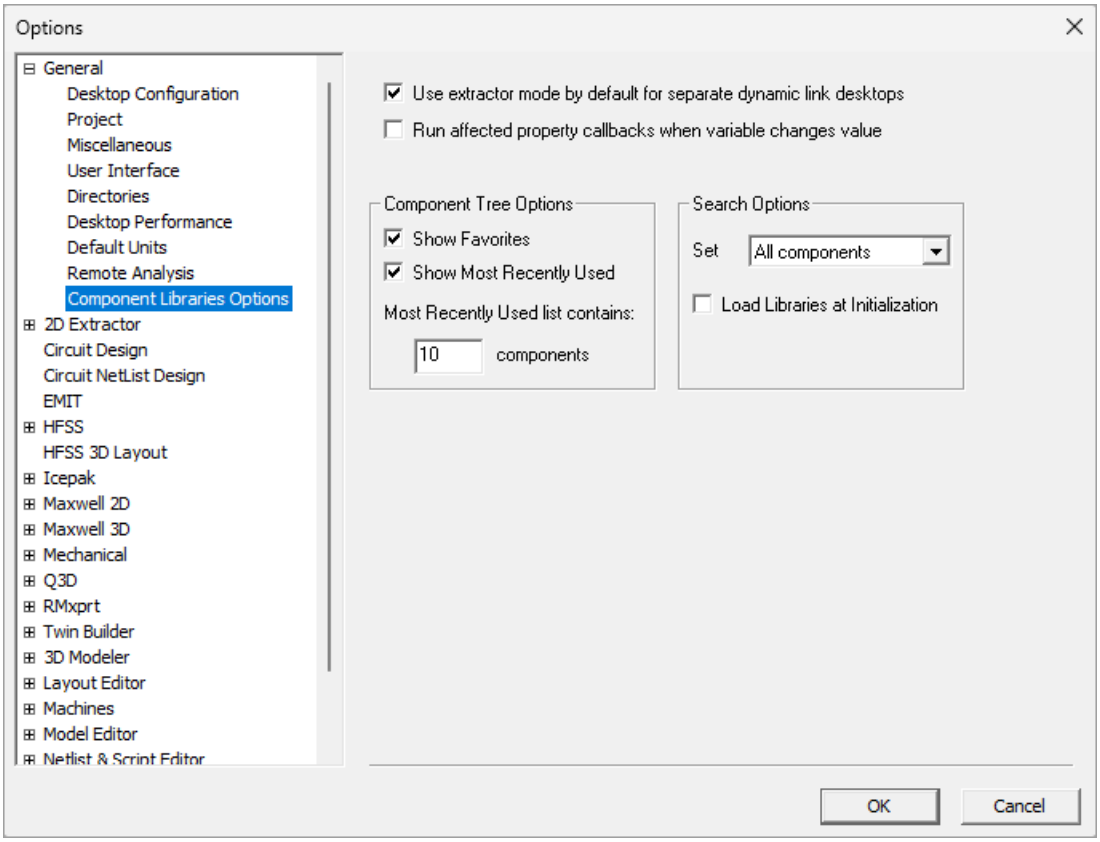
```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

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.

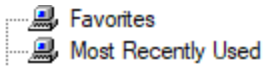
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

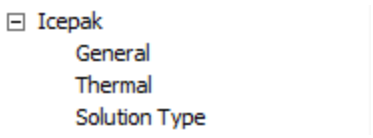
- **Set** – specifies the search set when searching the Component Libraries. Choose to search **All components**, **Current list only**, or **Append to current list**.
- **Load Libraries at Initialization** – this option can slow initialization.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Icepak Options

To set Icepak options:

1. Click **Tools>Options>General Options** to display the **Options** dialog box. Select the Icepak entry in the hierarchical window, and press the + sign to show the choices under Icepak.



2. Under **General** options, select or clear the following check boxes:

- Auto create region

When this is checked, the global region, which represents the computational domain, is automatically created when an Icepak design is inserted into a project. The region's padding can be edited in the Properties window.

- Save before solving
- Apply variation deletions immediately

  Checking this saves disk space.

- Save Optimetrics field solutions

  Checking this uses more disk space.

- Allow import of inactive objects from Classic Icepak models (.tzr)

When this is checked, inactive objects defined in the Classic Icepak project are imported and placed in a separate design named with the suffix "_inactive."

3. Under **Thermal**, select or clear the following check boxes and settings:

- Use Wizards for data input when creating new boundaries

  When this is checked, the creation of boundaries and native 3D components uses a Wizard to guide you through the process. When this is not checked, the creation of boundaries and native 3D components displays a dialog with tabs for different kinds of information.

- Duplicate boundaries/mesh operations with geometry

  When this is checked, you can duplicate a boundary or excitation when its geometry is pasted or duplicated. See Duplicating Boundaries and Excitations with Geometry.

- Visualize boundaries on geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When this is checked, boundaries on geometries are displayed. Unchecking this turns off boundary visualization, and speeds up the display for complex models.

4. Under **Solution Type**, you can set the default solution type options when you initially insert a project.
    - **Temperature and Flow**: for solving the energy equation for the temperature distribution together with the flow equations.
    - **Temperature Only**: for solving the energy equation for the temperature distribution only.
    - **Flow Only**: for solving the problem for flow only.

> **Note:**
>
> Not all option settings are applied immediately in the current design. **Use Wizards for data input for creating new boundaries**, **Visualize boundaries on geometry**, and **Solution Type** options take effect in newly created designs.

Setting 3D Modeler Options

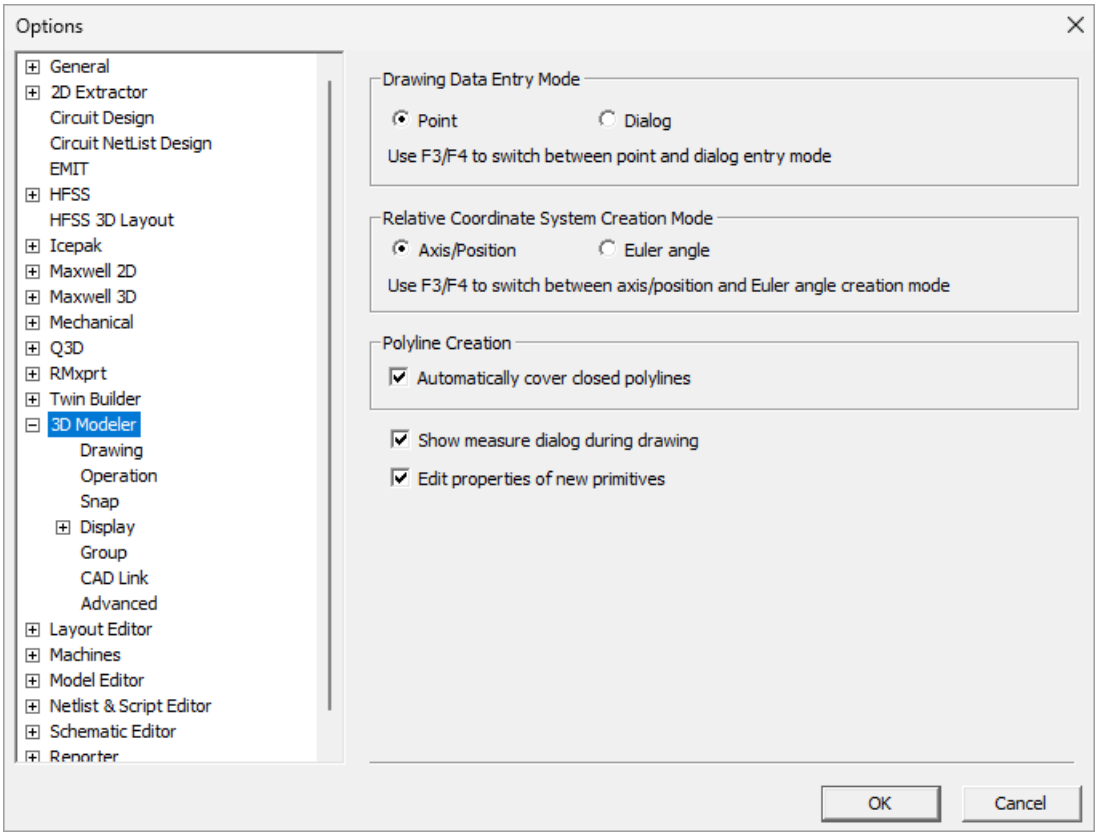
To set 3D Modeler options in Ansys Electronics Desktop:

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:

Ansys Electromagnetics Suite 2026 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 **3D Modeler** to display the associated options:

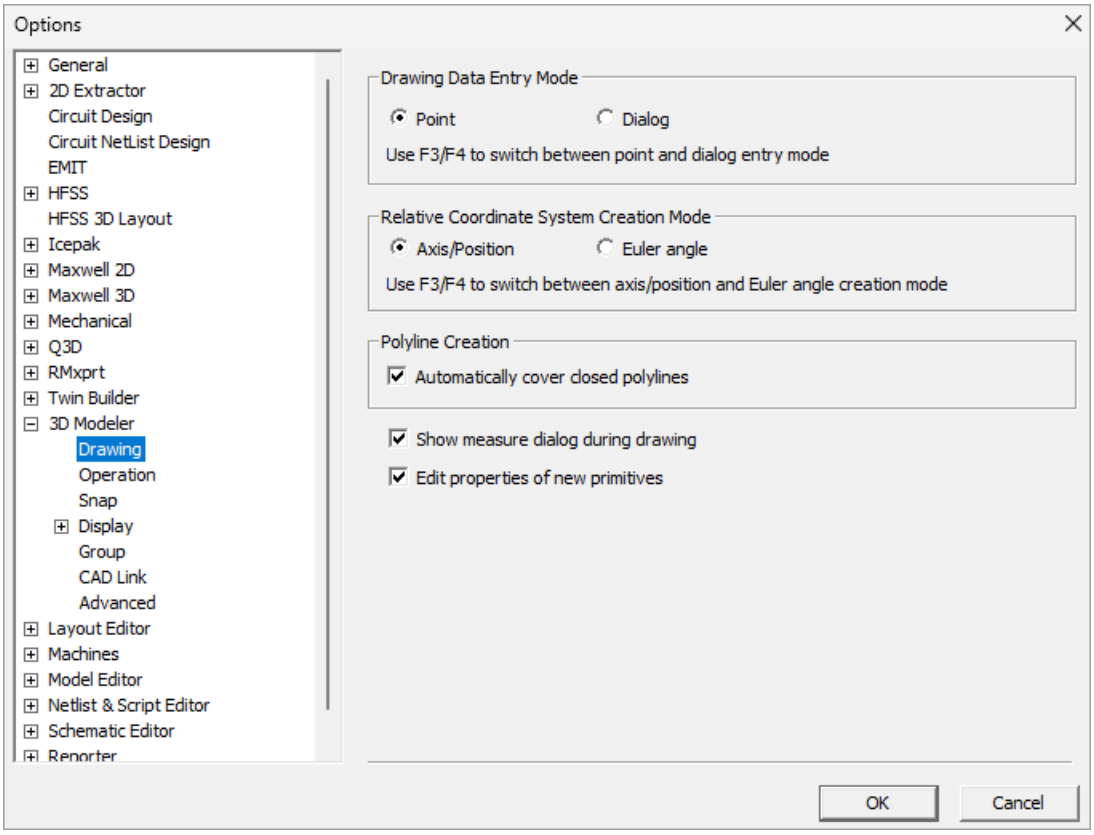
- Drawing
- Operation
- Snap
- Display
- Group
- CAD Link
- Advanced

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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.

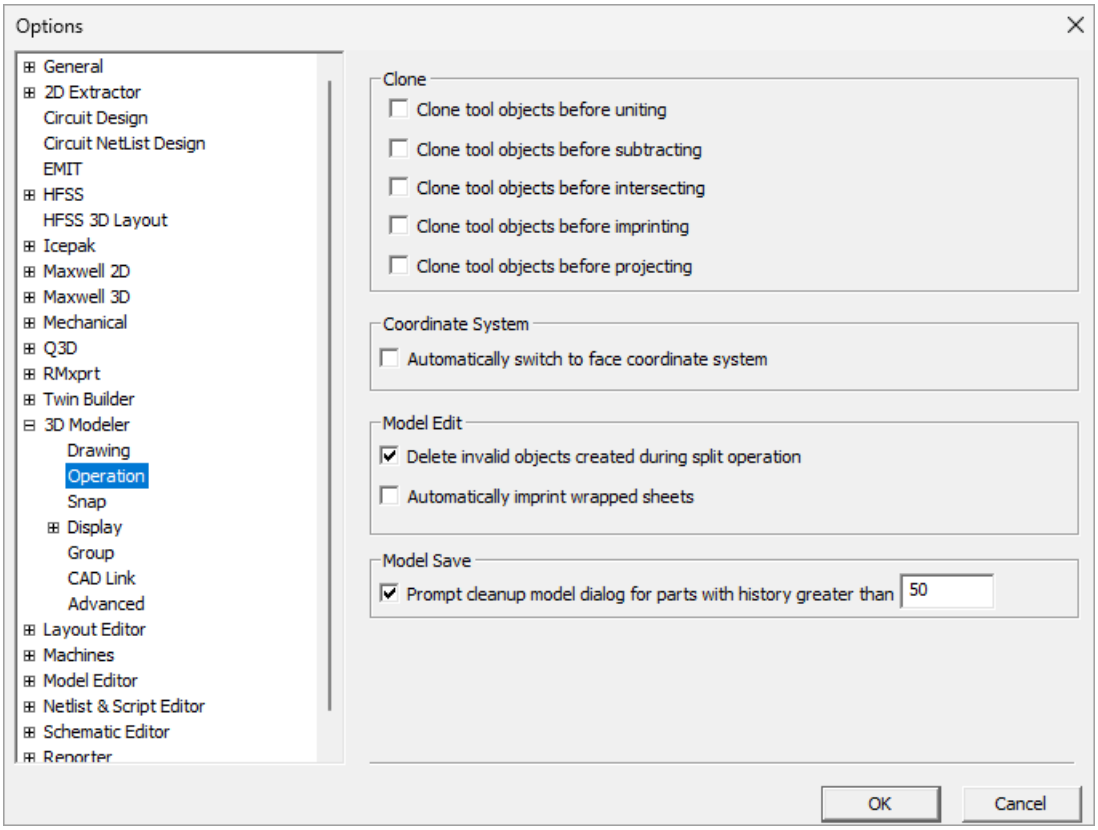
Other options are:

- **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 displays when you create a new object in the modeling workspace.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

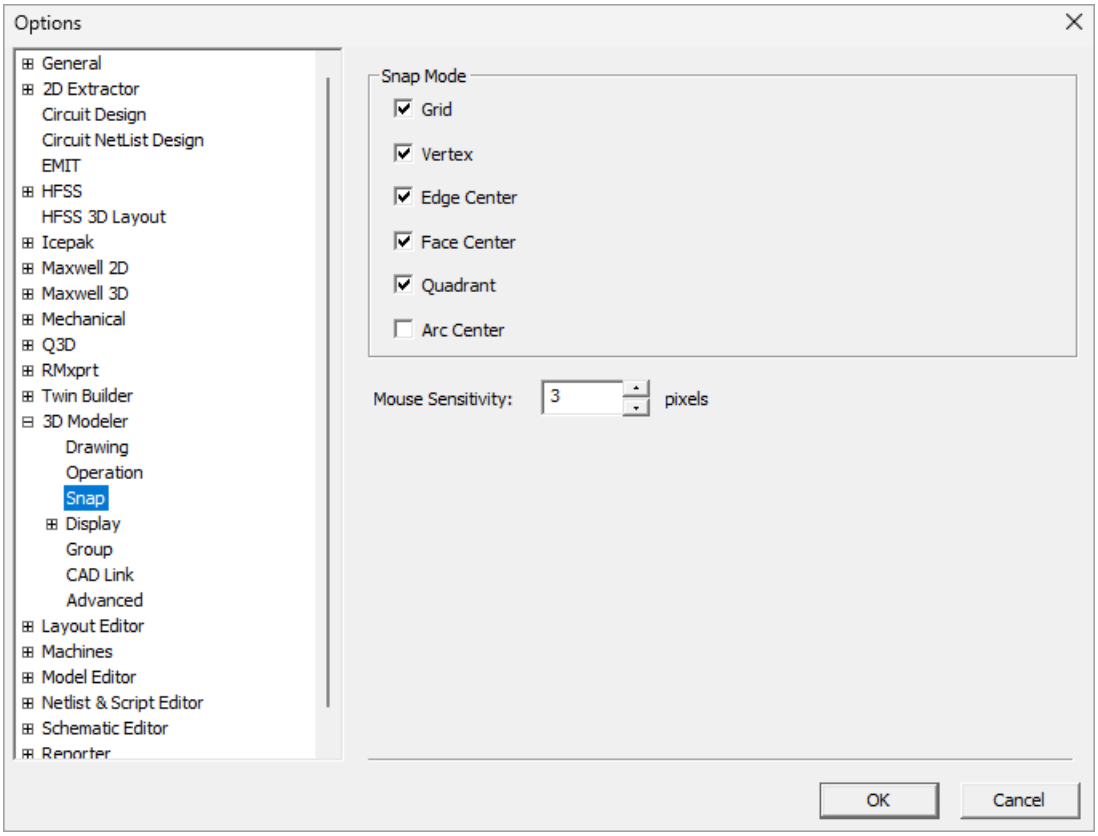
- **Delete invalid objects created during split operation** – When selected, the modeler deletes invalid objects created during split operation. When deselected, invalid objects can be freely created.
- **Automatically imprint wrapped sheets** – Imprinting is a boolean operation, whereas wrapping is 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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

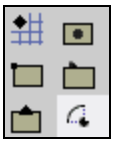
In the **Snap Mode** area, select the points you would like the cursor to snap to:

- Grid
- Vertex
- 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.



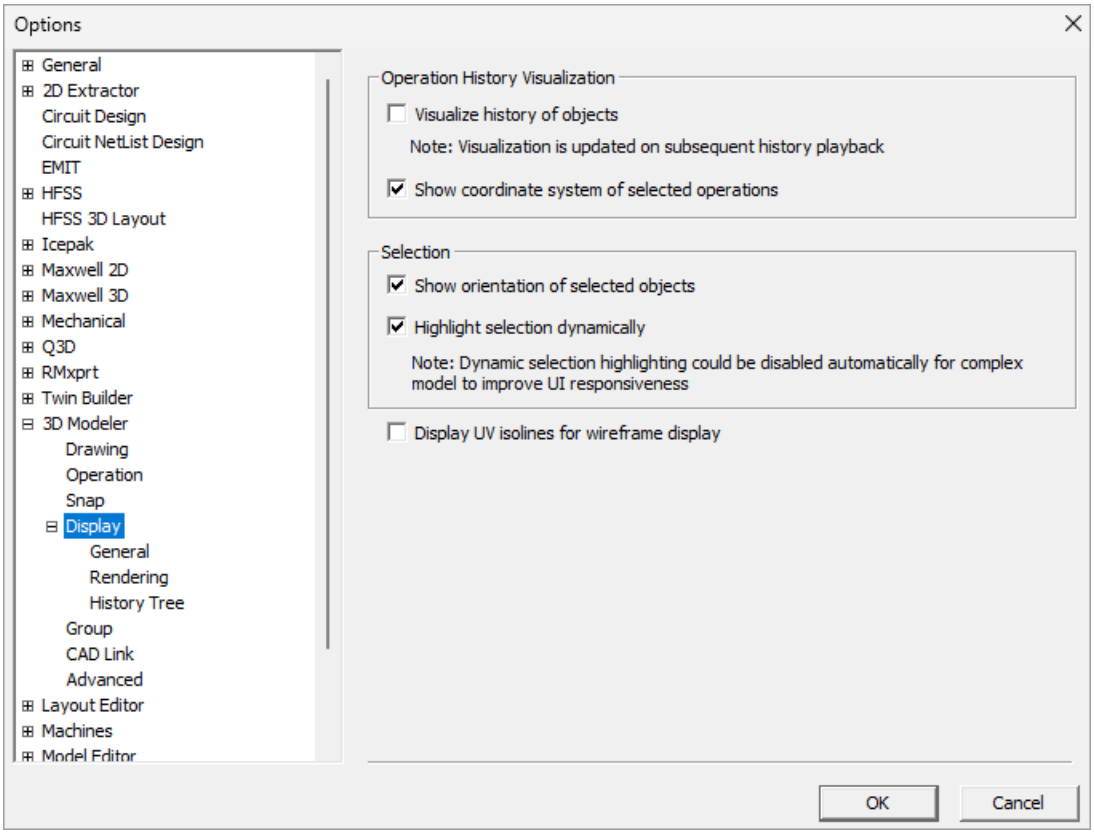
Hover over each icon for a tooltip indicating its equivalent menu option.

> **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



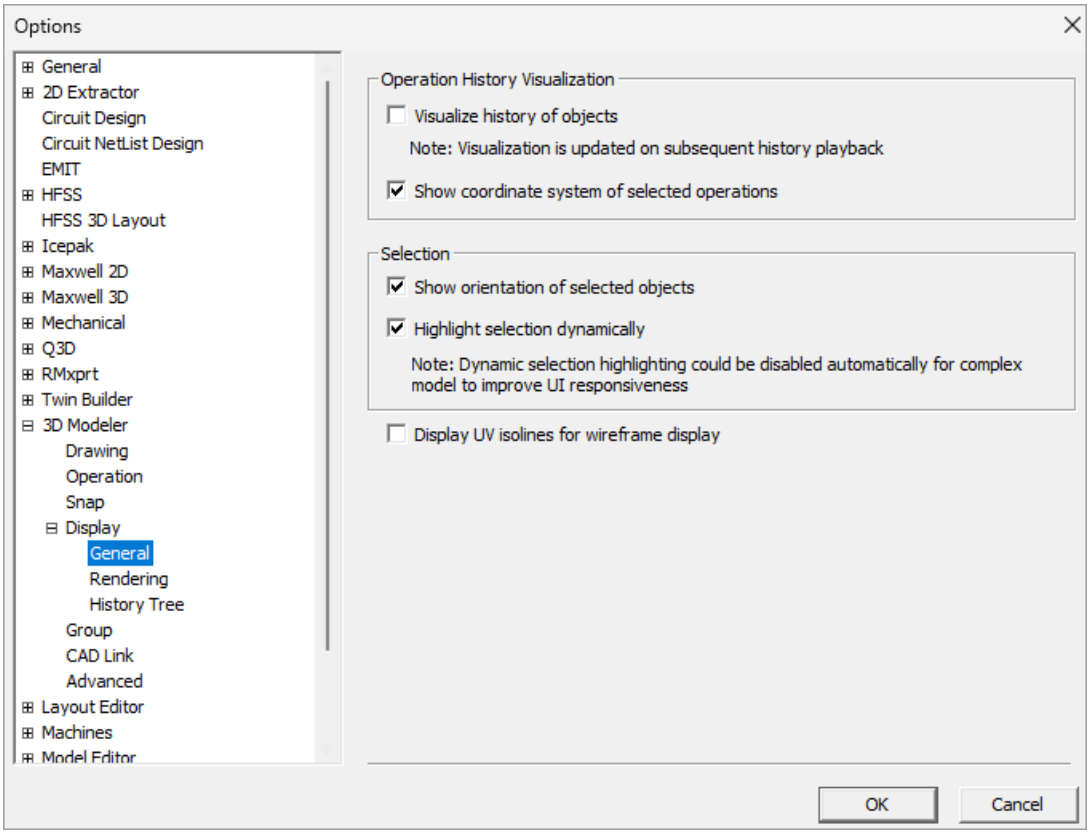
In the left pane, select the entries below **Display** to display the associated options:

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

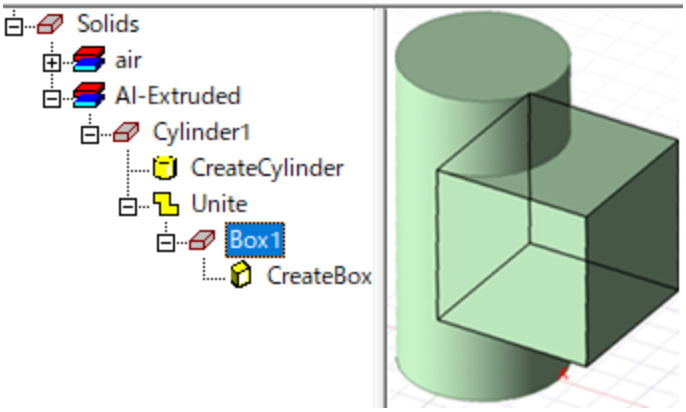


In the **Operation History Visualization** area, set the following options:

- **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 2026 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:

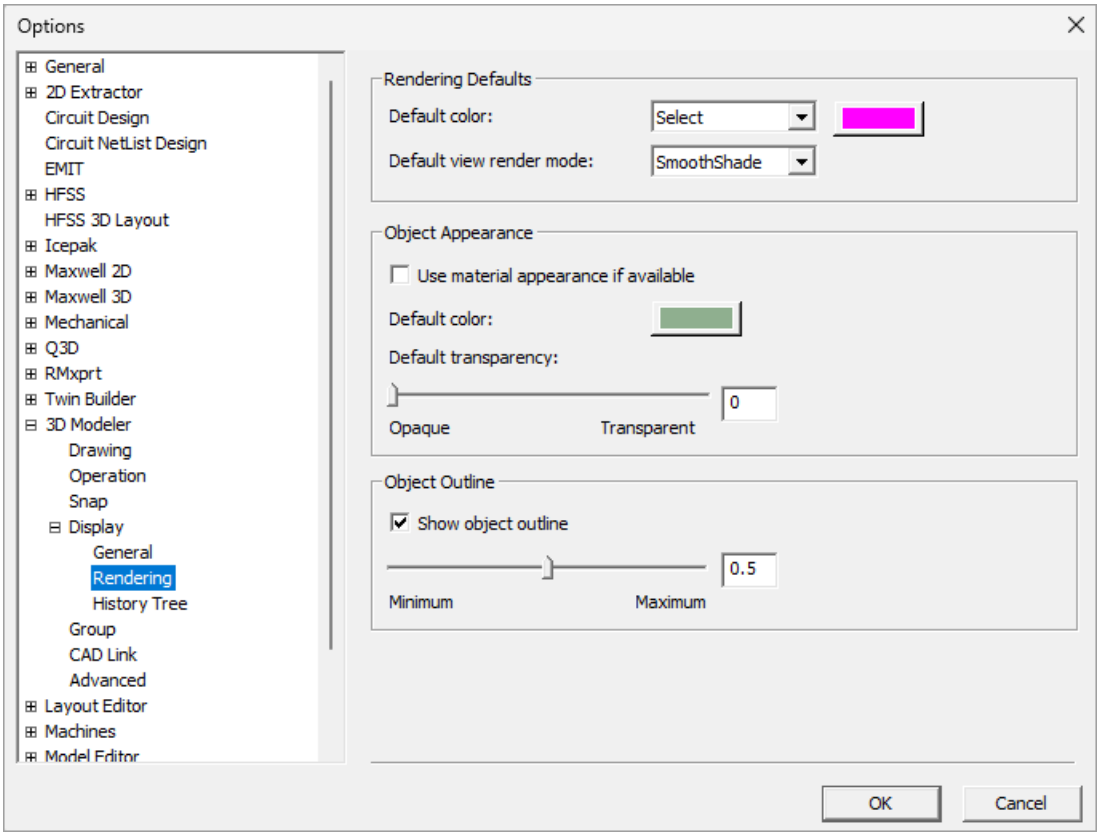
- **Show orientation of selected objects** – Select or clear.
- **Highlight selection dynamically** – This option causes objects or faces (depending on the selection mode) to be highlighted when you pass the mouse pointer over them. You may want to turn this off for complicated models as it can reduce 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 2026 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:

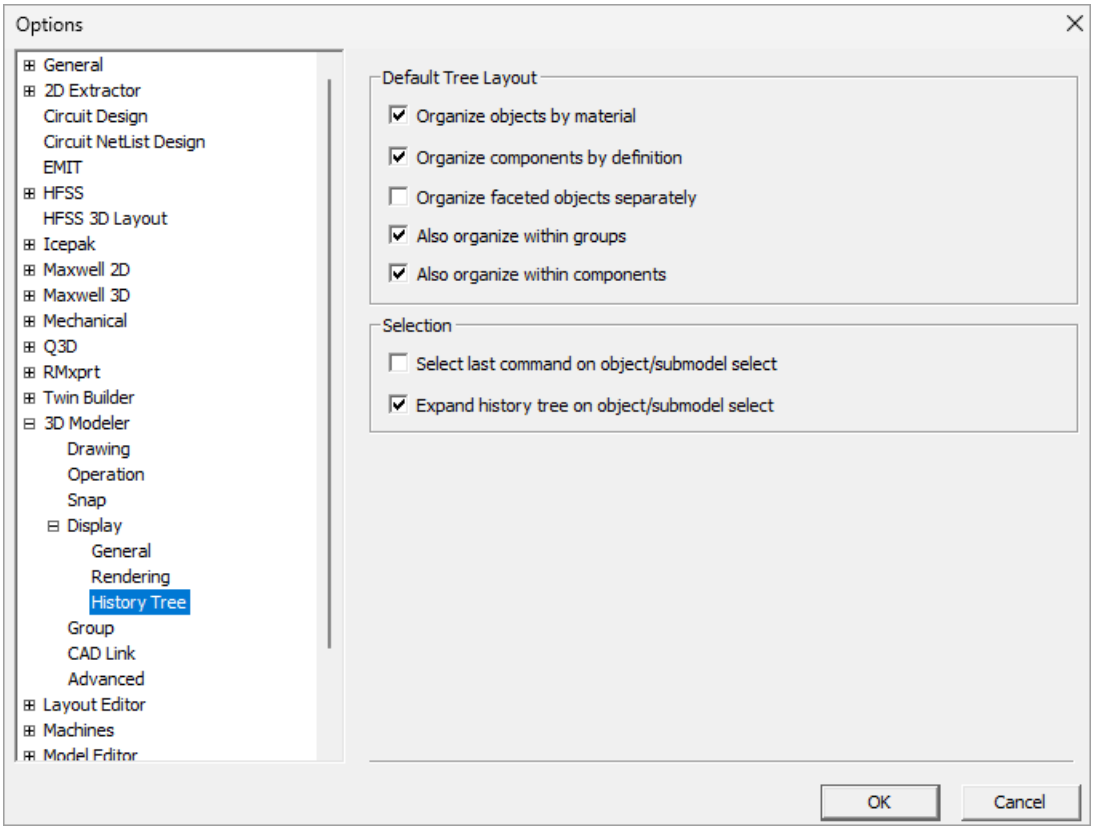
- **Use material appearance if available** – When this option is selected, default color and transparency used for newly created objects come from material settings. Some materials may not have default appearance specifications; in those cases, default color and transparency specified under **Rendering 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 2026 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**

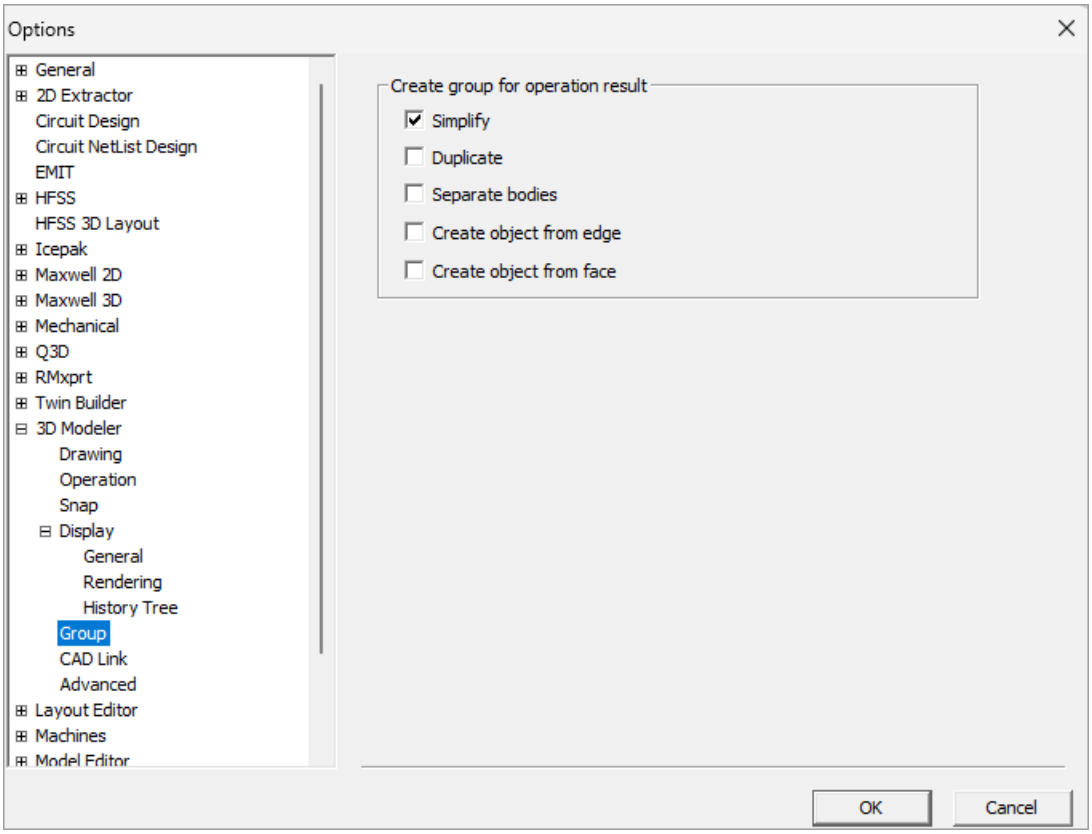
In the **Selection** area, set the following options:

- **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 2026 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**

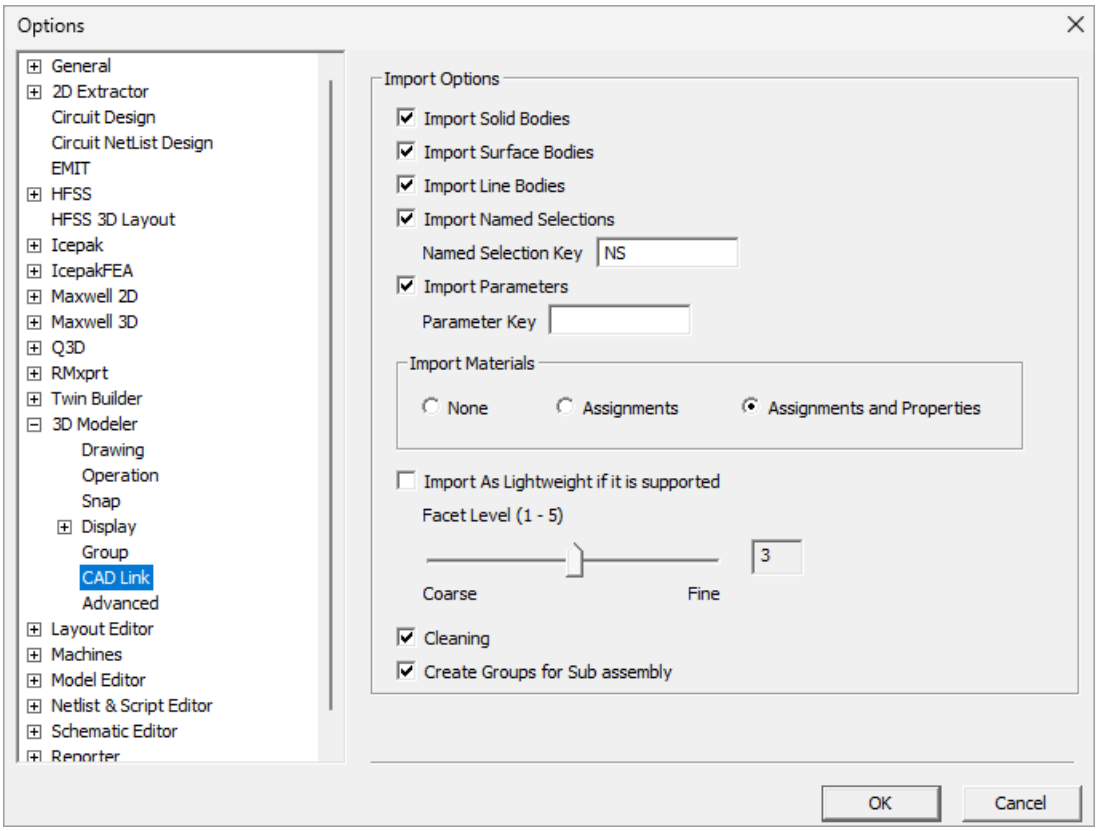
When any selected action is performed, a group will be created in the History Tree.

3D Modeler Options: CAD Link Options

Under 3D Modeler options, the **CAD Link** options determine how Ansys Electronics Desktop imports designs from other CAD programs. In addition to Ansys Discovery, Ansys Electronics

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Desktop can import models from NX, Solid Edge, Creo Parametric, Creo Elements/Direct, AutoCAD, and Inventor projects.

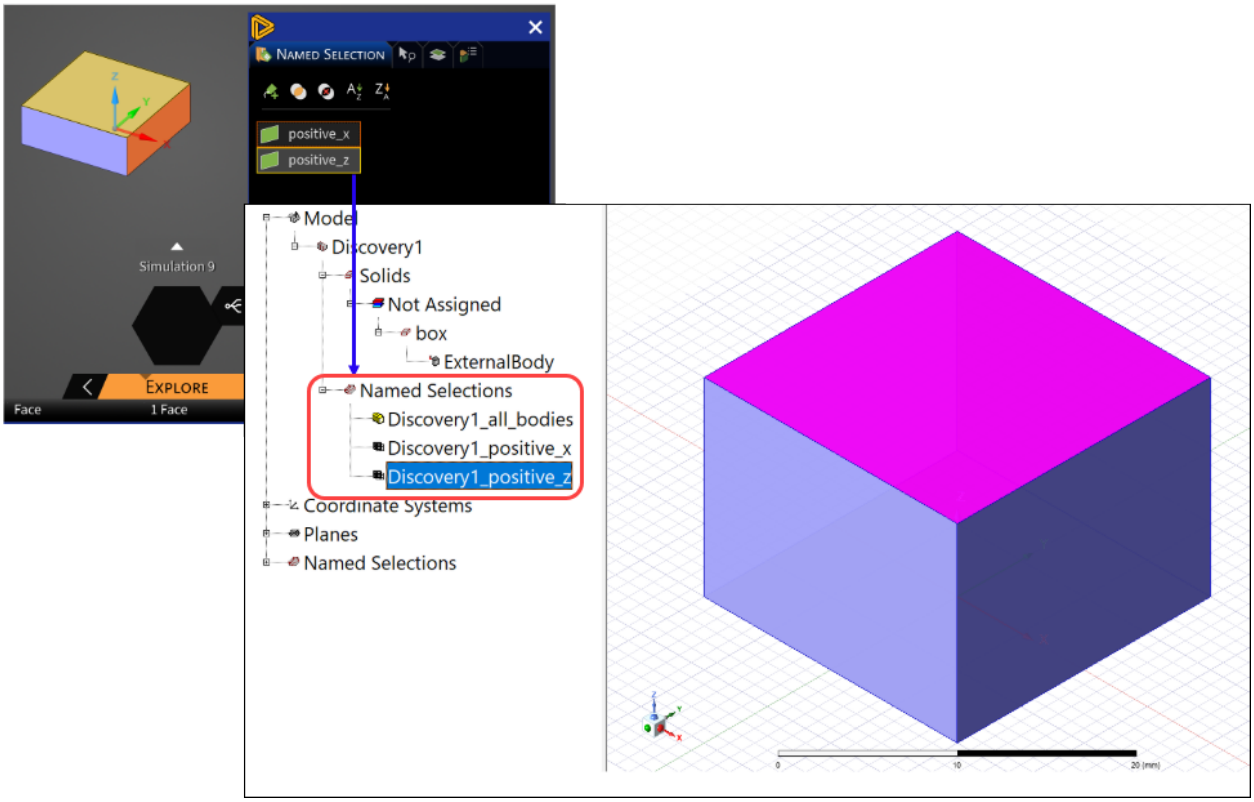


To specify CAD link settings, select or clear the following check boxes in the **Import Options** section:

- **Import Solid Bodies** – Enabled by default.
- **Import Surface Bodies** – Enabled by default.
- **Import Line Bodies** – Enabled by default. This option is not supported for Inventor models.
- **Import Named Selections** – Import groups of entities. The **Named Selection Key** can be used to filter out named selections.

  For example, faces can be grouped and named in Discovery, and if this option is checked, the named selections will be preserved during import and listed in the 3D Modeler history tree under a **Named Selections** branch.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note the following limitations for this option:

- For AutoCAD: Named selections are limited to entities that are assigned to different layers.
- For Creo Elements\Direct: Named selections are limited to bodies with assigned colors.
- For Inventor: Body names cannot be assigned, so ID preserving with named selection feature is not supported.

> **Tip:** Named selections can be used to preserve entity IDs during parametric variations. See 3D Modeler CAD Link for more details.

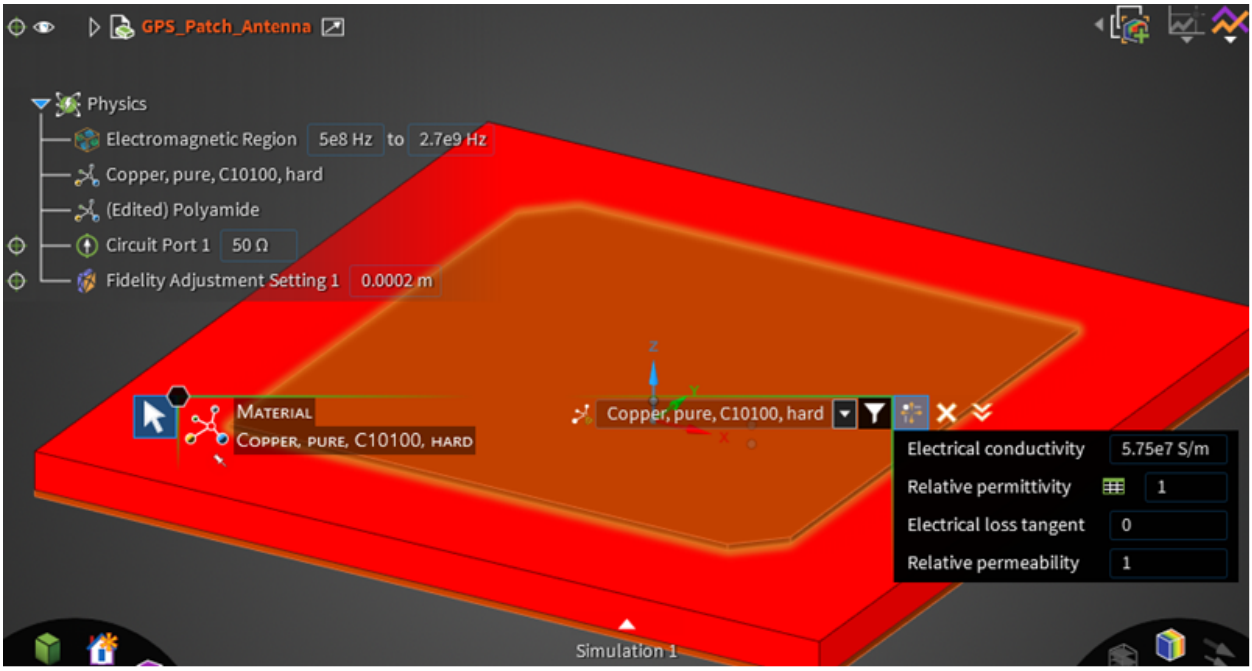
- **Import Parameters** – Enabled by default. Parameters are imported as unitless values and must have unique names. Note that Discovery model parameters are imported with units. This option is not supported for Creo Elements/Direct and AutoCAD.
  - **Parameter Key** – A string used to filter out parameters. By default, no string is entered and all parameters are imported.
- **Import Material Assignments** – **None** by default. This option is not supported for Creo Elements/Direct or AutoCAD.

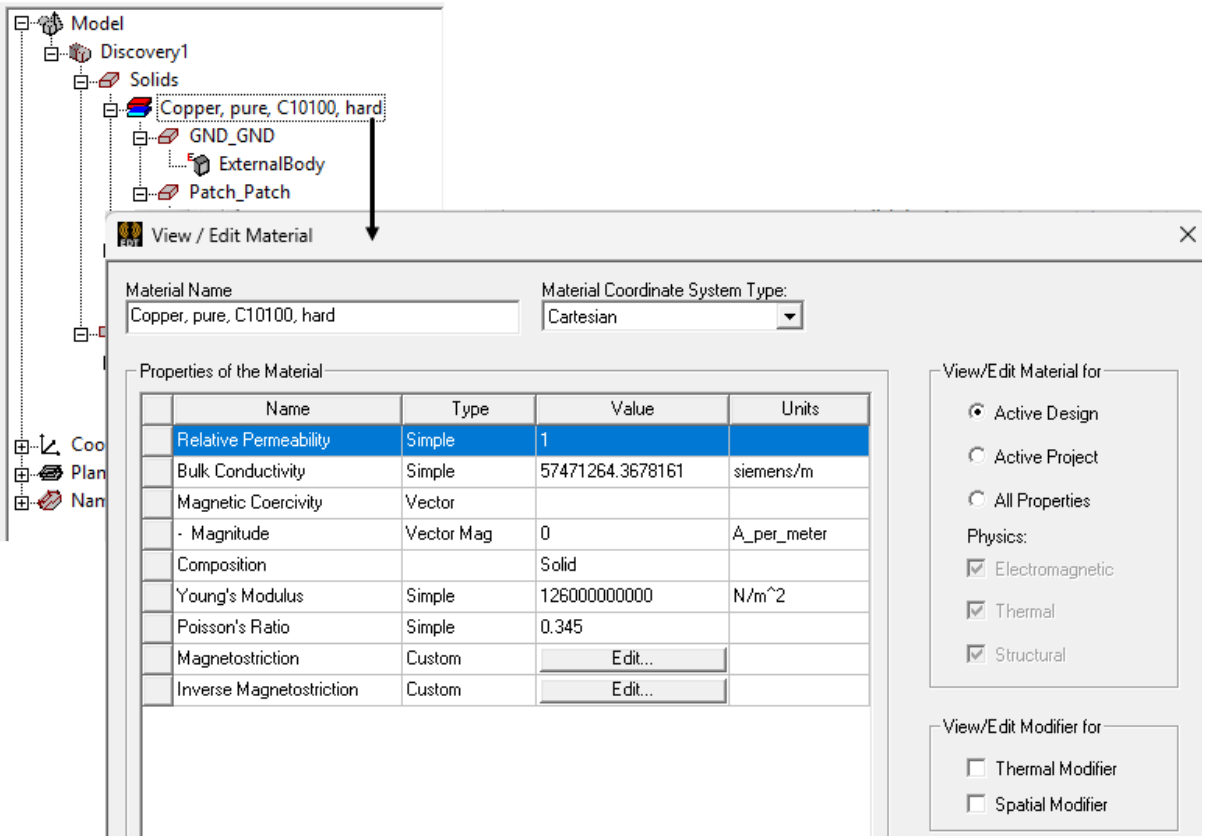
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Assignments** – Imports the material names without the material properties. The material properties will be obtained from a material with the same name that exists in the user's project. With this option, the system will first search for the material within the project's materials. If no matching material is found, the search will extend to the sys libraries. (See Working with Material Libraries for details on libraries.) If a material with the same name exists in a sys library, it will be imported into the project from the library. If a material with the same name cannot be found in the project, a violation will be issued after the link is created.
- **Assignments and properties** – Imports material with its material properties. The values must be in SI units. Discovery supports importing mechanical, electrical, and thermal properties. For other CAD programs, mechanical material property import is only available when connecting to Icepak and IcepakFEA designs.
  A project material will be created in AEDT with imported properties. Note that if a project material with same name already exists, then it will be used instead of a new material being created.

The example below shows a device imported from Discovery with the **Assignments and properties** option checked.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





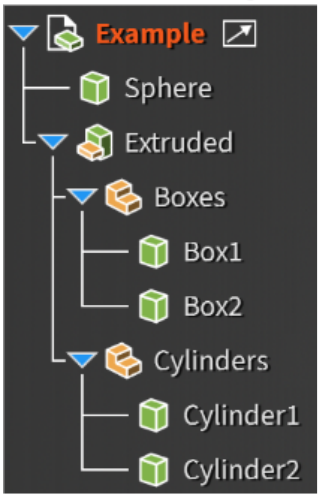
- **Import As Lightweight if it is supported** – This feature allows you to import lightweight geometry into a design that supports lightweight bodies. When the option is checked, the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

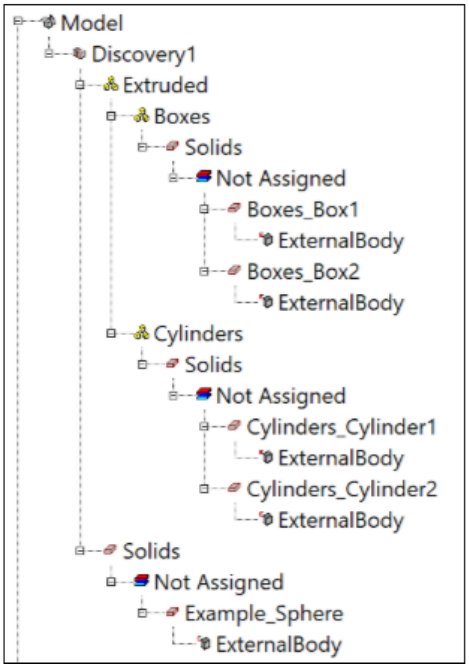
normal "unfaceted" parts in the CAD model are imported and faceted with the specified facet level. The Facet Level can be set on a 1-5 range, from coarse to fine. For CAD software other than Discovery, the faceted parts cannot be imported. For a faceted Discovery model, the facets will be imported directly, and the Facet Level setting will be ignored.

- **Clean** – Use to remove unwanted features when transferring geometry to Ansys Mechanical or Ansys Meshing. Clicking the check box will result in the removal of all unessential edges, vertices, and features; for example, small faces will be merged into larger faces.
- **Create Groups for Sub assembly** – If this option is checked, CAD sub-assemblies will be preserved. Each sub-assembly will be its own group in the Modeler history window.

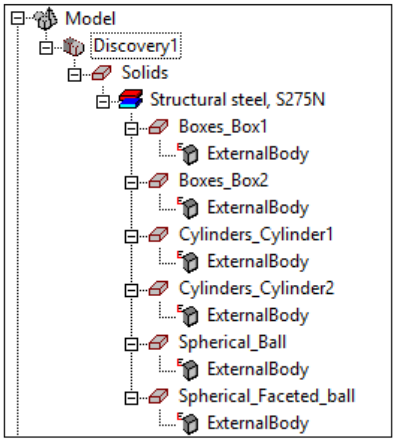
Discovery Model with Assembly



Imported model with Create Sub assembly checked



Imported model with Create Sub assembly unchecked



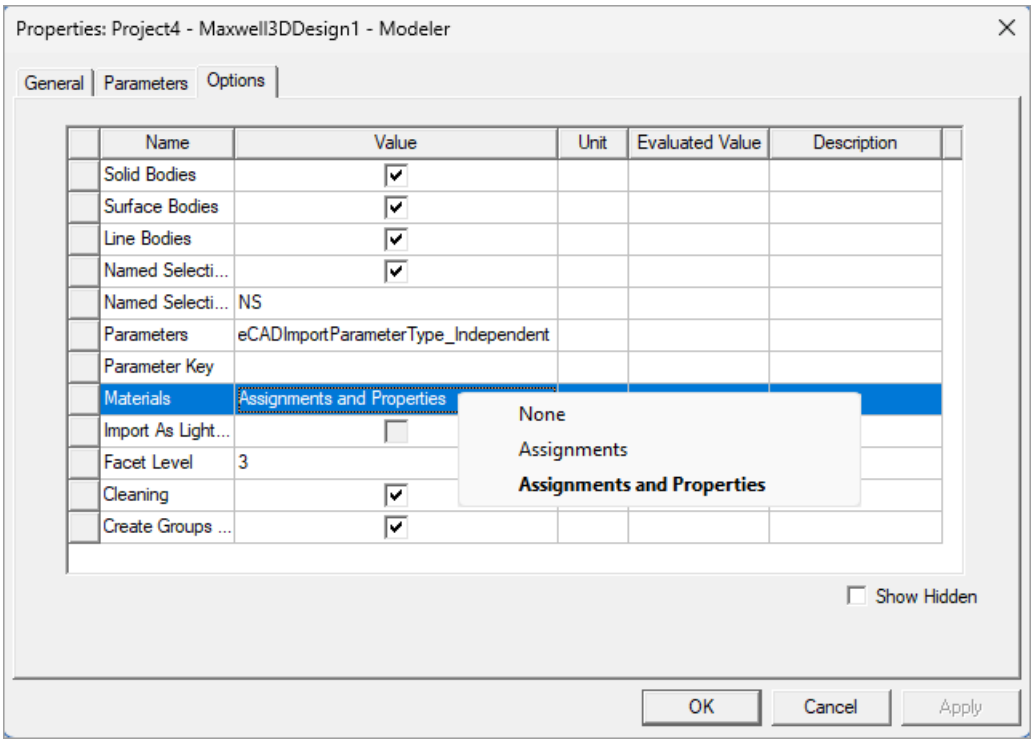
> **Note:**
>
> For a multi-part Discovery assembly to be preserved, its **Transfer Sharing** property must be set to **Group** from within Discovery.

Overriding CAD Link Settings for Individual Models

If you would like to override any of the settings made in the **CAD Link** options tab for an individual linked model, use the Properties window for the model to check, uncheck, or change a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting. An example of a Discovery model is shown below.

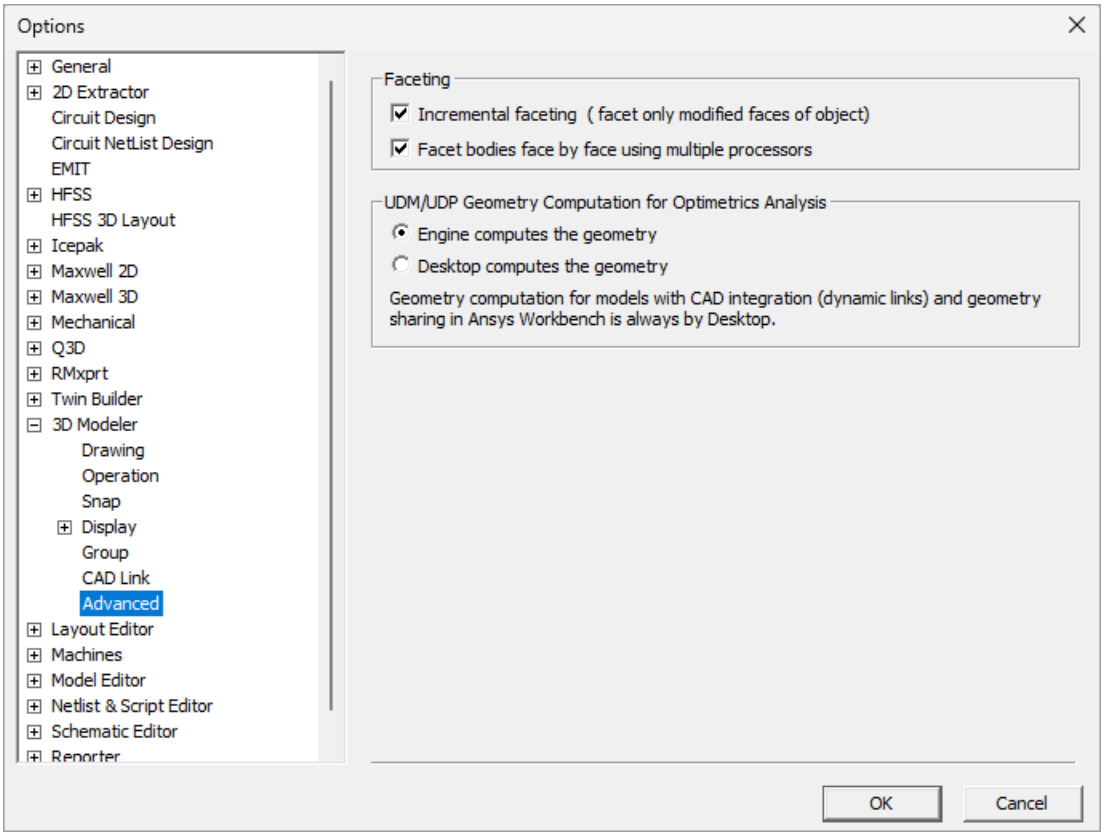


After changing parameters, refresh the link by right-clicking and selecting **Generate** or **Send Parameters and Generate**.

3D Modeler Options: Advanced Options

Under 3D Modeler options, **Advanced** options help configure faceting and UDM/UDP.

Ansys Electromagnetics Suite 2026 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.

Setting Reporter Options

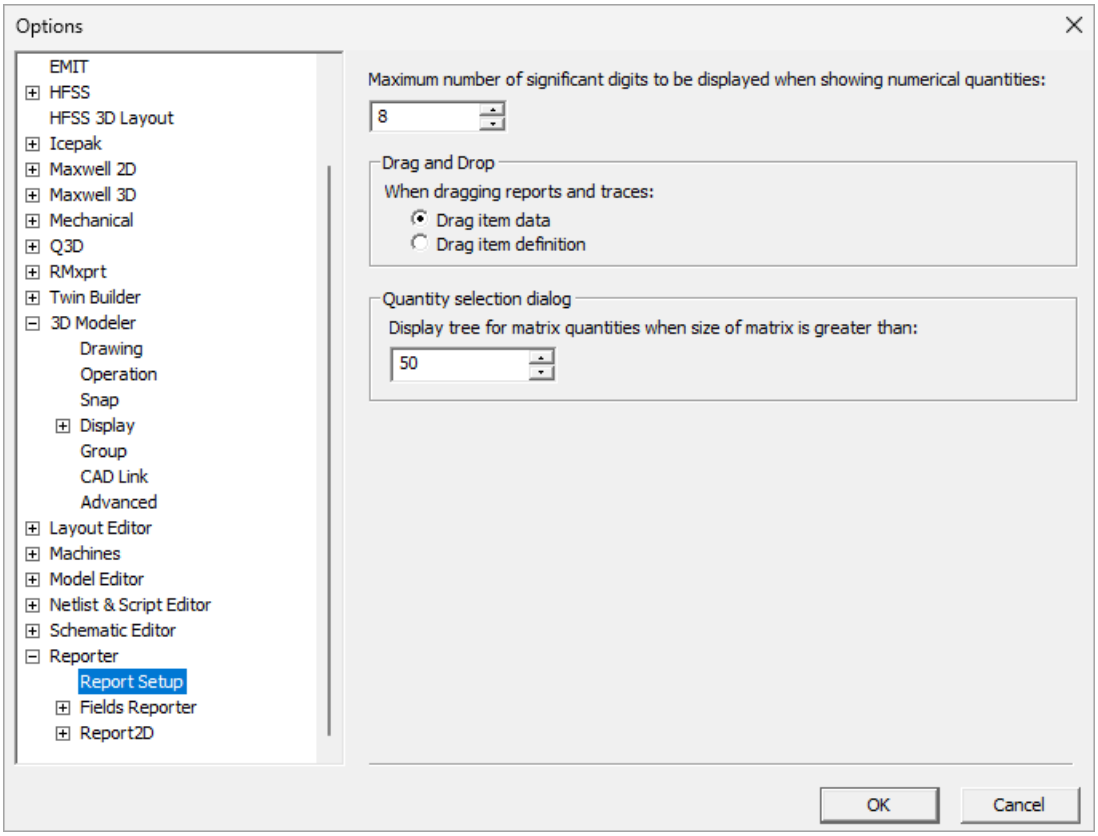
To set Reporter options in Ansys Electronics Desktop:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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. See: Using Clip Planes.

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
- Stacked
- Digital
- General
- Table

In the left pane, select a category to view those options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.
- **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**.
- **Axis Scaling** – Linear or Log. The default is linear.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.
- **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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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

- **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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

X Value Manual Format

The default number format is Auto. When number format is auto, X-value number format corresponds to X-axis number format with extra 2 precision.

- X Value Number format. Can be Auto, Decimal, Scientific, Integer.
- X Value Field Width. Default 3.
- X Value Field Precision, Default 2.

Y Value Manual Format

The default number format is Auto. When number format is auto, Y-value number format corresponds to Y-axis number format with extra 2 precision.

- Y Value Number format. Can be Auto, Decimal, Scientific, Integer.
- Y Value Field Width. Default 3.
- Y Value Field Precision, Default 2.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

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**.

Ansys Electromagnetics Suite 2026 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**.
- **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**.
- **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.

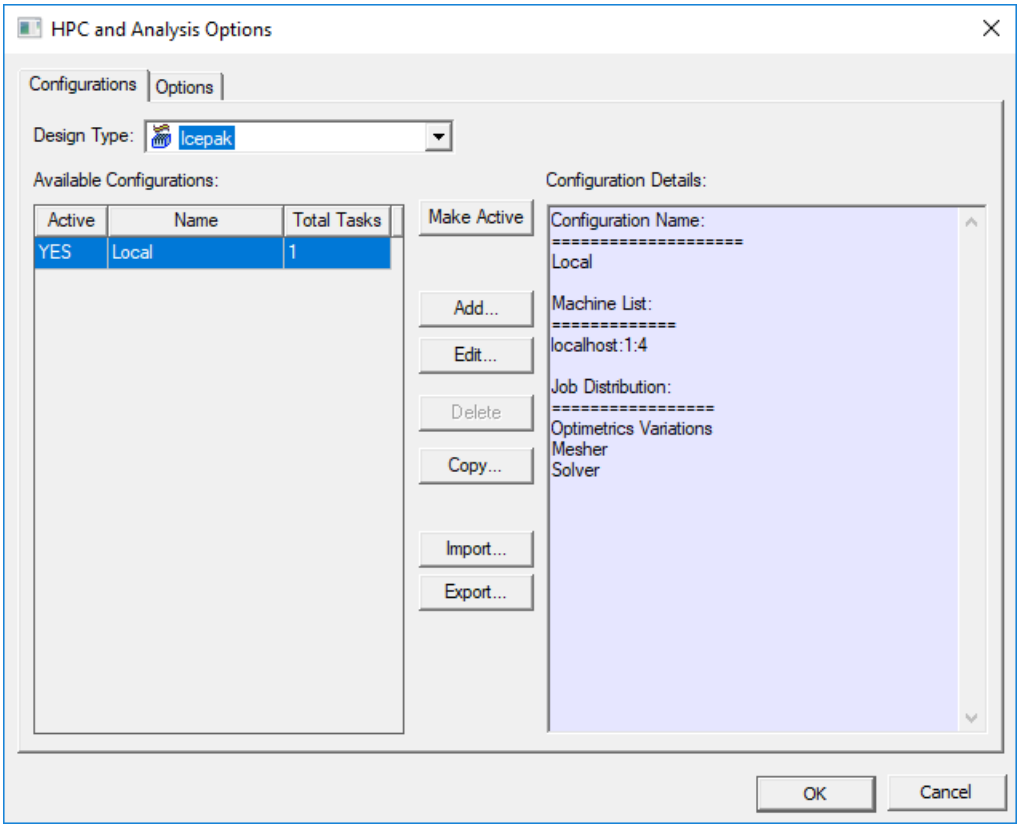
To set HPC and Analysis Options:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



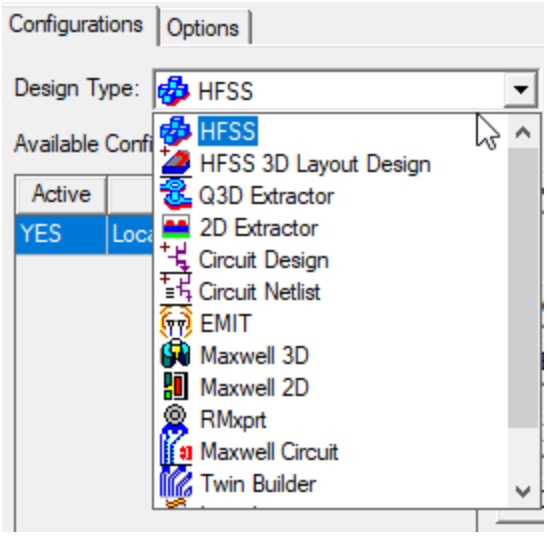
Configurations Tab

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 2026 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.

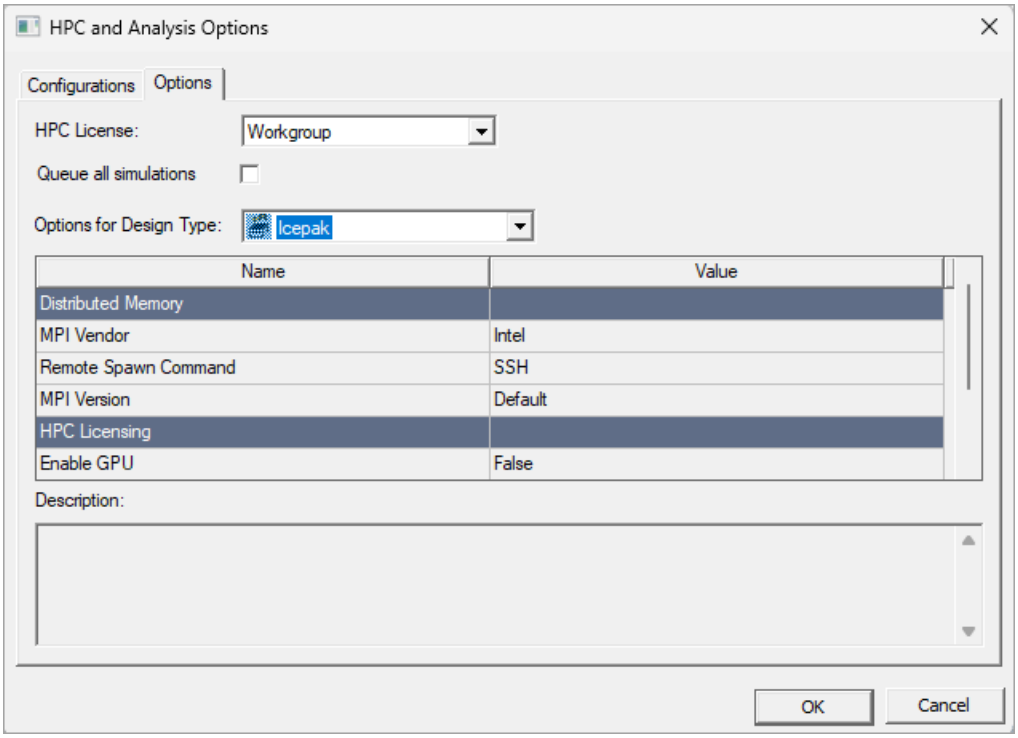
> **Note:**
>
> Importing analysis configurations always adds the imported analysis configurations to the current design type. If there is a name conflict between an imported analysis configuration and an existing analysis configuration, the imported configuration is renamed and a notification appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

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:

- 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

> **Note:**
>
> Solving with Intel MPI on a single Windows machine does not require MPI installation, but registration of the user's password with Intel MPI on the host is required if using password authentication. Solving with Microsoft MPI on a single Windows machine requires Microsoft MPI to be installed and the MS-MPI Launch Service to be started. These steps are not required for Microsoft MPI analysis on a Windows HPC cluster. Solving with Microsoft MPI on multiple Windows hosts is only supported on a Windows HPC cluster.
>
> Users running on Linux do not need to install MPI manually.

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. When set to Auto, the default is to use HPC Pack first, then Workgroup.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

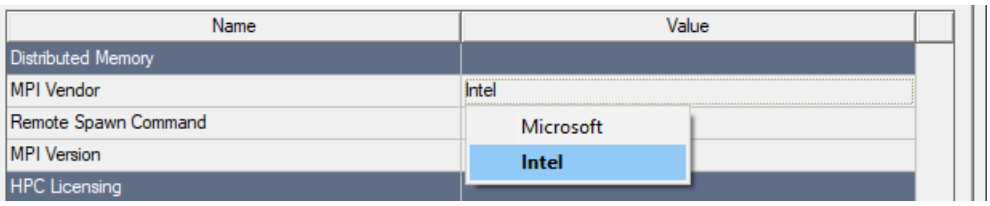
Ansys licensing supports distributed simulations when Electronics Desktop is called from other Ansys tools, such as optiSLang and Workbench. In such cases, distributed design points (variations) generally use HPC counts as described above.

> **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.

To configure options:

1. Under **Distributed Memory**, for solvers that use MPI (HFSS, HFSS 3D Layout, Icepack, Maxwell and Q3D) use the drop-down menu to select the MPI Vendor for the selected design type.



   The solvers use the industry standard Message Passing Interface (MPI) and can perform solutions that distribute memory use across machines in a cluster or network. Memory used by the MPI-enabled solver is therefore limited by the set of machines that is available rather than the shared memory available on any single machine. This allows you to simulate larger structures and to optimally reconfigure the cluster of machines for the problem at hand. For solving on a single machine, MPI is not required, nor does it provide an advantage.

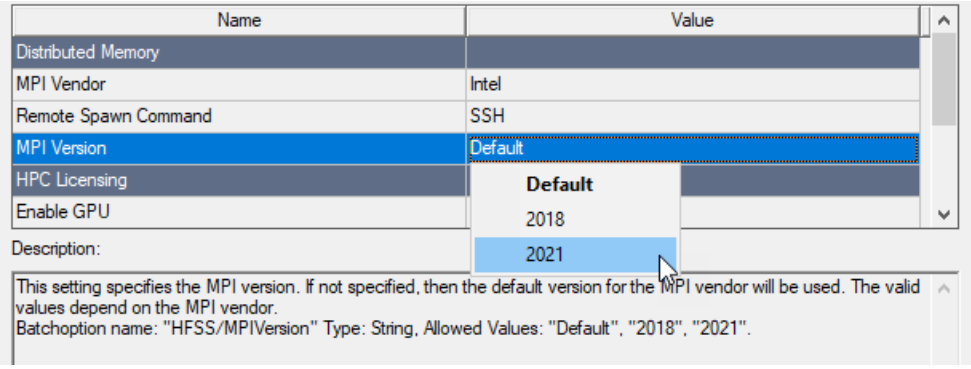
   To use the distributed memory solution you will need to install MPI software from a supported third party vendor on all the machines you intend to use.

   Depending on the MPI vendor, you may need to set passwords for authentication on the machines. Settings within each design type turn on distributed memory solutions and define the list of machines you intend to use.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify the MPI version. If not specified, the default version for the MPI vendor will be used. This setting is ignored if there is only one supported version for the selected MPI Vendor. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.
- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

InfiniBand Support for Windows

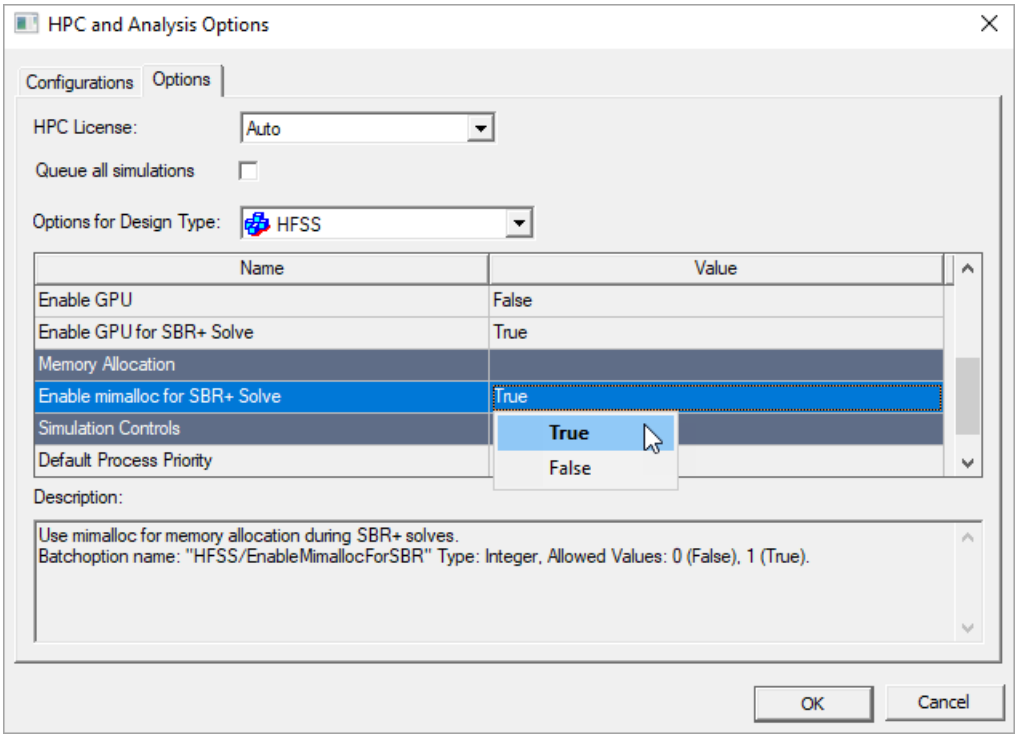
By default, MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). If you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to "eth" for the job. Also see Setting Intel MPI Interconnect for more details.

2. For Linux authentication, specify the Remote Spawn command as RSH or SSH (the default).

   For details on the requirements for GPU use, see Transient GPU Acceleration

3. **Memory Allocation**. The HFSS SBR+ solver leverages mimalloc by default on Windows to deliver better performance, especially in multi-threaded environments. On certain systems, this might cause the HFSS SBR+ not to execute. In this case, users can use the default Windows memory allocation by setting the "Enable mimalloc for SBR+ solve" option to "False"."

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. **Simulation Controls**: 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 Python scripts.

For information on editing configurations, see Editing Distributed Machine Configurations for Icepak.

Specifying the Remote Spawn Command (Linux)

An important step in using a high performance cluster is setting up authentication across machines so that the machines can be accessed without a password. By default, Icepak uses SSH authentication on Linux to spawn commands on the remote machines but also supports RSH. The selection of which to use is made on the **Options** tab of the **Tools > Options > HPC and Analysis**.

SSH

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will need to set up passwordless access to use Icepak on a Linux cluster with SSH or RSH. In general, for SSH, this is accomplished by:

1. Verifying that you have working SSH servers and clients on your machines.
2. Verifying that the server will accept passwordless logins. You may need to edit the /etc/ssh/ssh_d file to allow RSAAuthentication and PubkeyAuthentication.
3. Generating keys on the client system using the ssh-keygen program. Do not use a passphrase so that you can access the machine without a password.
4. Copying the public key generated in step 1 from the ~/.ssh directory to the server. The easiest way to transfer the keys is to use the ssh-copy-id program. Alternately, you can use any file transfer utility. If the server already has a list of existing keys for other clients add the new public key to the list.
5. Testing the connection. Log in to the client machine using the username that you used to create the identity keys. Open a new shell terminal and attempt to open an SSH login session. For example, type `ssh 192.168.0.4` (where the IP address is the address of the machine you are attempting to connect to). The server should allow you to log in without requesting a password.

Consult the documentation for your machines and network for detailed instructions.

RSH

If you choose to use RSH, you will need to make sure RSH is installed on all the machines and set up the machines so that you are not prompted for a password. There are different ways to set up passwordless RSH, so be sure to consult the documentation for your machines and network for detailed instructions.

Machine access using RSH without a password is often set up by editing the /etc/hosts.equiv file and adding entries for the hosts you would like to use without a password. This file lists hosts and users that are granted "trusted" access to the system.

If you look at the /etc/hosts.equiv file you should have something similar to the following:

Contents of the /etc/hosts.equiv file:

```
job1.n1.com
job2.n1.com
job3.n1.com
```

The machines job1, job2 and job3 can connect without a password. You may also need to verify that the files /etc/hosts.allow and /etc/hosts.deny are empty. Consult your local documentation for detailed instructions and troubleshooting suggestions.

Scheduler

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel".

When submitting a job using the AnsysEM job submission GUI, the Remote Spawn Command for an analysis may be specified using the batchoption with pathname 'DesignType/RemoteSpawnCommand', where DesignType is the Design Type to analyze. The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel". To specify the value 'Scheduler' for this option for a scheduler job, the Remote Spawn Command must be specified using the 'DesignType/RemoteSpawnCommand' batchoption in the product command line when the product is launched. In addition, the 'DesignType/MPIVendor' batchoption must be specified with value "Intel" in the product command line when the product is launched. For interactive scheduler jobs, the Remote Spawn Command and the MPI Vendor may be specified with batchoptions or as design type options in the **HPC and Analysis Options** dialog box.

Setting MPI Version

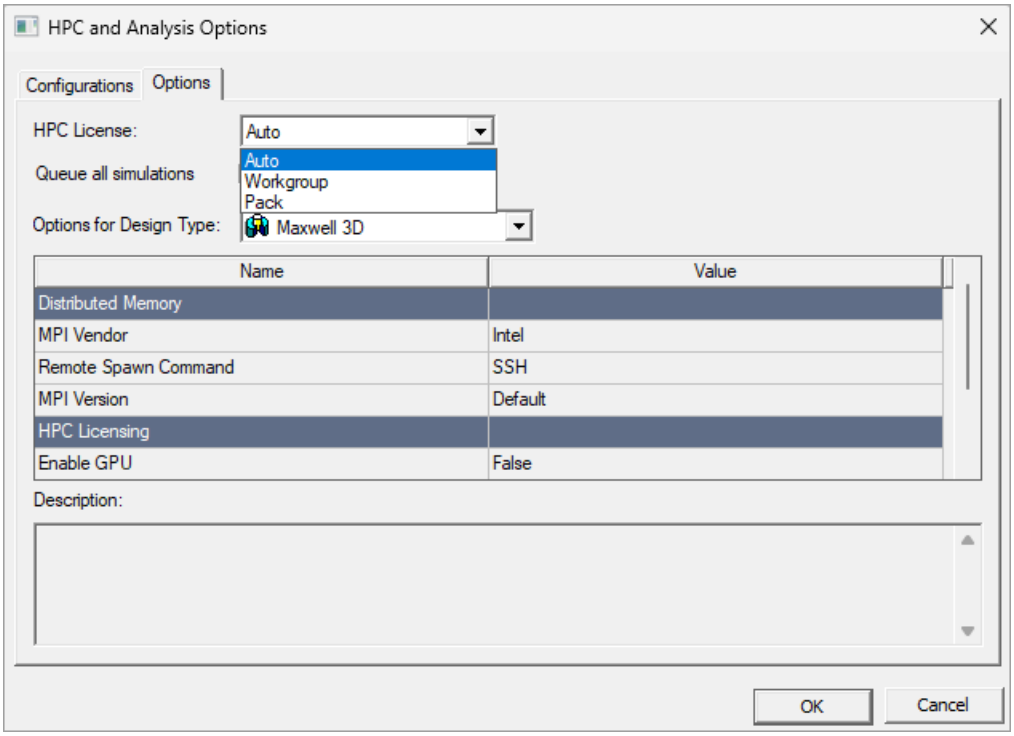
When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

Licensing Settings Tool

HPC licensing includes a choice called “Auto”, along with Workgroup and Pack. The “Auto” choice is available in the HPC License combo box 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



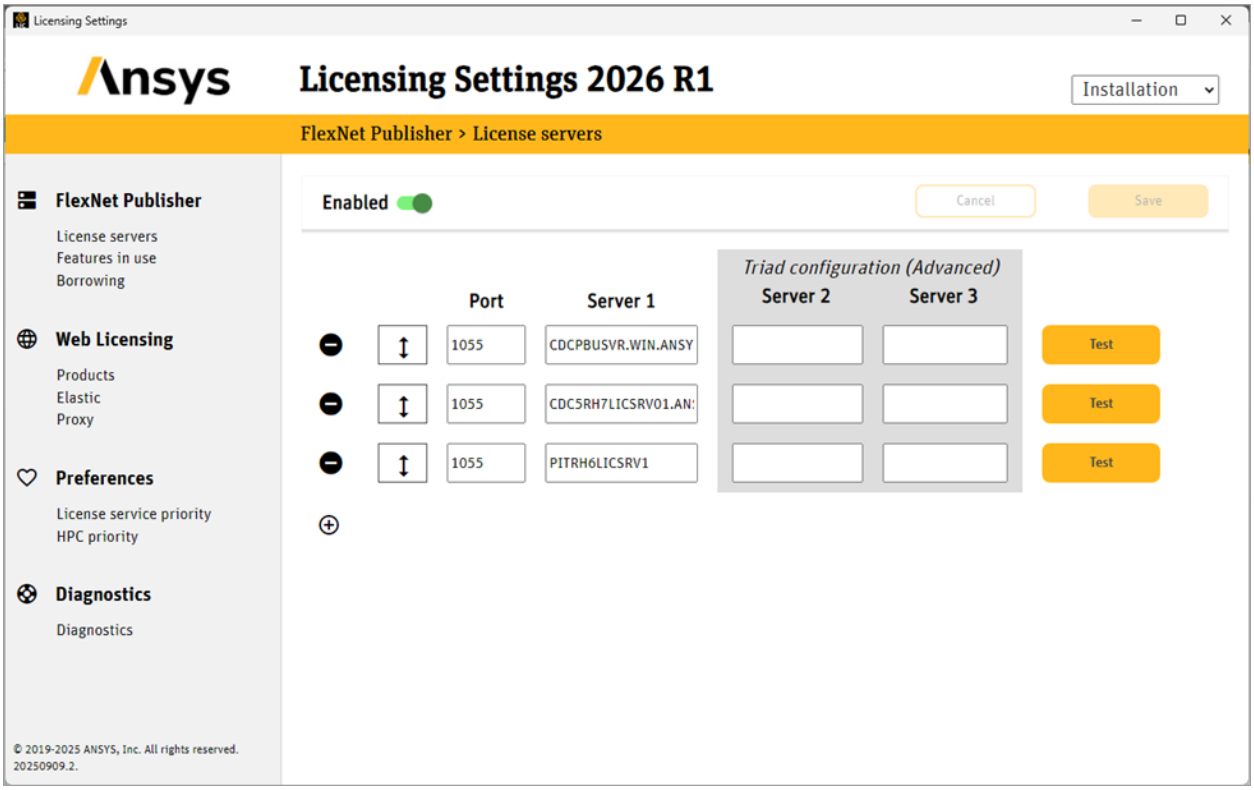
Auto means delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool, which is a separate application installed along with Electronics Desktop, accessible from the **Tools** menu. The HPC settings specified by the Licensing Settings tool are shared by other Ansys products. 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.

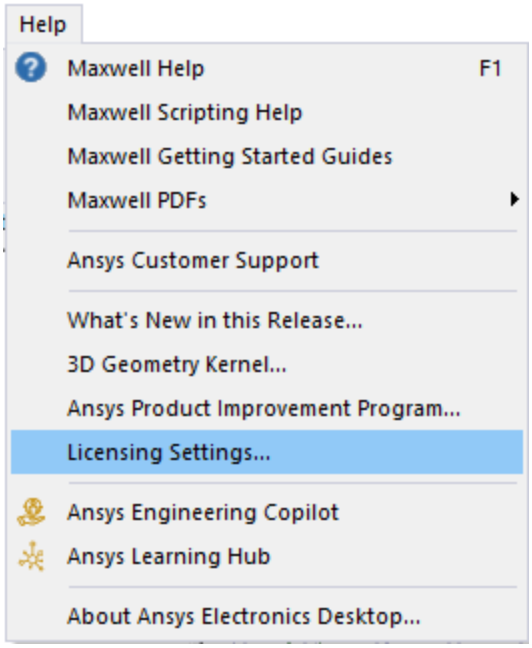
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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\v261\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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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\<*Version*>\AnsysEM\config	*<Installation_Directory>*/ansys_inc/<*Version*>/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 ElectronicsDesktop2026.1)
- **%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 lower-case letter "v" followed by last two digits of the product version's year and the minor release number, without a decimal point (such as v261)

Ansys Electromagnetics Suite 2026 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 the current software version, 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\ ElectronicsDesktop2026.1\config	/home/jsmith/Ansoft/ ElectronicsDesktop2026.1/config
host independent user options	user	user.XML		
host dependent default options	install_machine	host123.XML	"C:\Program Files\ANSYS Inc\v261\ AnsysEM\config"	/opt/ansys_inc/v261 /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 exit the application, the file at the "user_machine" level is created (<*hostname*>_user.XML). The other files are only created if you do one of the following:

- Use the Choose Registry Type command to switch to a machine-independent registry (user.XML), or
- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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, ElectronicsDesktop2026.1. 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\v261\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.

Ansys Electromagnetics Suite 2026 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 -Get Command

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, ElectronicsDesktop2026.1. 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\v261\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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example:

```
UpdateRegistry -GetKeys [ <pattern> ] -
ProductName <name> [ -Case ]
```

Required:

<name>	The application or product name and version. For example, ElectronicsDesktop2026.1. 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\v261\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, ElectronicsDesktop2026.1. 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\v261\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 2026 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 2026 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 2026 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 ElectronicsDesktop2026.1
```

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 ANSYSElectronicsDesktop2026.1
-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 ElectronicsDesktop2026.1
-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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2026.1
-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 ElectronicsDesktop2026.1
-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.

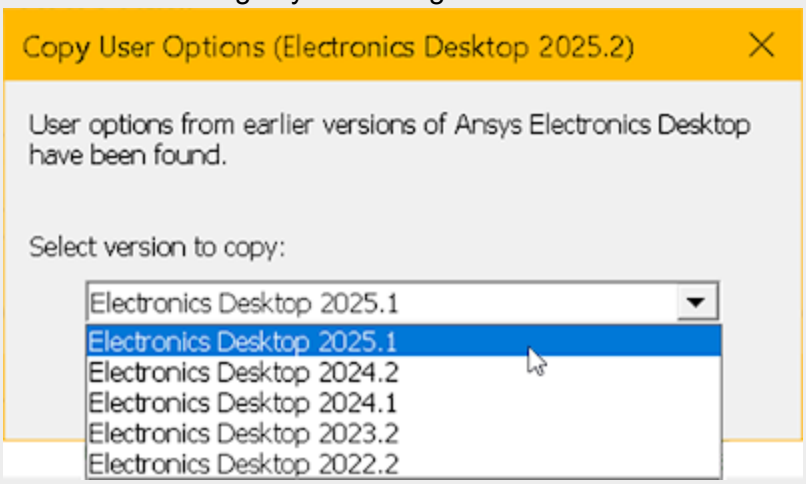
When you change an option's value using the Ansys Electronics Desktop UI, the new value is stored in the currently active user options registry. You can use the UpdateRegistry tool to add, modify, or remove settings. However, you cannot use the Desktop UI to *remove* settings from the registry. *You must use the UpdateRegistry tool to do that.*

If you have not already created a user options configuration file (either host-specific or host-independent), before launching the Ansys Electronics Desktop application for the first time, all settings will come from the *host-specific default* options configuration file or the *installation default options* configuration file. In a host-specific user options configuration file, any settings applicable to a different host from the current one will not be carried over to the new host. This behavior may be inconvenient if the user has preferred option settings that differ from the default settings, which apply to all users, especially if the user runs the application on a number of different hosts. In this case, the user may choose to use a host-independent registry and set the option values in this type of options file. See Understanding Registry Tools for more information. Host-independent option values can be used on all new hosts, overriding any values set by the administrator to apply to all users. Changes made in the UI can affect the user's host-specific or host-independent registry, depending on the currently specified registry type.

Ansys Electromagnetics Suite 2026 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 program detects earlier major and minor versions of same application on the same host. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays the earlier versions you can select, from which the registry will be migrated.



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 ElectronicsDesktop2026.1
-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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2026.1 -
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 ElectronicsDesktop2026.1 -
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 ElectronicsDesktop2026.1
-RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
-RegistryValue 250 -RegistryLevel user
```

Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml

```
"C:\Program Files\ANSYS Inc\v261\AnsysEM">UpdateRegistry -Set -
ProductName ElectronicsDesktop2026.1 -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\v261\AnsysEM\">UpdateRegistry -Set -
ProductName ElectronicsDesktop2026.1 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "n"
```


Ansys Electromagnetics Suite 2026 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:

1. Navigate to **Tools** > **Options** > **General Options**.
2. In the tree at the left side of the dialog box, expand the **General** branch and select **Directories**.
3. 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.

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

Ansys Electromagnetics Suite 2026 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-independent, user-specific temporary directory	user	default.cfg	"<*InstallationDirectory*>\ANSYS Inc\v261\AnsysEM\config"	<*InstallationDirectory*>/ansys_inc/v261/AnsysEM/config
Host-dependent, default temporary directory	install_machine	<*hostname*>.cfg		
Installation default temporary directory	install	default.cfg		

Ansys Electromagnetics Suite 2026 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 ElectronicsDesktop2026.1)
- **%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 lower-case letter "v" followed by last two digits of the product version's year and the minor release number, without a decimal point (such as v261)

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 2026 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 2026 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 2026 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 **integer** 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). Therefore, 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 processors for 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 2026 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.

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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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
```

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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'
'HFSS/SolverOptions/NumProcessors'=4
'HFSS/SolverOptions/NumProcessorsDistrib'=4"
\\host123\projects\testproject.adsn
```

Note that the batchoptions pathnames 'HFSS/SolverOptions/NumProcessors' and 'HFSS/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 2026 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 comprising 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:

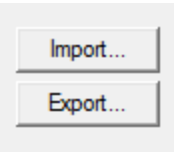
  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

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 2026 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\v261\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 **26.1**). Also, substitute the desired *text_file_path* (such as **%UserProfile%\Desktop**).

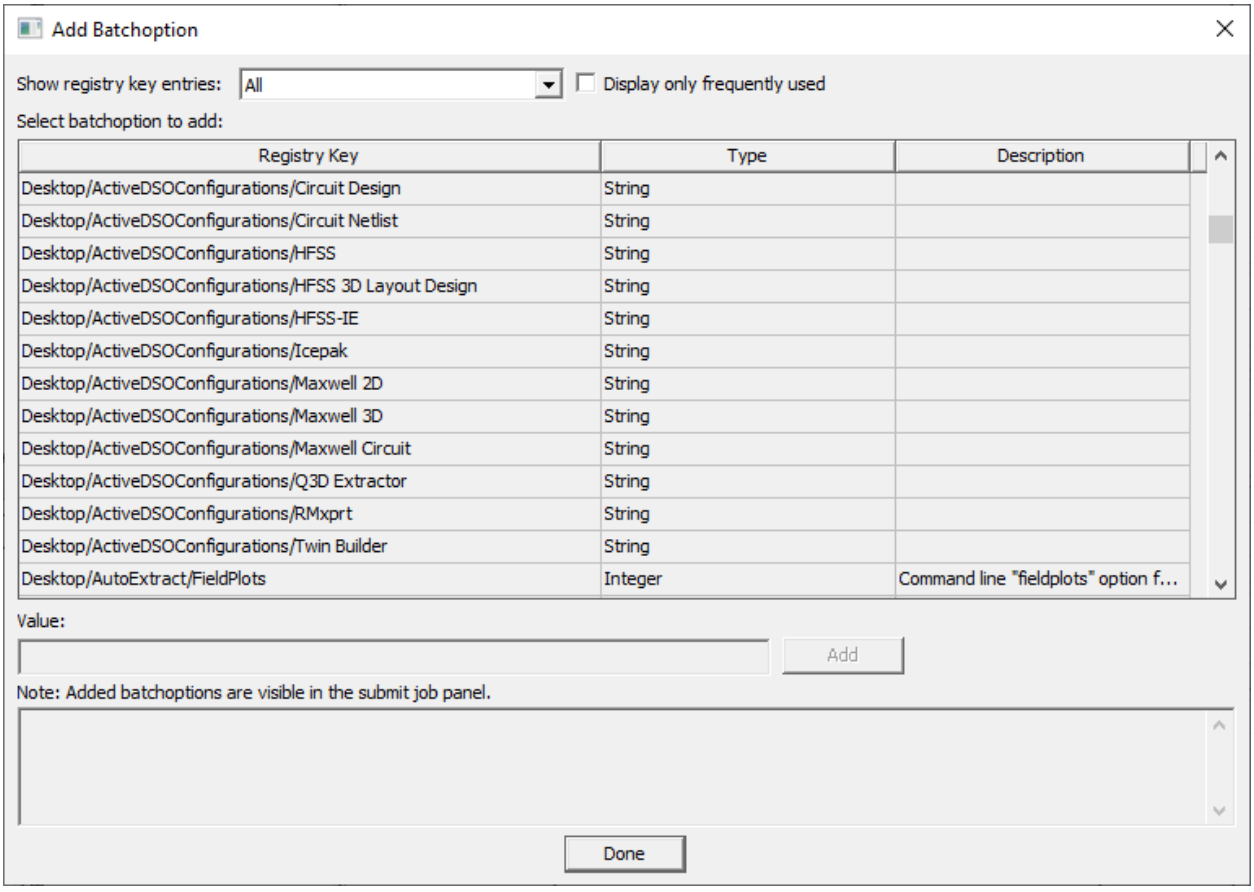
   The resulting **Batchoptions.txt** file will contain a nearly complete list of available options.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

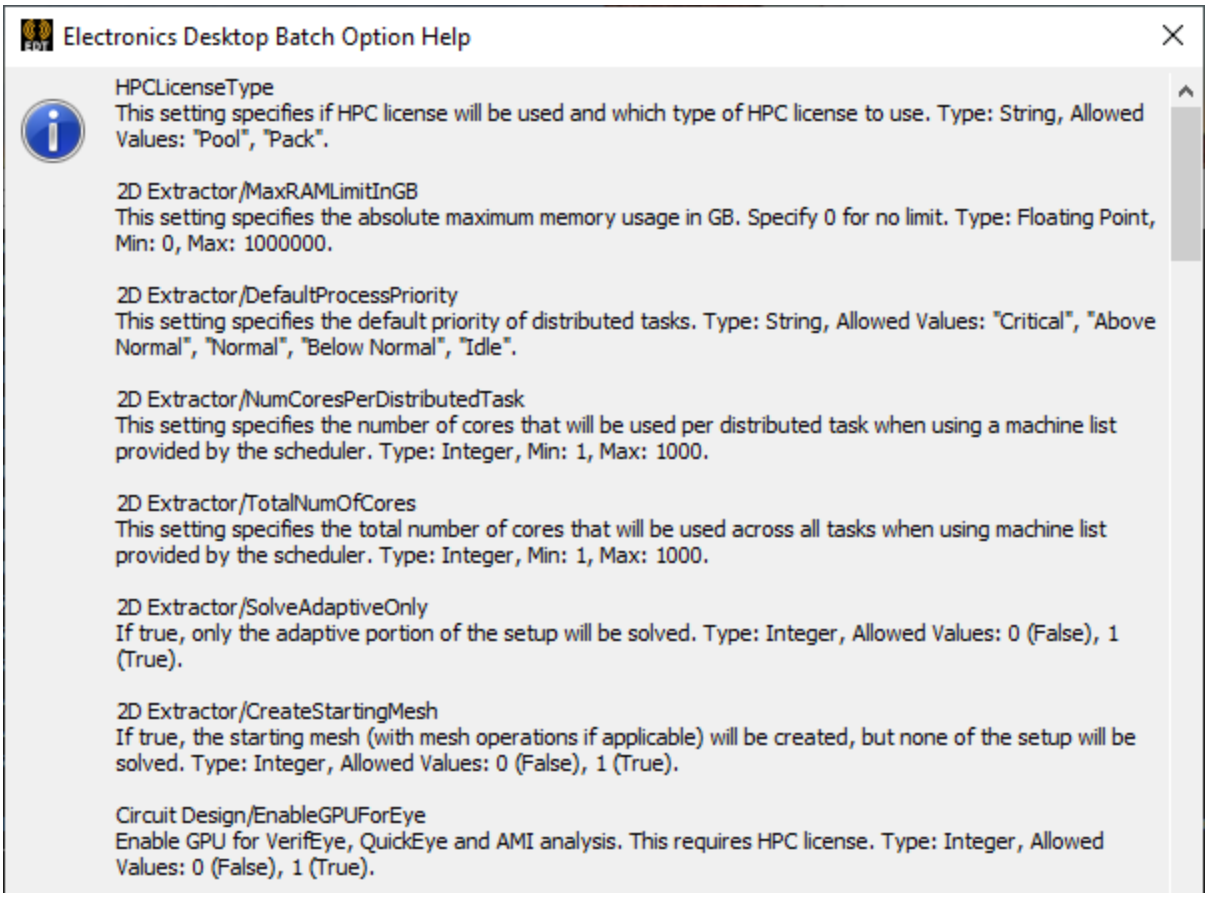
> **Note:**
>
> For special product-specific options that are neither available from the GUI nor listed by the UpdateRegistry tool, see Special Batch Options. Even though the -GetKeys switch does not list these special options, you can still use the UpdateRegistry tool to set them.

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 2026 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 2026 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

Design Settings

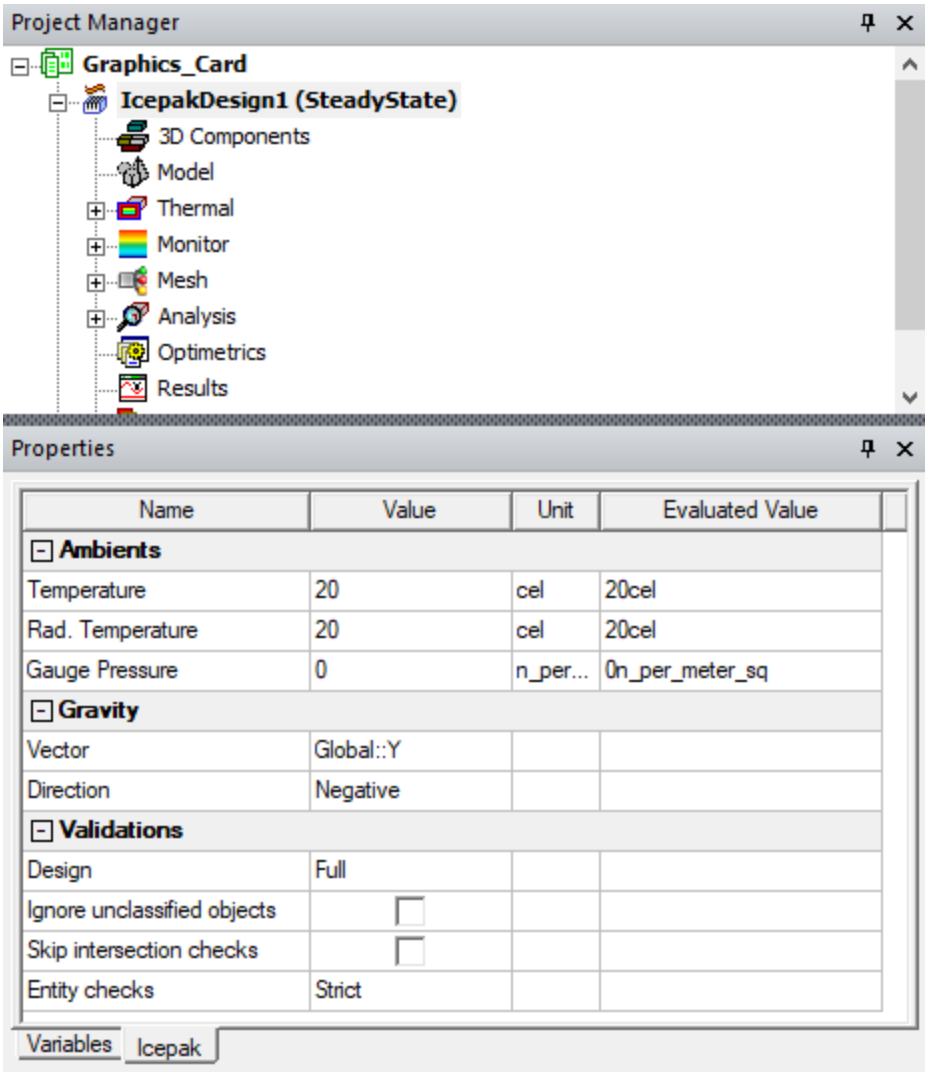
The tabs and settings within the *Design Settings* dialog box are dependent on the design type, and there is a product-specific topic for each applicable design type. Please click the appropriate link below for the product of interest:

- HFSS
- HFSS Transient
- HFSS 3D Layout
- Icepak
- IcepakFEA
- Q3D
- 2D Extractor

Design Settings for Icepak

The **Icepak** > **Design Settings** command displays the **Icepak Design Settings** dialog with tabs for **Ambient Conditions**, **Gravity**, and **Validations**. Selecting a current design in the **Project** tree also displays the Design Settings in **Properties** window under the solver tab.

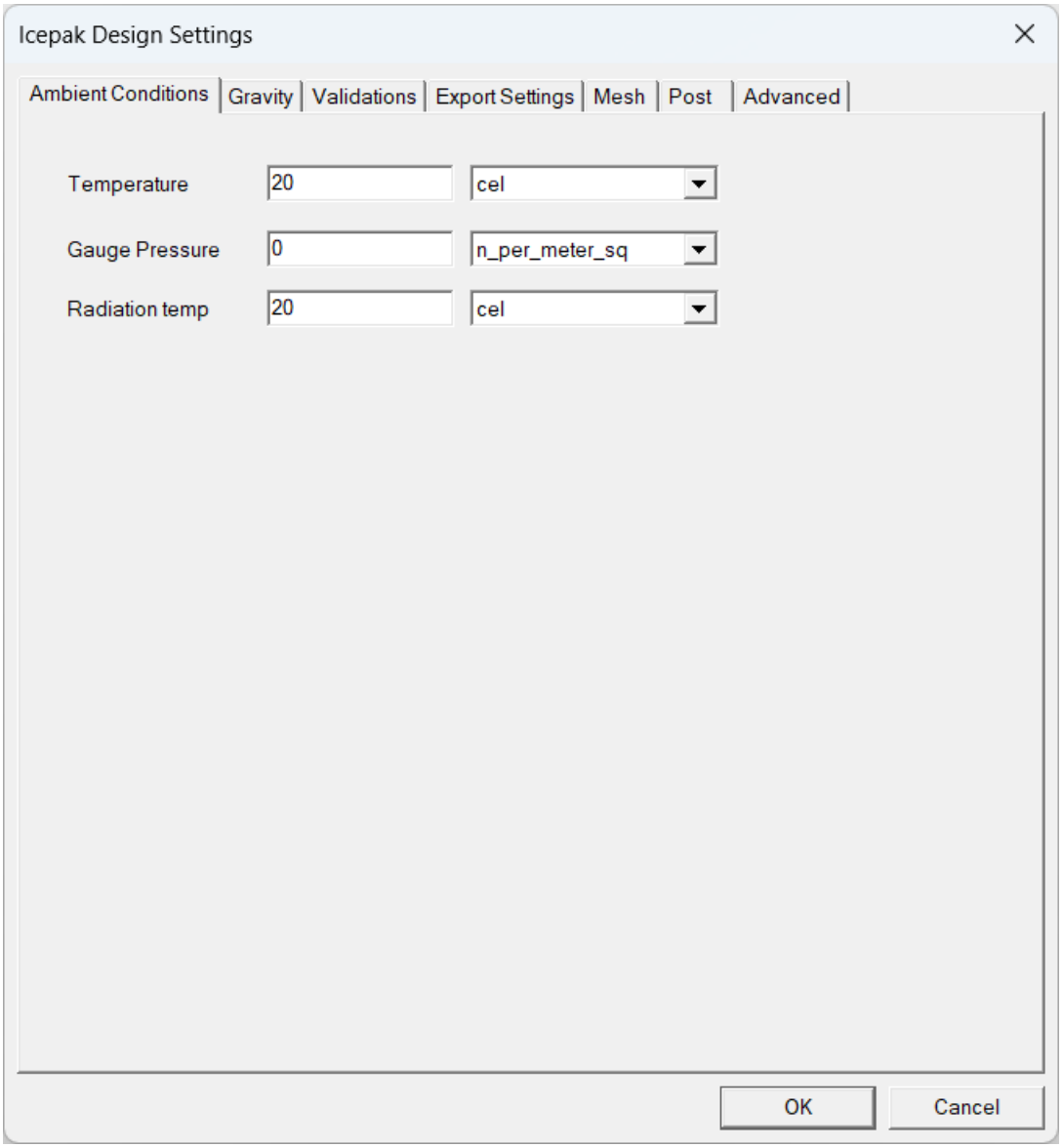
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ambient Conditions Tab

The **Ambient Conditions** tab allows you to specify the default ambient values for **Temperature**, **Gauge Pressure**, and **Radiation temperature**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Gravity Tab

When including the effect of gravity for a natural convection simulation, select the appropriate coordinate direction from the **Gravity Vector** drop-down menu. Also, select **Positive** or **Negative** to select in which direction to apply the gravitational acceleration. The gravitational acceleration in Ansys Icepak is 9.80665 m/s^2 in the selected coordinate direction.

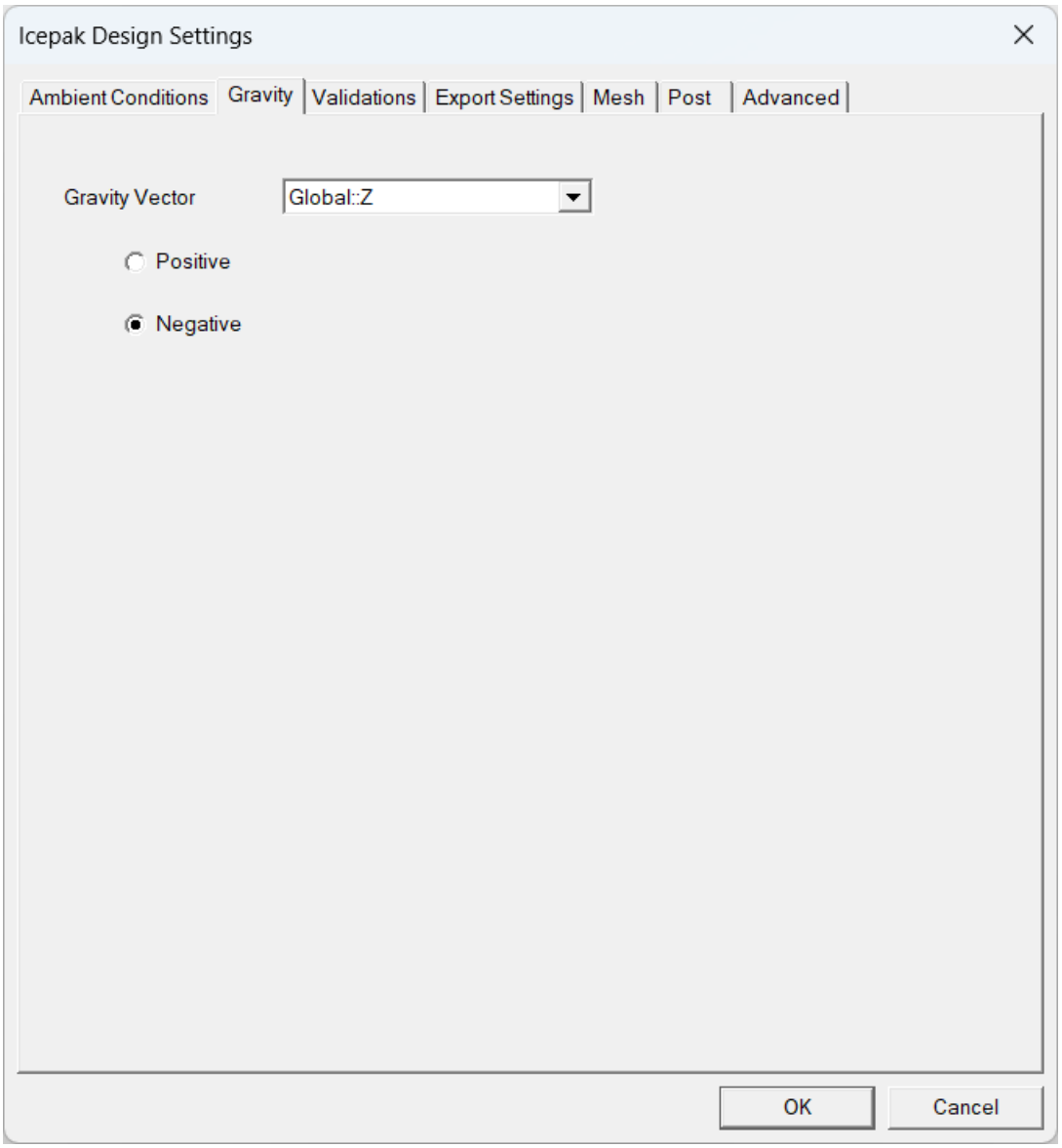
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

To include the effect of gravity in the simulation for natural convection models, select **Include Gravity**in the Icepak Solve Setup Dialog.

Note:

The **Gravity** tab is displayed only when the **Solution Type** is set as **Temperature and Flow**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Validations Tab

The **Validation** tab offers choices for model validation and Icepak validations to control the extent of validations performed, and therefore the time involved. You can adjust the degree to which the software checks a model for faults that could jeopardize mesh accuracy. There are four levels of model validation that a user can specify for a given design: **Warning Only**, **Basic**, **Strict**, and **None**. Note that this setting affects only the "3D Model" stage of a design validation. The default **Entity Check Level** is **Basic**.

- The **Warning Only** entity check setting allows all models to pass 3D Model validation regardless of any faults that are found. These faults are posted in the message window as warnings.
- The **Basic** entity check setting allows most models to pass 3D Model validation. Some faults are flagged as model errors, thereby prohibiting a design from proceeding to the meshing stage of an analysis. You must correct such errors before attempting to analyze the design under the Basic setting, or you must change the entity check level to Warning Only.
- The **Strict** entity check setting enforces a tighter tolerance for model faults than the Warning Only and Basic settings. All model faults that are 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, or you must change the entity check level to Basic or Warning Only.
- The **None** setting disables model validation.

The following two additional model validation options (check boxes) are available:

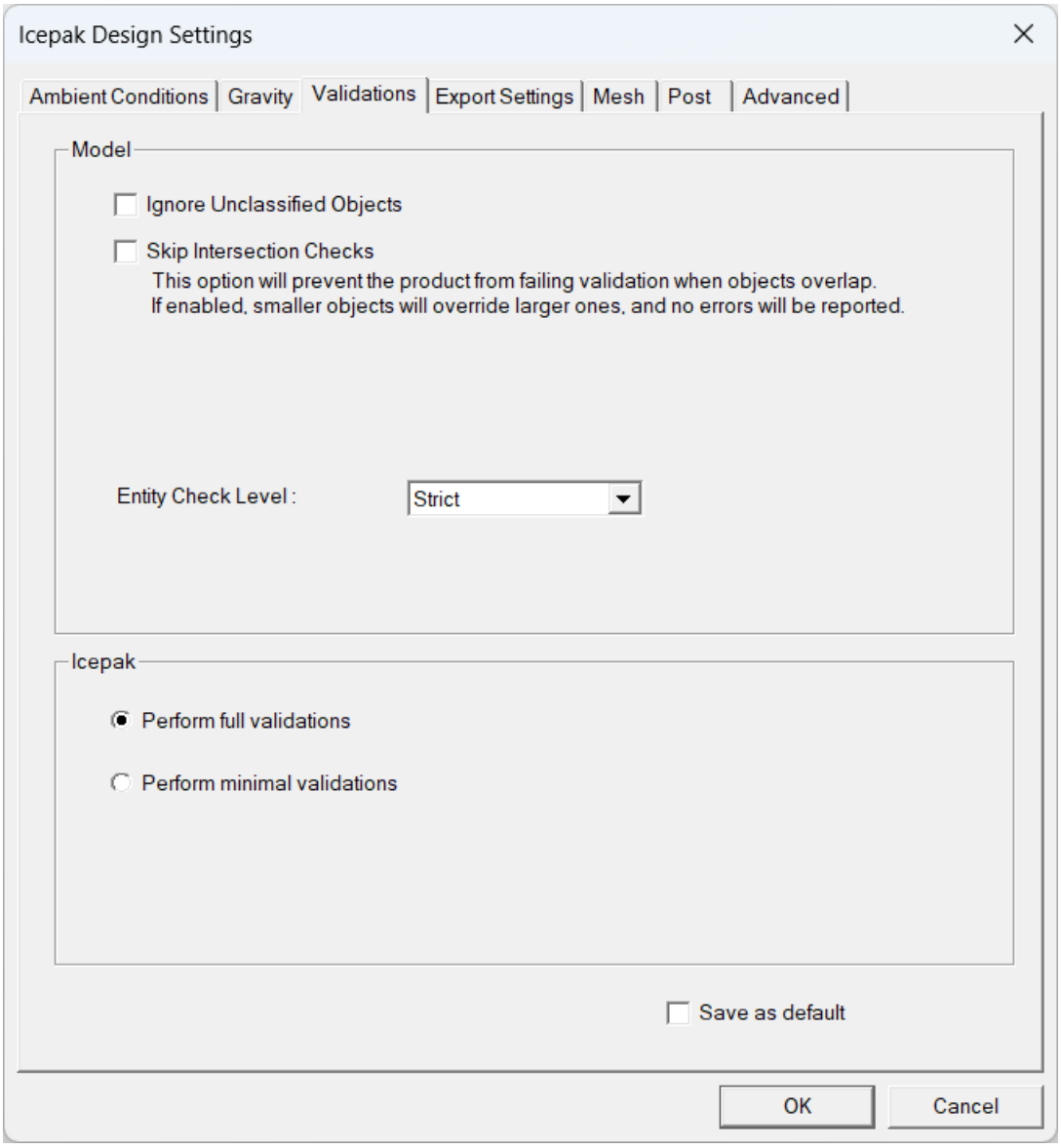
- **Ignore Unclassified Objects:** Generally, all objects in the 3D Modeler's History Tree are classified as solid, sheet, line, or point objects. However, in some cases, a drawing operation may produce null geometry, or it may fail and produce invalid geometry. For example, subtracting a tool object 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 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:** Do not check for object interference (that is, overlapping object volumes). In Icepak, you should prioritize objects so that the solver knows which object takes precedence when objects overlap each other.

Design validation settings allow you to choose either:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Perform full validations** (the default setting) – All design validations are performed.
- **Perform minimal validations** – All design validations except boundary overlap validation are performed.



Export Settings Tab

The **Export Settings** tab allows you to configure export options for Icepak designs.

Under **Export Monitor data**, enable **Export monitor data after completing simulation** to automatically save monitor data in a selected location after the successful completion of a simulation. The data exported includes:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Solution residuals
- Monitor points
- Network temperatures

By default, the save location is in the project directory. To save the monitor data in different location, select **Override** and the browse button to open the **Choose Directory** dialog box.

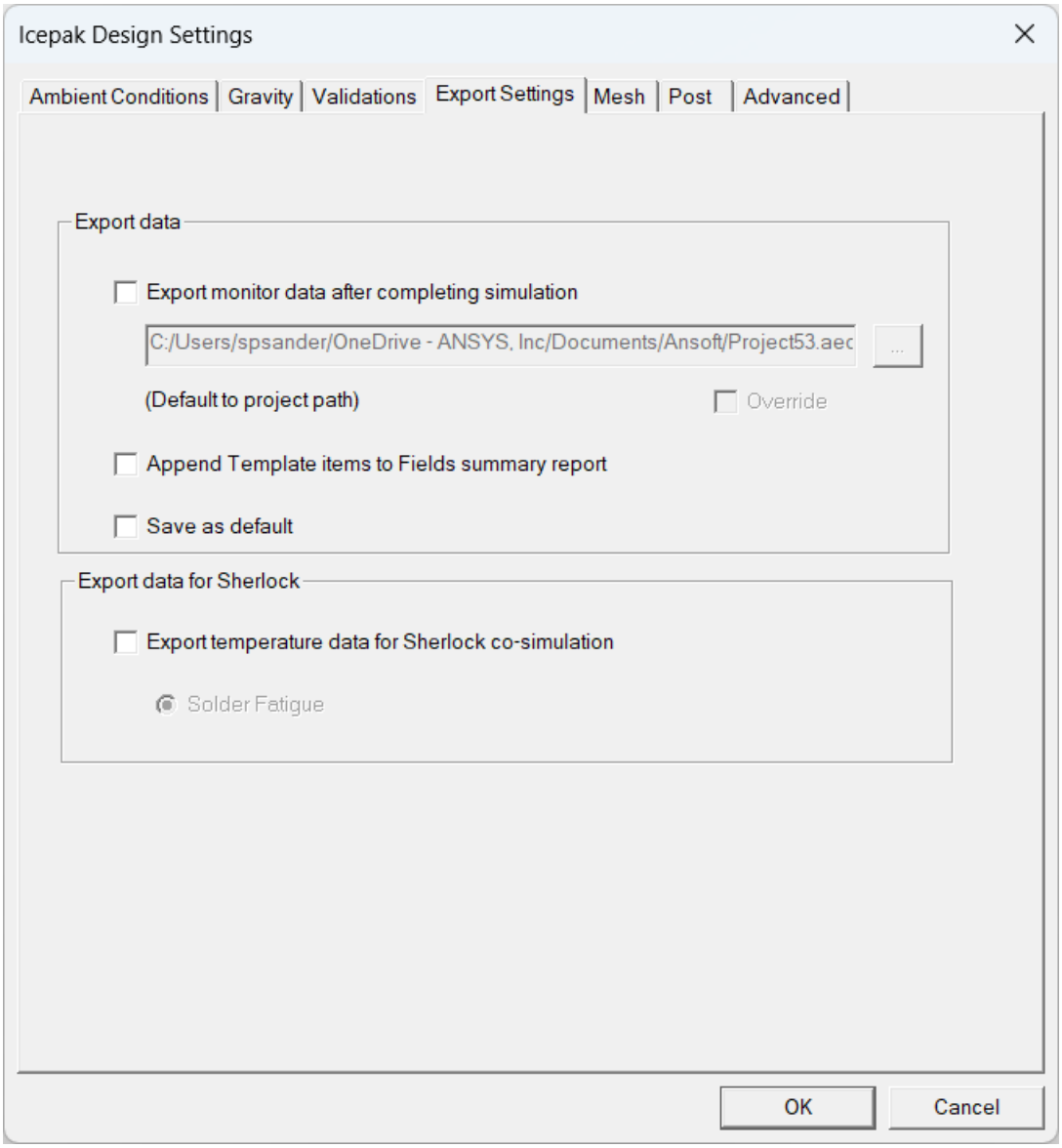
> **Note:** RedHawk CTM temperature data files are automatically generated when a design contains a CTM component. The files are saved in the project directory unless **Override** is enabled and a different directory is specified.

Under **Export data for Sherlock**, enable **Export temperature data for Sherlock co-simulation** to automatically generate temperature data files for use in Ansys Sherlock. By default, the directory "*projectname*.aedtexport" is created in the project directory (the location where the .aedt project file and folder are saved). The file path is displayed in the **Message Manager**.

- Select **Solder Fatigue** to generate a thermal map (.tmap) file consisting of temperature values for the surface mesh elements on the top and bottom faces of the PCB.

  > **Note:**
  >
  > For **Solder Fatigue**, the design must contain at least one PCB.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Tab

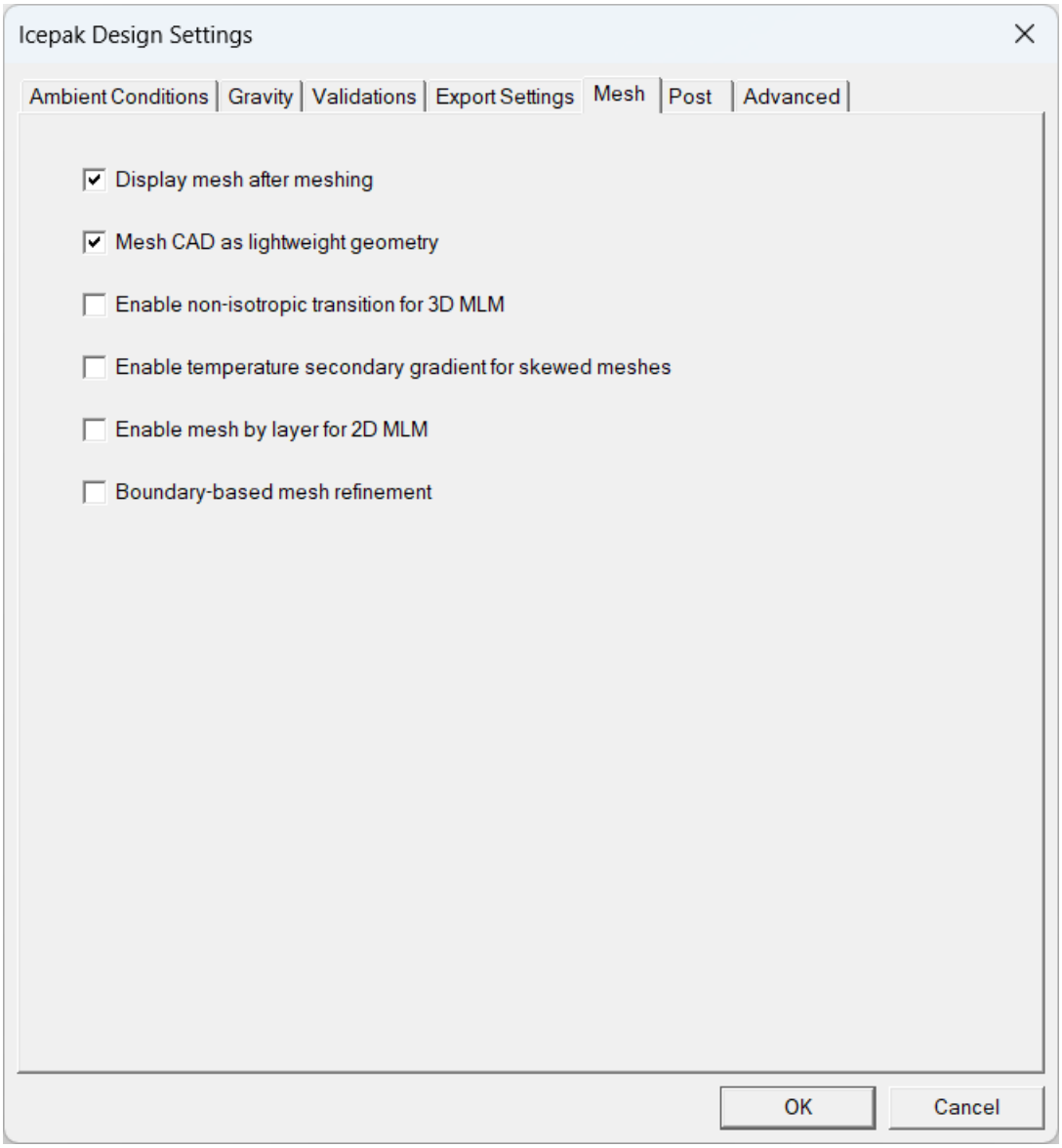
The **Mesh** tab allows you to enable meshing options for Icepak designs.

- Select **Display mesh after meshing** to automatically open the **Mesh Viewer** and display the mesh after generating a mesh.
- Select **Mesh CAD as lightweight geometry** to simplify the meshing of imported CAD geometry, thereby reducing the solution time. While matching the solution performance of lightweight geometry, this option retains the full geometry information such that users can make use of the individual CAD faces to assign specific boundary conditions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Enable non-isotropic transition for 3D MLM** is used to determine how the 3D cut cell mesh generation method subdivides cells near objects. By default, 3D cut cell uses isotropic subdivision which split a hexahedron cell into 8 children hexahedra cells. If **Enable non-isotropic transition for 3D MLM** is enabled, the subdivision is done anisotropically. The non-isotropic subdivision will create *N* number of mixed cell types (hexahedra, tetrahedra, prisms and pyramids). The non-isotropic subdivision will lead to a faster meshing time but will contain more cells than an isotropic subdivided mesh.
- **Enable temperature secondary gradient for skewed meshes** is used to enforce temperature secondary gradient (in advanced solver settings) even in the presence of highly skewed meshes. Temperature secondary gradient might improve accuracy but has a stricter quality requirement on meshes.
- **Enable mesh by layer for 2D MLM** is used to create 2D multi-level mesh according to different geometry layers.
- **Boundary-based mesh refinement** is used to insert a layer of prism mesh into Joule heating blocks to improve numerical stability. When user-specified advanced mesh settings are enabled, boundary-based mesh refinement also ensures that there are at least three layers of mesh between heat sink fins.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Post Tab

The **Post** tab allows you to enable post-simulation options for Icepak designs.

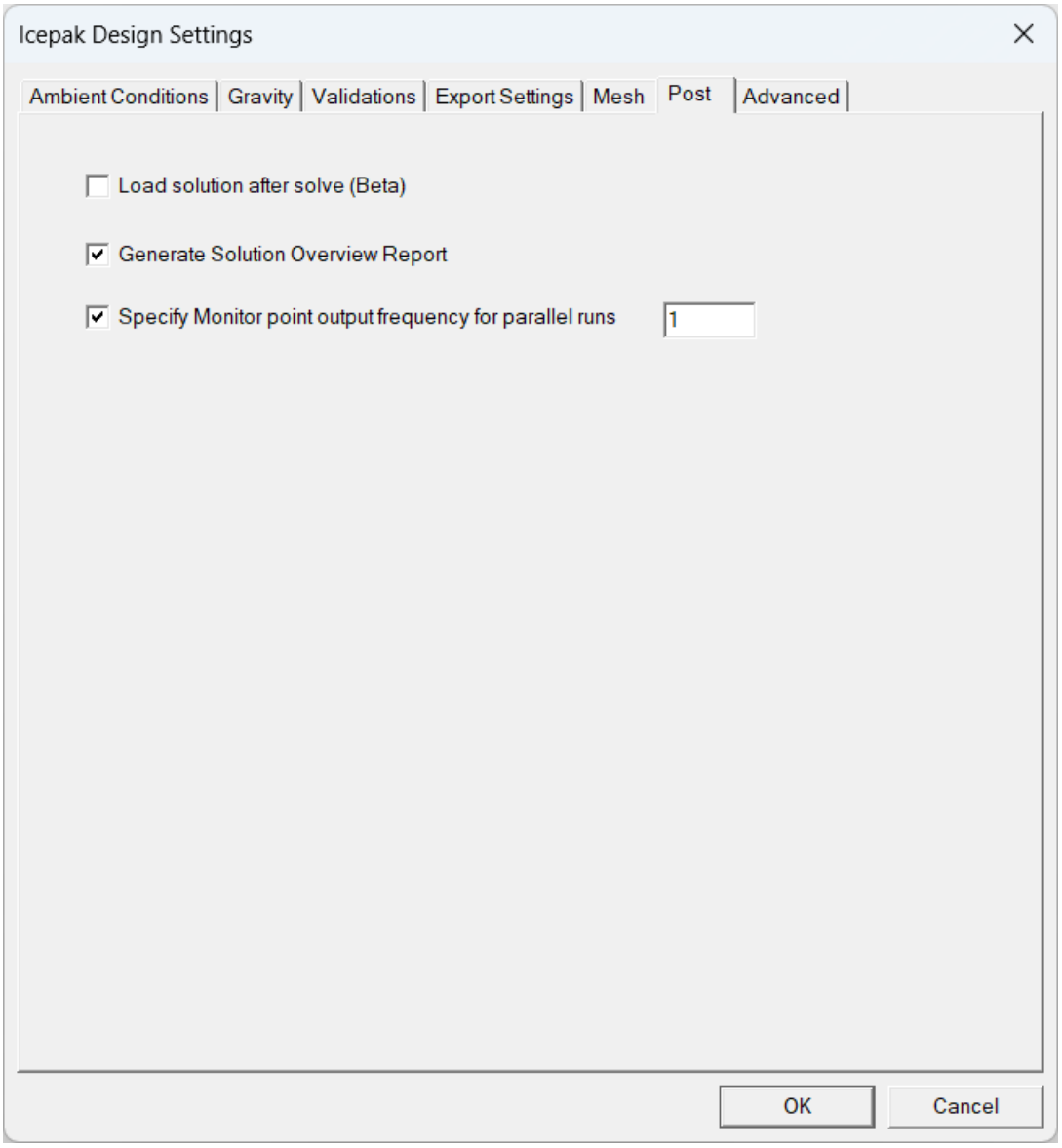
- Enable **Load solution after solve** (beta) to automatically load the solution after the simulation completes. This options allows you to quickly post-process the simulation results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When **Load solution after solve** is enabled for parametric trials, Electronics Desktop caches results data for all variations, which may result in an accumulation of data and poor performance if your machine does not have enough memory.

- Enable **Generate Solution Overview Report** to create a solution overview report after the simulation completes. In the **Project Manager**, right-click on the solve setup and select **Solution Overview** to review the report.
- Enable **Specify Monitor point output frequency for parallel runs** and enter a value to define how often monitor point data should be published during a parallel solution process.

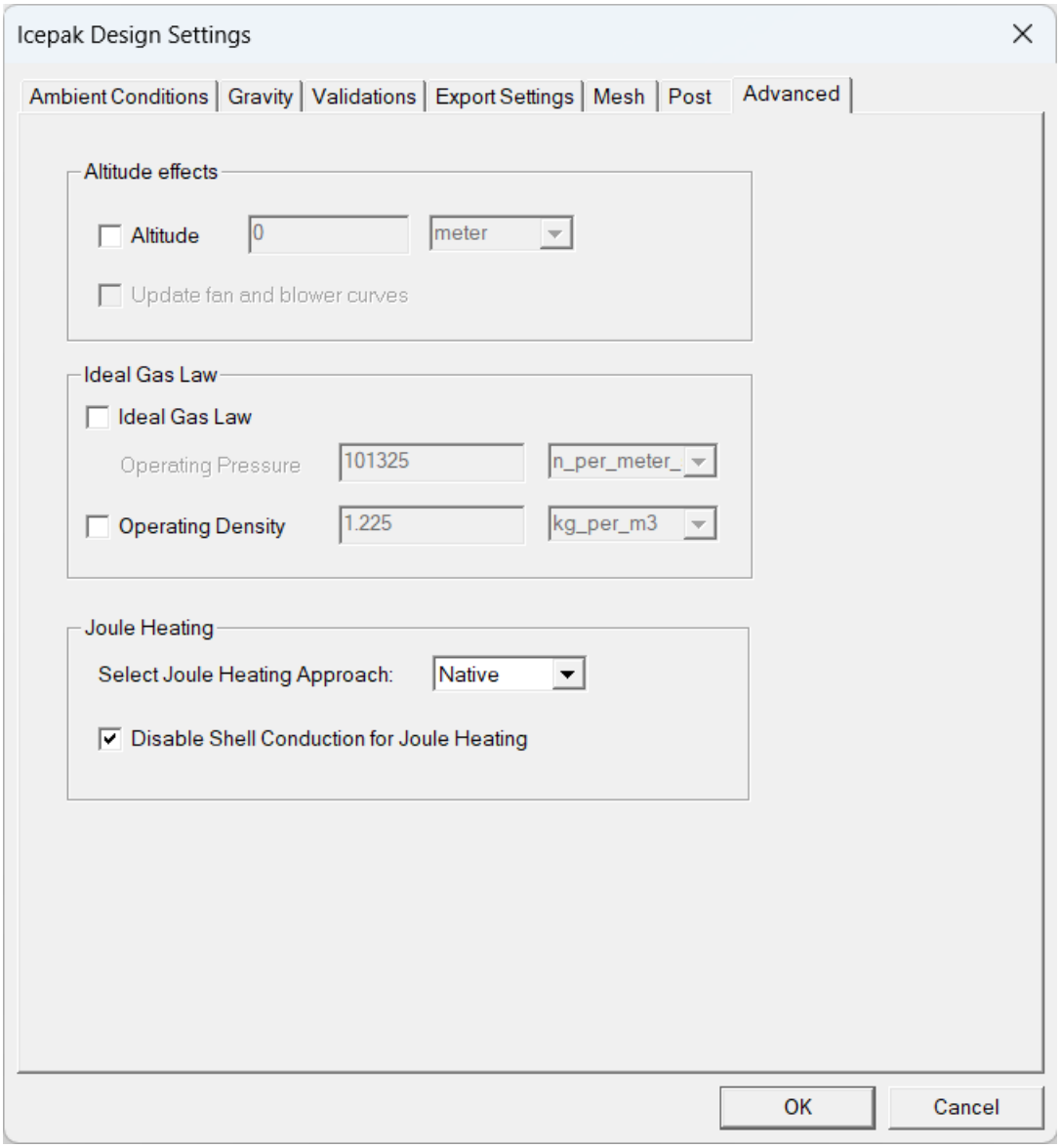
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Advanced Tab

The **Advanced** tab allows you to enable options to account for the effects of altitude and the Ideal Gas Law.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Altitude Effects

The density variation of air from sea level to higher altitudes can vary considerably. In addition, as the altitude increases, the mass flow rates of fans are reduced.

- Select **Altitude** option and specify the altitude and unit. Icepak computes the density of air at a particular altitude based on the International Standard Atmosphere, ISO 2533:1975 published by the International Organization for Standardization (ISO).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a fan or blower is present in the model, enable **Update fan and blower curves**. This automatically updates the fan characteristic curves based on the ratio of densities of air at the specified altitude and the sea level. Note that this option updates the curves for characteristic curve fans only. Fixed flow fans will have the same constant mass/volume flow at higher altitudes as they do at sea level, albeit with a lower air density.

> **Note:** The effect of altitude can be modeled by Icepak only when the default fluid is air.

Ideal Gas Law Effects

> **Note:** When altitude effects are modeled, Ideal Gas Law cannot be used, though note that the operating pressure and density are updated after running a simulation with altitude effects enabled.

Icepak provides options for defining temperature-dependent fluid density. By default, the Boussinesq approximation model is used.

> **Note:**
>
> For temperatures greater than 115 Celsius, it is recommended to use the Ideal Gas Law model for natural convection simulations.

> **Note:** You can also apply a thermal modifier in the material properties. See Specifying Thermal Modifiers for more information.

Alternatively, you can enable **Ideal Gas Law** (see Incompressible Ideal Gas Law), which should be used when pressure variations are small enough that the flow is fully incompressible but you want to use the ideal gas law to express the relationship between density and temperature (for example, for a natural-convection problem). The ideal gas law should not be used to calculate time-dependent natural convection in closed domains.

When using the ideal gas law, also specify the **Operating Pressure**.

> **Important:** The input of the operating pressure is of great importance when you are computing density with the ideal gas law. You should use a value that is representative of the mean flow pressure. The operating pressure is set to 101325 Pa by default, which is the atmospheric pressure at sea level. The operating pressure will decrease with increasing altitude.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If needed, enable **Operating Density** and specify the operating density (for ideal gas law or Boussinesq). By default, Icepak will compute the operating density by averaging over all elements. In some cases, you may obtain better results if you explicitly specify the operating density instead of having Icepak compute it for you. For example, if you are solving a natural-convection problem with a pressure boundary, it is important to understand that the pressure you are specifying is p_s' in Equation 6 under Definition of the Operating Density. Although you will know the actual pressure p_s, you will need to know the operating density in order to determine p_s' from p_s. Therefore, you should explicitly specify the operating density rather than use the computed average. The specified value should, however, be representative of the average value.

> **Important:** If you use the ideal gas law and you have created a new fluid material or copied a fluid material, make sure that you specify the correct molecular weight for the new or copied material.

Joule Heating

Icepak provides two options to model the heat generated by the flow of electrical current through an electrical conductor (joule heating) namely, the **Native** and **UDS** (user-defined scalar) joule heating models.

- **Native** joule heating uses the Ansys Fluent electric potential model, which allows for predicting the electric potential field and its effects in electrically conducting solids.

  > **Important: Native** joule heating cannot be used in simulations when conducting plates are present in the model with the shell conduction option enabled. As a result, **Disable Shell Conduction for Joule Heating** must be selected when shell conduction plates are present in the model. This option turns off heat conduction in the plane of the plate and only allows heat conduction normal to the plane.

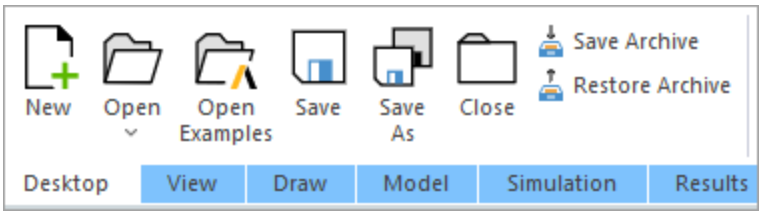
- **UDS** joule heating solves an additional conservation equation for the electric potential. While this option is quite similar to the native joule heating model and models the joule heating power accurately for all objects, it is prone to solver instability, especially for skewed meshes.

Opening Projects

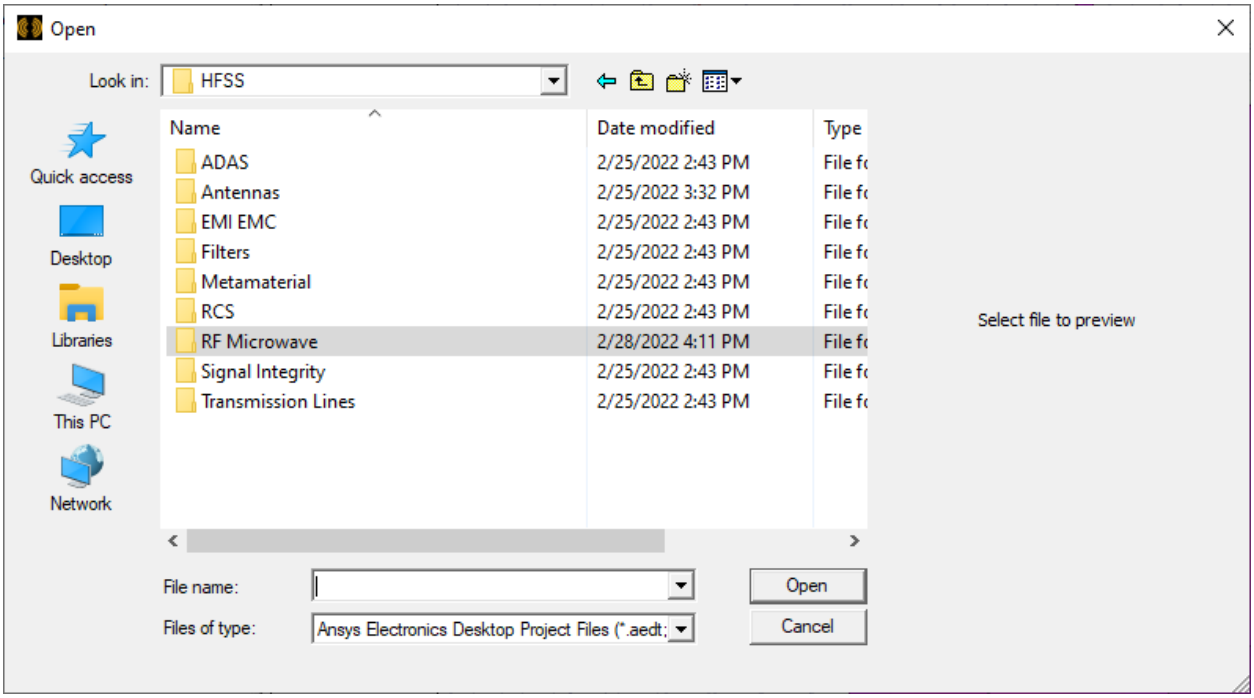
Open a previously saved project using the **File** > **Open** command.

1. Click **File** > **Open** or click the **Open** icon on the **Desktop** tab of the ribbon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



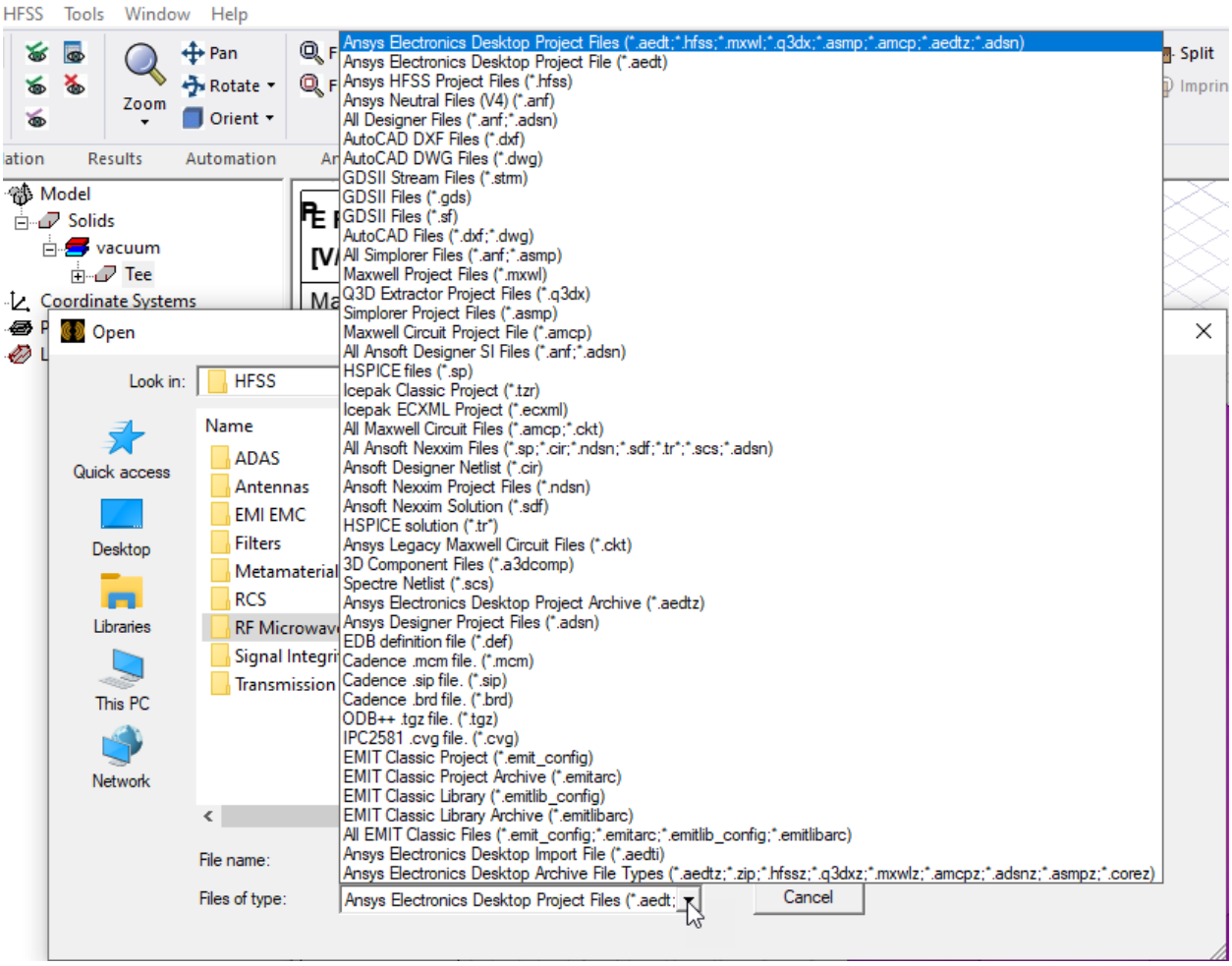
2. Use the file browser to find the project file.



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 drop-down menu for Files of

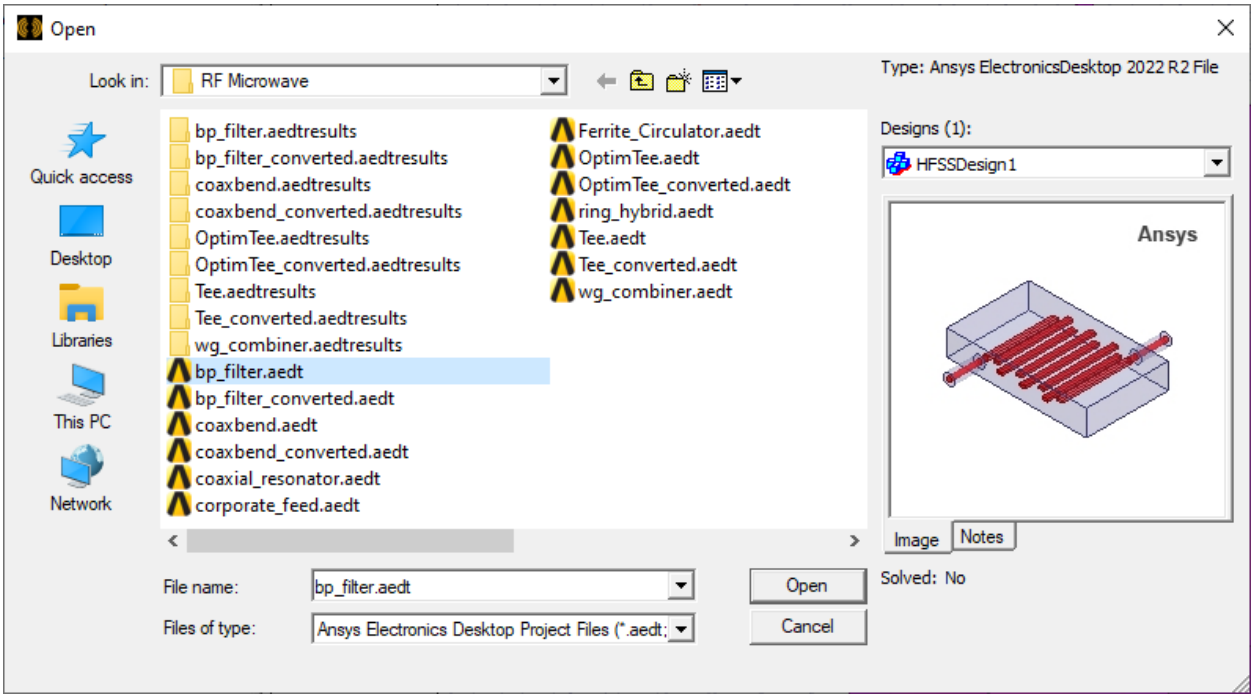
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

type 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 corresponding 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 2026 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:

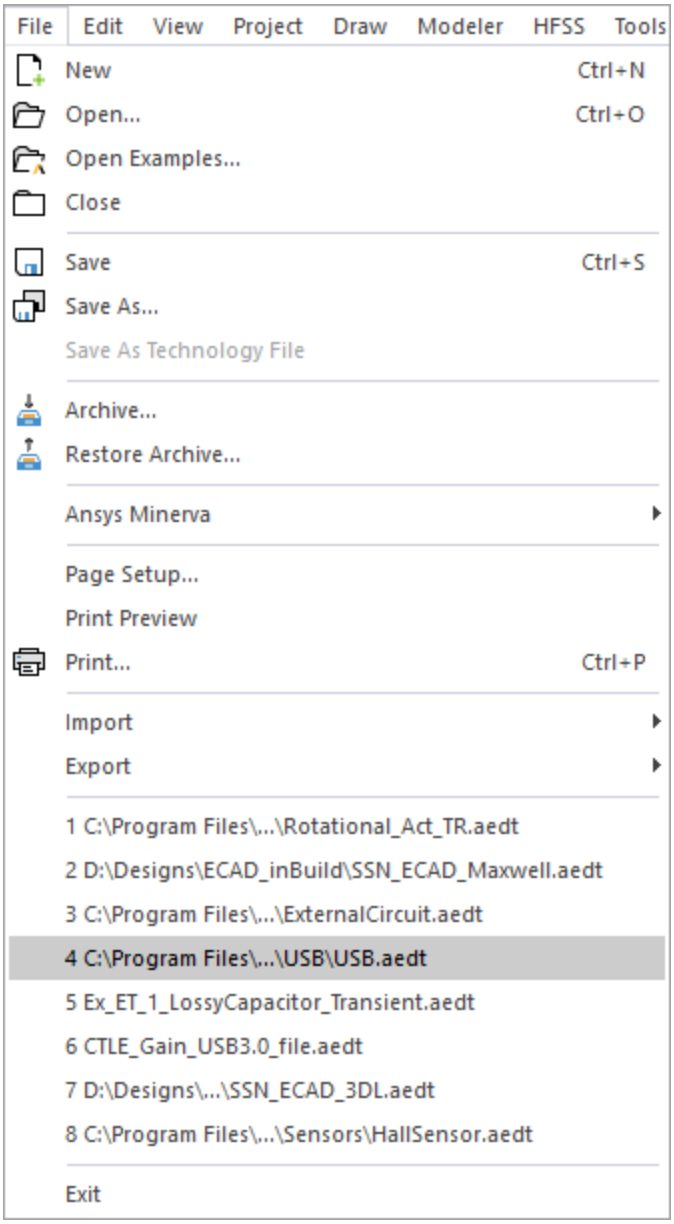
- Dragging a project file icon to the Open icon
- Dragging a project file icon to the desktop
- Double-clicking a project file icon

> **Note:**
>
> Icepak classic projects saved as *.tzr files in the Icepak classic application can be opened in the Electronics Desktop. In the **Open** dialog box, from the **Files of type** drop-down list, select **Icepak Classic Project (.tzr)**. See Opening Icepak Classic Projects for more information.

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 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Opening Icepak Classic Projects

You can migrate steady-state and transient projects from the Icepak Classic application to the Electronics Desktop. Generate a .tzr file in Icepak Classic in order to convert it into an Electronics Desktop Icepak design.

Object priorities, material settings, mesh settings, parametric trials, point and surface monitors, solver settings, post-processing objects, summary reports are converted in addition to Icepak Classic objects.

> **Note:**
>
> See EC XML and Icepak Classic Project Conversion Limitations for more information on what is not currently supported.

Icepak Classic objects are composed of both geometry and thermal and fluid properties. When opening a .tzr file in the Electronics Desktop, Icepak classic objects are converted into geometry in the history tree and 3D modeler window, and thermal properties are converted into boundary conditions, which are displayed in the Project Manager and assigned to the corresponding geometry in the 3D Modeler window.

Icepak Classic object geometry is converted into geometry in the Electronics Desktop based on geometry settings in Icepak Classic. The following sections detail the conversion of Icepak Classic object properties into Icepak native components and/or thermal boundary conditions.

- Block
- Network
- PCB
- Wall
- Plate
- Heatsink
- Fan
- Grille
- Source
- Enclosure
- Opening
- Resistance
- Cabinet
- Parametric Trials

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inactive Objects

Inactive objects are not imported into the Electronics Desktop unless the following environment variable is defined:

TZRImport_Inactive_Items

Set the definition of the variable to 1.

> **Note:** Inactive objects are imported into a separate Icepak design.

Linux System Configuration

On Linux systems, if you encounter the following error messages when attempting to open an Icepak Classic project, first check the system environmental variables and ensure that any python-related variables that specify path settings are disabled. If there are no python-related environmental variables defined, navigate to /usr/bin/ to locate the preinstalled version of python and rename it.

[error] The type initializer for 'IronPython.Modules.PythonNT' threw an exception. ---- Scripting Engine Initialization

[error] TypeInitializationException: The type initializer for 'IronPython.Modules.PythonNT' threw an exception. In file "/tmp/fileMOOvjk2198615392889450/aedt.py", line 1 ---- While executing script: /tmp/fileMOOvjk2198615392889450/aedt.py

Accelerate Classic Icepak File Import

To improve the TZR import speed, you can use non-graphical batch execution to import the model in AEDT.

Export Classic Icepak File

1. Open Classic Icepak.
2. From the **File** menu, select **Export** > **Ansys Electronics Desktop Script**.
3. In the **Select file for Ansys Electronics Desktop script** dialog box, browse to a save location and click **Save**. A python script file is generated in the save location.

Import a Classic Icepak File

1. In Windows, open the **Environment Variables** dialog box and create the following variable:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYSEM_FEATURE_SF6694_NON_GRAPHICAL_COMMAND_EXECUTION_ENABLE

2. Define the variable value as 1.
3. Open a command prompt and run the following command:

   "C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt.exe" -RunScriptAndExit D:\Users\username\aedt.py -LogFile D:\Users\username\AEDT_Message_Manager.log -Ng

   where D:\Users\username\aedt.py is the file path to the location of the previously generated python script file.

Note: In the command line, ensure the Ansys software version number corresponds to the version you are using.

Import Multiple Classic Icepak Files with Non-Graphical Batch Converter

Note: This workflow imports single Classic Icepak files in addition to multiple files.

1. Open a command prompt and run the following command:

   cd to "C:\Program Files\ANSYS Inc\v261\AnsysEM\syslib\TZRBatchConverter"

2. Run the following command:

   "C:\Program Files\ANSYS Inc\v261\AnsysEM\common\commonfiles\CPython\3_10\winx64\python\python.exe" tzr_converter.py -t D:\Users\tzrs -s D:\Users\aedtfiles -a "C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt.exe" -p 4

   where D:\Users\tzrs is the file path to the location of folder containing the Classic Icepak file(s).

Note: In the command line, ensure the Ansys software version number corresponds to the version you are using.

Optionally, to view argument definitions, enter the following command after step 1:

"C:\Program Files\ANSYS Inc\v261\AnsysEM\common\commonfiles\CPython\3_10\winx64\python\python.exe" tzr_converter.py -h

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
C:\Users>cd C:\Program Files\AnsysEM\v221\Win64\syslib\TZRBatchConverter

C:\Program Files\AnsysEM\v221\Win64\syslib\TZRBatchConverter>"C:\Program Files\AnsysEM\v221\
Win64\common\commonfiles\CPython\3_7\winx64\Release\python\python.exe" tzr_converter.py -h
usage: tzr_converter.py [-h] [-t TZRPATH] [-s SAVEPATH] [-a AEDTPATH]
                        [-p PARALLEL]

Batch converter for converting classic icepak tzr files to AEDT files

optional arguments:
  -h, --help            show this help message and exit
  -t TZRPATH, --tzrpath TZRPATH
                        folder location with all classic icepak tzr files.
                        Ex:-t D:\Users\tzr_source_files
  -s SAVEPATH, --savepath SAVEPATH
                        folder location to save corresponding AEDT files and
                        logs. Ex:-t D:\Users\folder_for_aedt_files
  -a AEDTPATH, --aedtpath AEDTPATH
                        AEDT install location. Ex:-t C:\Program
                        Files\AnsysEM\vxxx\Win64\ansysedt.exe
  -p PARALLEL, --parallel PARALLEL
                        Please provide parallel options. Ex: -p 3
```

Icepak Classic Block Conversion

The table below details Icepak Classic Block properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Block Properties	Electronics Desktop Boundary Condition
Radiation is disabled and total power is non-zero	Block
Reference temperature radiation is enabled	Source
Reference temperature radiation and External Conditions are enabled	Source, and a Stationary Wall for each face
External conditions is enabled	Stationary Wall for each face
Individual sides are specified with Thermal condition as Fixed heat	Source(s) at the specified face(s)
Individual sides are specified with Thermal condition as Fixed heat and Resistance enabled	Conducting Plate at the specified face(s)
Reference temperature radiation is enabled and Thermal condition is Internal Conditions	Stationary Wall(s) at the specified face(s)
Network Block type is selected	Network at the specified face(s)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If the Icepak Classic project contains an ECAD block, the ECAD file must exist as specified in the INFO file, or it must be included in the same directory as the .tzr file. If so, a PCB native component is created in the Electronics Desktop. If the file does not exist as specified, a 3D bounding box is created.

Icepak Classic Network Conversion

The table below details Icepak Classic Network properties and their effect on boundary condition assignments in the Electronics Desktop.

Note: Network are not converted from .ecxmls files.

Icepak Classic Network Properties	Electronics Desktop Boundary Condition
Network object with no convective links or boundary nodes	Network

Icepak Classic PCB Conversion

The table below details Icepak Classic PCB properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic PCB Properties	Electronics Desktop Boundary Condition
PCB type is Compact and Total power is non-zero	Block
PCB type is Compact or Detailed and Low side and/or High side radiation is enabled	Conducting Plate(s) with radiation at the high side and/or low side trace layers
PCB type is Detailed, Thermal Specification is Total, and Total power is non-zero	Block
PCB type is Detailed and Thermal Specification is Upper/Lower	Conducting Plates with power at the high side and/or low side trace layers

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If the Icepak Classic project contains an ECAD PCB, the ECAD file must exist as specified in the INFO file, or it must be included in the same directory as the .tzr file. If so, a PCB native component is created in the Electronics Desktop. If the file does not exist as specified, a 3D bounding box is created.

Icepak Classic Wall Conversion

The table below details Icepak Classic Wall properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Wall Properties	Electronics Desktop Boundary Condition
Wall type is Symmetry	Symmetry Wall
Wall type is Stationary and Effective thickness is enabled	Stationary Wall
Wall type is Stationary and Effective thickness is disabled	Object with thickness equal to the Wall thickness with a Stationary Wall for each face

Icepak Classic Plate Conversion

The table below details Icepak Classic Plate properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Plate Properties	Electronics Desktop Boundary Condition
Thermal model is Adiabatic thin	Adiabatic Plate
Thermal model is Conducting thin	Conducting Plate
Thermal model is Conducting thick or Hollow thick and Total power is non-zero	Block
Thermal model is Conducting thick or Hollow thick and Low side and/or High side Resistance is enabled	Stationary Wall(s) at the high side and/or low side
Thermal model is Conducting thick or Hollow thick, Low side and/or High side is enabled, Resistance is disabled, and Radiation is enabled	Source(s) at the high side and/or low side
Thermal model is Contact resistance	Conducting Plate

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Classic Heatsink Conversion

The table below details Icepak Classic Heatsink properties and their effect on Icepak native component creation and boundary condition assignments in the Electronics Desktop.

Icepak Classic Heatsink Properties	Electronics Desktop Native Component	Electronics Desktop Boundary Condition
Type is Detailed and Detailed fin type is Extruded	Heatsink with Fin Type set to Extruded	n/a
Type is Detailed and Detailed fin type is Cross cut extrusion	Heatsink with Fin Type set to Crosscut Extrusion	n/a
Type is Detailed and Detailed fin type is Bonded fin	Heatsink with Fin Type set to Extruded	Conducting Plate assigned to all the bottom faces of the fins
Type is Detailed and Detailed fin type is Cylindrical pin	Heatsink with Fin Type set to Cylindrical Pin	n/a
Type is Detailed and Interface resistance is enabled	Heatsink with Fin Type set according to Detailed fin type	Conducting Plate assigned to all the bottom faces of the fins

Icepak Classic Fan Conversion

The table below details Icepak Classic Fan properties and their effect on Icepak native component creation in the Electronics Desktop.

Icepak Classic Fan Properties	Electronics Desktop Native Component
Model is 2d and Shape is Circular	Fan with Model set to 2D and Shape set to Circular
Model is 2d and Shape is Rectangular	Fan with Model set to 2D and Shape set to Rectangular
Model is 3d and Shape is Circular	Fan with Model set to 3D and Shape set to Circular
Model is 3d and Shape is Rectangular	Fan with Model set to 3D and Shape set to Rectangular

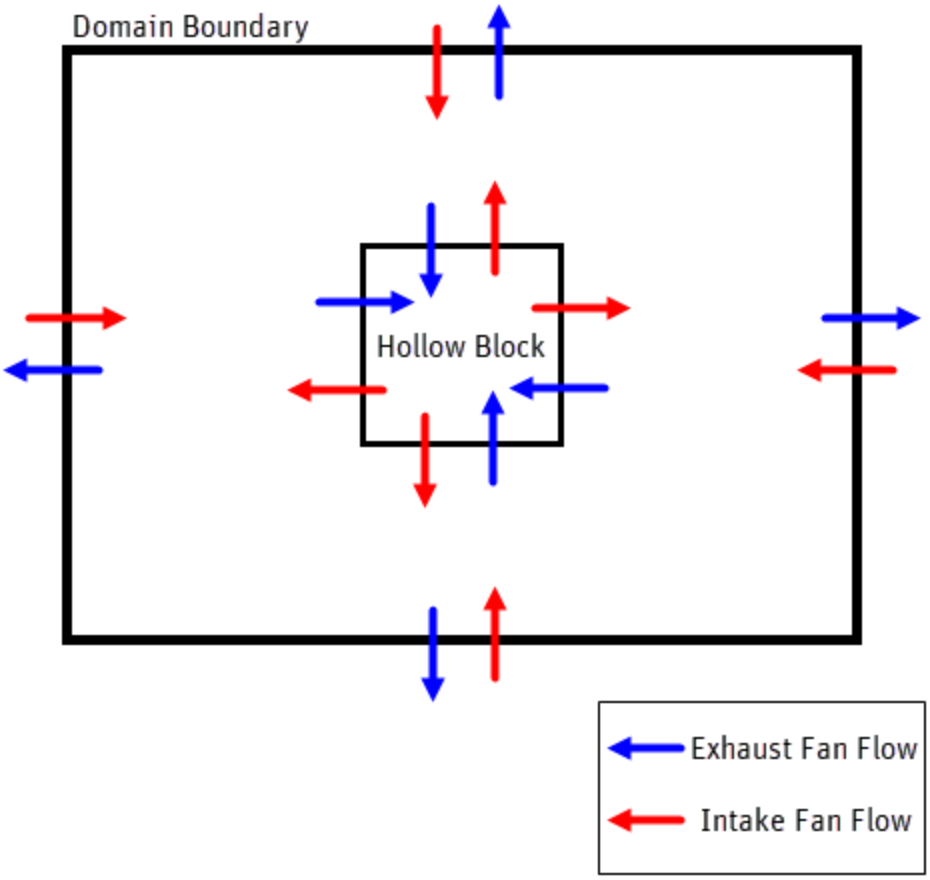
Fan Flow Direction

Fan flow direction is applied based on the Fan type setting (Internal, Intake, and Exhaust). The following rules are applied:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Internal fans: flow direction is set to the direction specified in Icepak Classic.
- Intake fans
  - Located on the domain boundary: flow direction is set to be normal to the domain, facing inward.
  - Not located on the domain boundary: the Electronics Desktop default is set unless the fan is located on the surface of an object with Solve inside turned off. Then the direction is set to be normal to the domain, facing outward.
- Exhaust fan
  - Located on the domain boundary: flow direction is set to be normal to the domain, facing outward.
  - Not located on the domain boundary: the Electronics Desktop default is set unless the fan is located on the surface of an object with Solve inside turned off. Then the direction is set to be normal to the domain, facing inward.

The following image illustrates Intake and Exhaust fan flow direction on the domain boundary and on the surface of an object with Solve inside turned off.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Classic Grille Conversion

The table below details Icepak Classic Grille properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Grille Properties	Electronics Desktop Boundary Condition
All Grilles	Grille

Icepak Classic Source Conversion

The table below details Icepak Classic Source properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Source Properties	Electronics Desktop Boundary Condition
2D Sources	Source
3D Sources	Block

Icepak Classic Enclosure Conversion

The table below details Icepak Classic Enclosure properties and their effect on boundary condition assignments in the Electronics Desktop. For Enclosures, each side (Min X, Max X, Min Y, Max Y, Min Z, and Max Z) may be assigned a boundary condition depending on its settings.

Icepak Classic Enclosure Properties	Electronics Desktop Boundary Condition
Min/Max X, Y, or Z is Thick, Power is non-zero, and Radiation is disabled	Block
Min/Max X, Y, or Z is Thick, Power is non-zero, and Radiation is enabled	Source
Min/Max X, Y, or Z is Thin, Power is non-zero, and/or Radiation is enabled	Conducting Plate

Icepak Classic Opening Conversion

The table below details Icepak Classic Opening properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Opening Properties	Electronics Desktop Boundary Condition
All Openings	Opening

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Classic Resistance Conversion

The table below details Icepak Classic Resistance properties and their effect on boundary condition assignments in the Electronics Desktop.

Icepak Classic Resistance Properties	Electronics Desktop Boundary Condition
All Resistances	Resistance

Icepak Classic Cabinet Conversion

The table below details Icepak Classic Cabinet properties and their effect on boundary condition assignments in the Electronics Desktop. For a Cabinet, each side (Min X, Max X, Min Y, Max Y, Min Z, and Max Z) may be assigned a boundary condition depending on its settings.

Icepak Classic Cabinet Properties	Electronics Desktop Boundary Condition
Min/Max X, Y, or Z is a Wall	Wall (see Icepak Classic Wall Conversion for more information)
Min/Max X, Y, or Z is an Opening	Opening
Min/Max X, Y, or Z is a Grille	Grille

Icepak Classic Parametric Trials Conversion

The table below details how Icepak classic parametric trials are converted to the Electronics Desktop.

Icepak Classic	Electronics Desktop
Parameters and optimization panel > Design variables	Project variables
Parameters and optimization panel > Trials	Optimetrics > Parametric Setup
Variable applied to Icepak classic object geometry	Variable applied to corresponding geometry
Variable applied to Icepak classic object property	Variable applied to corresponding boundary condition

Icepak Classic Project Conversion Limitations

The following features and settings are not supported and, therefore, not converted when opening an Icepak Classic .tzr file in the Electronics Desktop.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When opening a Classic Icepak project consisting of many mesh operations or boundary conditions, the conversion process may take longer to complete.

Naming

- The Electronics Desktop limits the number of characters for a given object name to 60 characters. Icepak Classic objects with names longer than 60 characters are truncated.
- The Electronics Desktop limits the number of characters for a given material name to 45 characters. Icepak Classic objects with names longer than 45 characters are truncated.
- Minor naming differences may occur (e.g., block.1 converted to block_1).
- Names that are considered unique in Icepak Classic (e.g., *block.1* and *Block.1*) may be considered duplicate in the Electronics Desktop. To avoid incorrect assignment of boundary conditions, names are made unique by appending "_1" to the name (e.g., *block_1* and *Block_1_1*).
- For all assemblies/mesh regions, *_mr* is appended to the assembly name.
- Point monitors will be prepended with *pmon_*, and face monitors are prepended with *fmon_*.
- All non-ASCII characters and special characters are translated as an underscore character (_).
- When opening an Icepak Classic file in the Electronics Desktop on a Linux system, the file name and file path must not contain more than one space.

Project Settings

- Object groups

Mesh Settings

> **Note:**
>
> In Icepak Classic, an assembly's local Maximum element size can be overridden by global mesh settings. Ensure that Maximum element size for mesh regions in the Electronics Desktop are set appropriately before meshing.

- Hexa unstructured mesh type is converted to Mesher-HD
- Hexa cartesian mesh type is converted to Mesher-HD

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Operation

- Mesh operations are not supported for Packages and all unsupported "Icepak Classic Objects" on page 3-125.
- A mesh operation is not created for a fan guard.

> **Note:** Mesh operations for Icepak 3D Native Components (Fans, Heatsinks, and PCBs) can be assigned on individual pieces of geometry that compose the component.

Mesh Reuse Settings

> **Note:** For Fans, Heatsinks, and PCBs, you can apply mesh reuse settings to individual pieces of geometry.

- Assemblies
- Cabinet
- Fan
- Heatsink
- PCB
- Groups
- Sheet bodies are not supported. Only solid bodies are supported.

> **Note:** Mesh reuse is supported for objects with converted geometry. For example, a Cabinet with a wall specified as a side is supported, whereas a block with thermal properties specified for an individual side is not supported.

Object Priority Settings

The following are not assigned priority values:

- Assemblies
- Unsupported "Icepak Classic Objects" on page 3-125
- Inactive Objects

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solver Settings

- Surface to surface radiation is converted to Discrete ordinates radiation.

  > **Note:**
  >
  > In the Electronics Desktop, Discrete ordinates is enabled even when Surface to surface radiation contains no objects selected under Participating objects in the Icepak Classic project.

- Autosave interval for steady-state simulations

High Performance Computing Settings

- The number of cores and tasks must be specified in the Electronics Desktop after converting a Classic Icepak file.

Transient Settings

- Solution restart settings
- Joule heating properties for block and network boundary conditions
- Fields summary times (defaults to 0s)

Materials

- Biaxial materials
- Anisotropic materials

Geometry

- Geometry with applied texture
- A polyhedron whose base and top have differing numbers of sides is not supported (e.g., the base is a hexagon and top is a pentagon).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Classic Objects

> **Note:**
>
> Inactive objects in Classic Icepak are converted as suppressed objects in Electronics Desktop.

- PCBs with MCAD substrates that contain ECAD traces inside are not supported.
- Packages
  - ECAD-based packages: flip-chip and PBGA packages are supported; others are not supported.
  - Non-ECAD-based packages: stacked die, package-on-package, and QFN are not supported.
  - MCAD substrates with ECAD traces inside are not supported
- Periodic boundaries

Blocks

- SIwave profile (solid blocks)
- LED source (blocks)
- MCAD blocks with ECAD
- Blocks with ECAD with Resize object with traces enabled

Networks

- For some cases, the schematic editor displays Page ports.

Walls

- Moving wall

Plates

- Temperature limit

Resistances

- Flow direction
- Species

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PCBs

- Only ECAD based PBGA and Flip-chip package types are supported.
- ECAD PCB with MCAD-based substrate
- PCBs with ECAD with Resize object with traces enabled
- Multiple ECAD Layout Designs under the same original project in an Icepak Design
- SIwave profile

Packages

- Mesh operations
- Non-ECAD packages of type Stacked Die, Package on Package, QFP, and QFN
- ECAD package with MCAD-based substrate
- Non-ECAD packages are not imported as a 3D component.

Heatsinks

- Simple heatsinks
- Cylindrical fin heatsinks: Staggered pin alignment
- Cells between pins (a per-object mesh setting)

Fans

- Polygonal fan with a non-uniform radius
- MCAD fan geometry is imported without thermal properties.

  > **Note:** In Classic Icepak, a 3D MCAD fan cannot be created. When a 2D MCAD fan is imported into the Electronics Desktop, the geometry is imported as an MCAD sheet. No native fan component is associated with the imported MCAD geometry and, therefore, does not appear in the Project Manager or History tree.

- Gauge pressure

Grilles

- Velocity loss coefficient is set to automatic because device/approach options are unsupported.
- Flow direction
- Species

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sources

- Spatial profile for 3D sources
- Temperature limit
- LED
- Species
- Fixed temperature for 3D sources
- Transient joule heating properties for 3D sources

Openings

- Species
- Turbulence
- Spatial profile for openings with mass flow specified

Chip Thermal Models

- CTM powermaps are converted to individual sources.

Assemblies

- Assemblies with unsupported "Icepak Classic Objects" on page 3-125
- Per-object mesh settings for Packages
- An active object under an inactive assembly is not imported unless the TZR_Inactive_ items environment variable is set.
- Assemblies superimposing the cabinet object are imported. The mesh region geometry must be edited to exist within the computational domain (Region) prior to meshing or running the simulation.

Monitors

- Surface monitors on a PCB, Fan, or Heatsink
- Point monitors assigned to a 3D CAD source object with a Thermal specification of Fixed Temperature are not imported.

Post-processing

- Show mesh for plane cuts, object faces, and point reports
- Summary report information on Block-based PCB objects

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

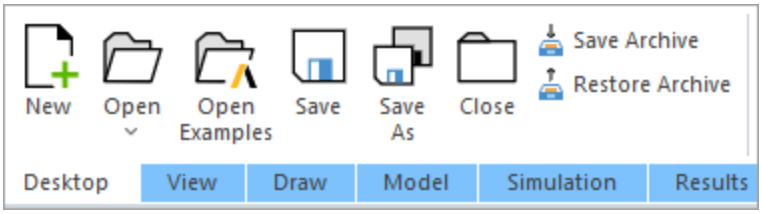
- For light-weight imported CAD geometries, only the full object can be selected when creating object faces and field summary entries.
- For light-weight imported CAD geometries in Field Summary entries, Inside, Interior, and Sides options are not supported.

ECAD

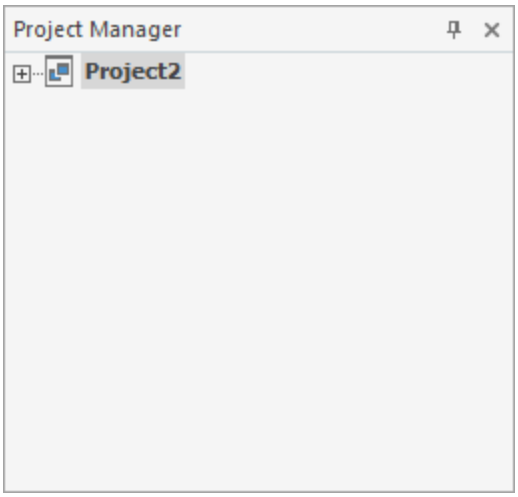
- For PCBs, Packages, and Blocks with ECAD, only non-zero thickness layers are imported.
- Projects with any ECAD cannot be imported into Ansys Electronics Desktop Student.

Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

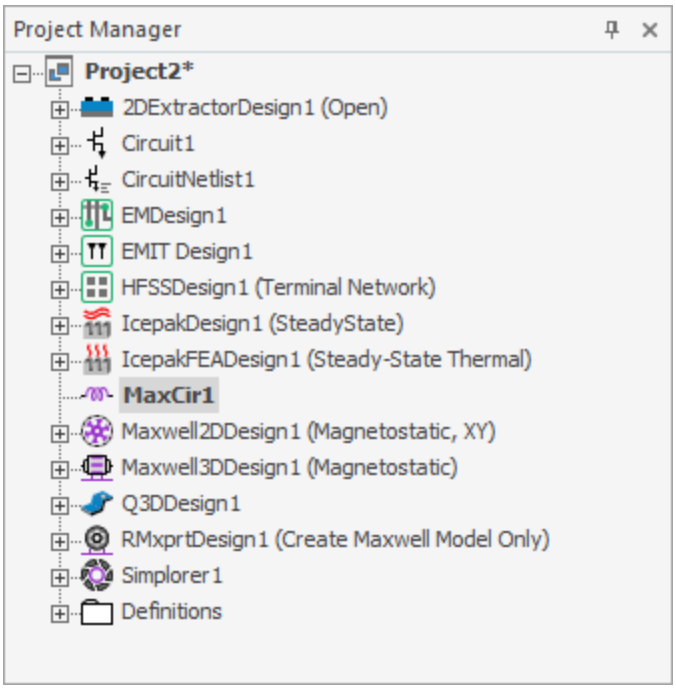


A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current project folder.



You can insert designs of any type into the project, where they are stored in the project tree.

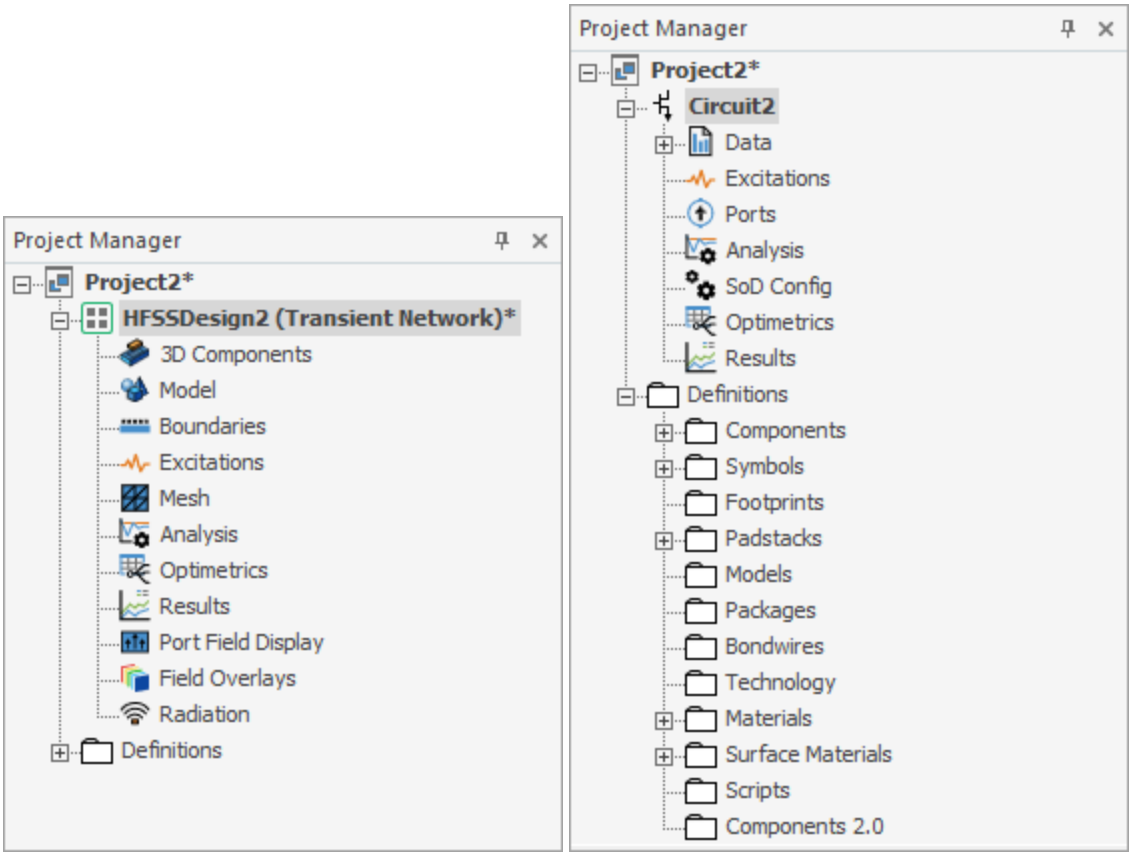
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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. The contents of the tree will be determined by the design type.

Ansys Electromagnetics Suite 2026 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

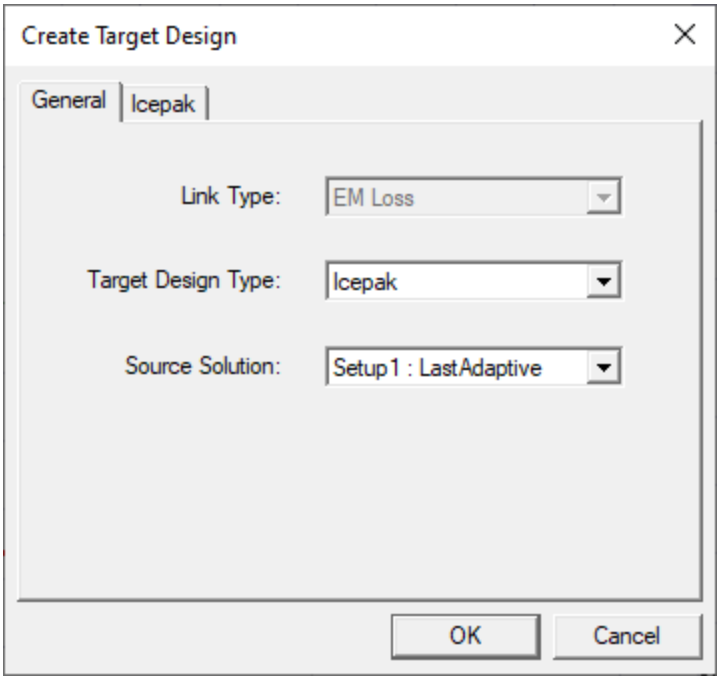
In an HFSS, Q3D, or Maxwell 3D design, **Create Target Design** automatically generates and inserts a thermal design (Icepak or IcepakFEA Thermal) into your project.

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 2026 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. For Icepak designs, specify **Forced Convection** or **Natural Convection** on the **Icepak** tab.
    - For **Forced Convection**, specify **Flow Speed** and **Flow Direction**.
    - For **Natural Convection**, specify the **Gravity Vector Direction**.
5. Click **OK**.

The Icepak design is generated based on the source design and selections in the **Create Target Design** dialog box:

- Model geometry is created with corresponding material assignments. Vacuum objects are not created in target Icepak designs.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Icepak.
- The Region (computational domain) is created with the appropriate dimensions.
- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.
- Design settings are specified.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

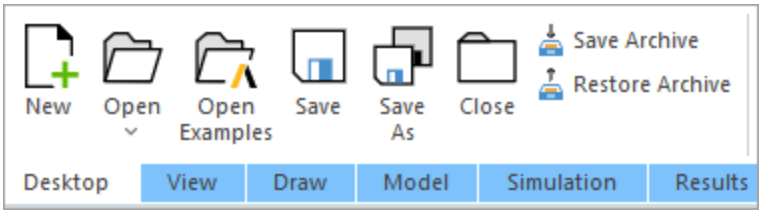
    - Solution type: **Steady State**
    - Problem type: **Temperature and Flow**
- For forced convection simulations, the following are defined:
    - A free opening boundary condition with a velocity specified is assigned to the Region face to create forced flow.
    - A free opening boundary condition is assigned to the face of the Region opposite to the opening with velocity specified.
    - In the **Icepak Solve Setup** dialog box, forced convection **Solve Setup Defaults** are defined.
- For natural convection simulations, the following are defined:
    - **Gravity Vector**
    - Free opening boundary conditions are assigned to each face of the air Region.
    - In the **Icepak Solve Setup** dialog box, natural convection **Solve Setup Defaults** are defined.

Saving Projects

Use the **File** > **Save As** command or select the **Desktop** tab of the Ribbon and click the **Save As** icon to:

- 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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:** When you recover a project's auto-save file you *cannot* recover any solutions data; recovering an auto-save file means you will lose any solutions data that existed in the original project file.

To recover project data in an auto-save file, if Ansys Electronics Desktop containing a design has unexpectedly crashed:

1. Launch Ansys Electronics Desktop from your desktop.
2. Click **File** > **Open**.
3. Select the original Project*n*.aedt project file for which you want to recover its Project*n*.aedt.auto auto-save file.

   The **Crash Recovery** window appears, giving you the option to open the original project file or the auto-save file.

4. Select **Open project using autosave file** to recover project data in the auto-save file**, and then click OK.** Ansys Electronics Desktop replaces the original project file with the data in the auto-save file.

   Ansys Electronics Desktop immediately overwrites the original project file data with the auto-save file data, removing the results directory (solutions data) from the original project file as it overwrites to the auto-save file.

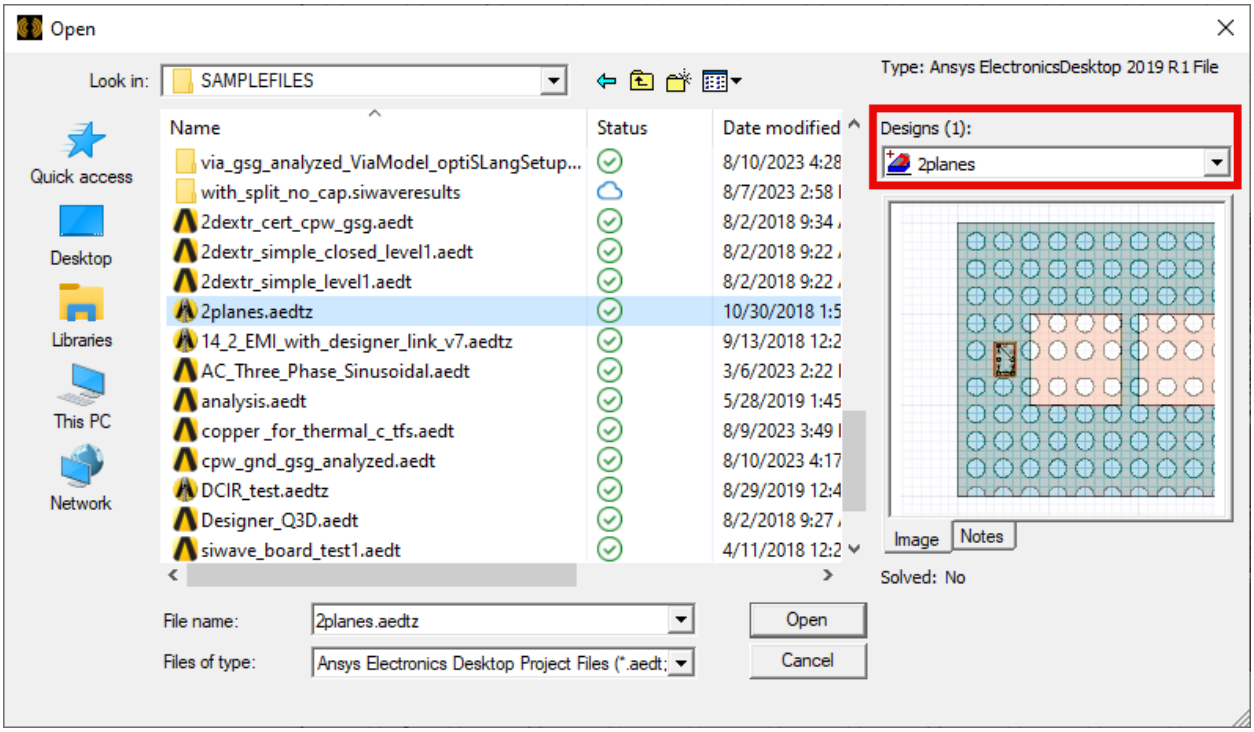
> **Warning:** If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

Choosing a Preview Image

When opening an *.aedt project file or *.aedtz archive, a preview image appears on the right side of the **Open** window.

The **Designs** drop-down menu allows users to preview the designs in a project before opening it.

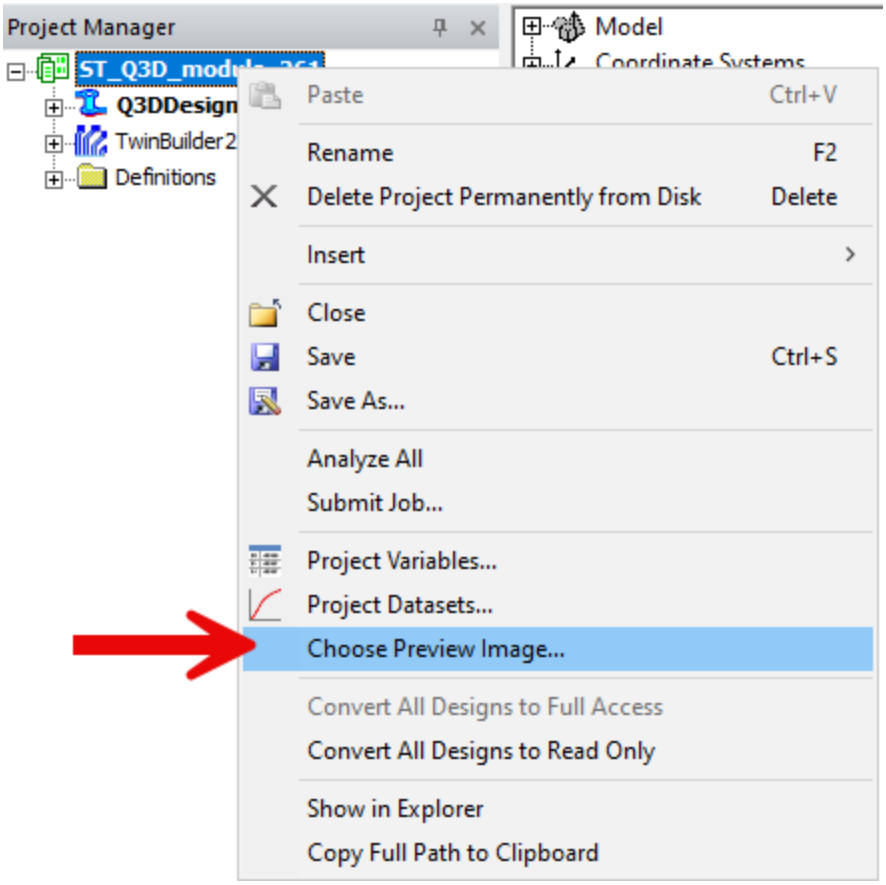
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



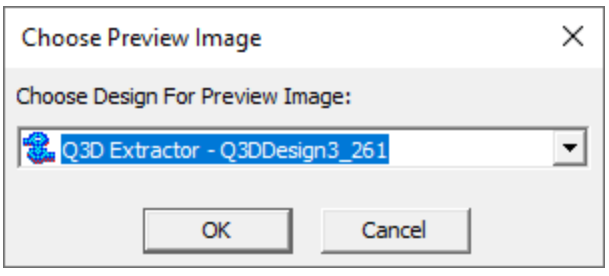
For a project containing multiple designs, users can select which design previews by default:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Project Manager**, right-click the project and select **Choose Preview Image**.



The **Choose Preview Image** window appears.



2. Use the drop-down menu to select a design.
3. Click **OK**.

Importing and Exporting Projects and Data

Ansys Electronics Desktop can import and export a variety of file types.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Files

You can import the following types of files to Icepak:

- 2D model (GDSII) files
- 3D model files
- DXF and DWG format files
- IDF files
- EC XML Files
- AEDTI 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)
- 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)

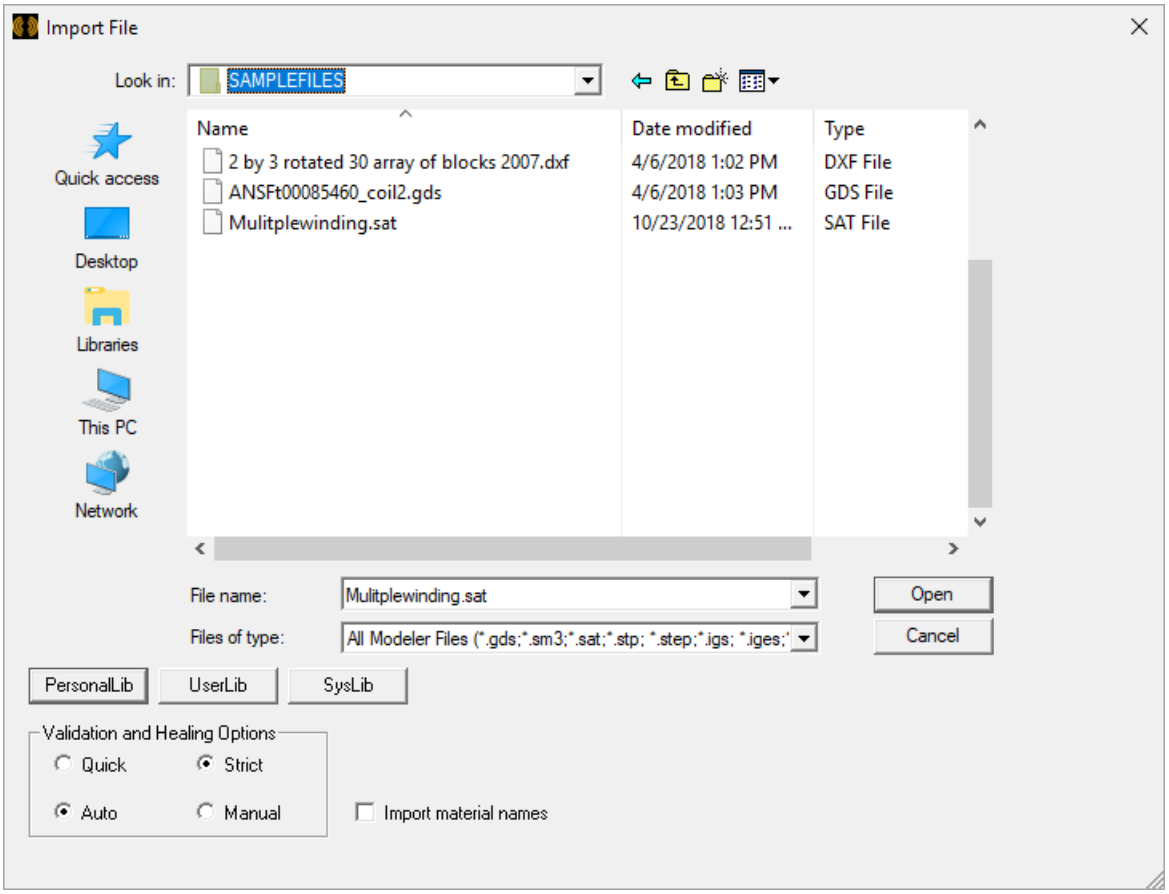
For more information, see Heal.

To import a file:

1. Click **Modeler > Import**.

   The **Import File** window appears.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

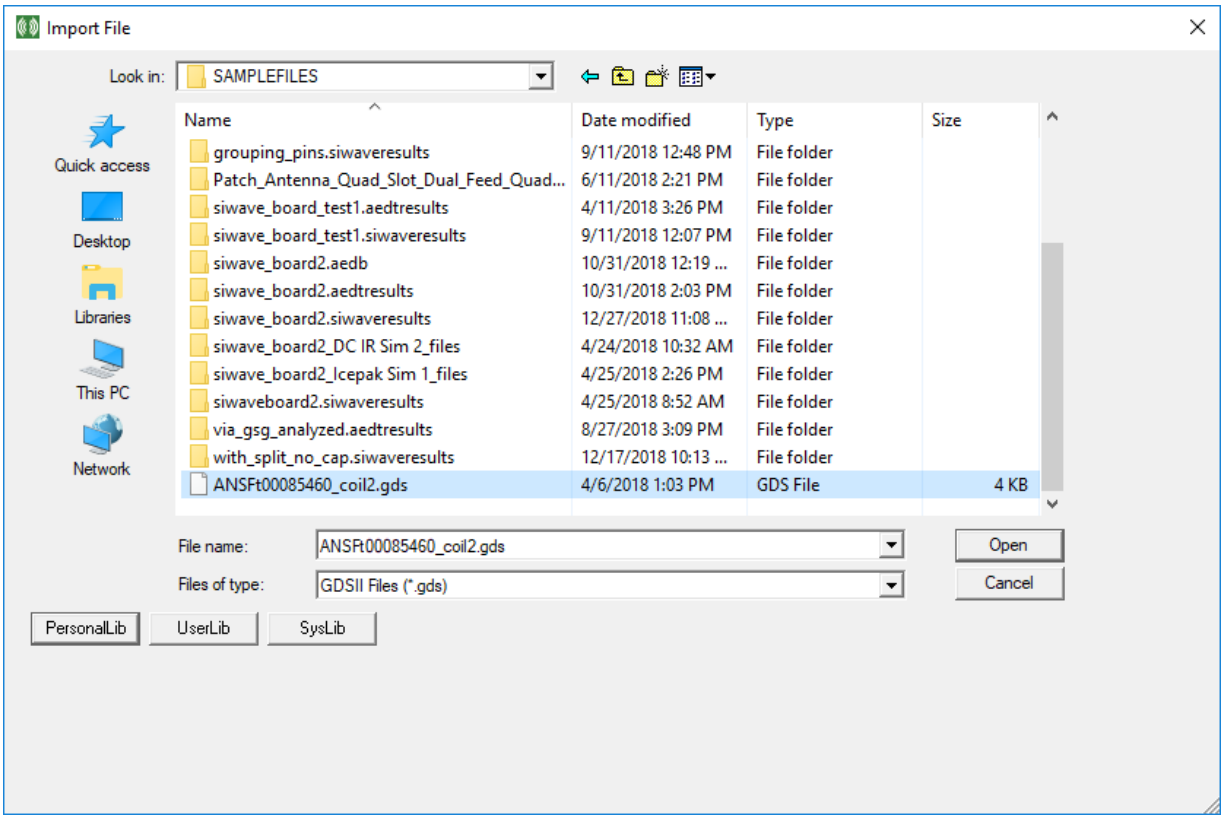
Note:

- 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**.

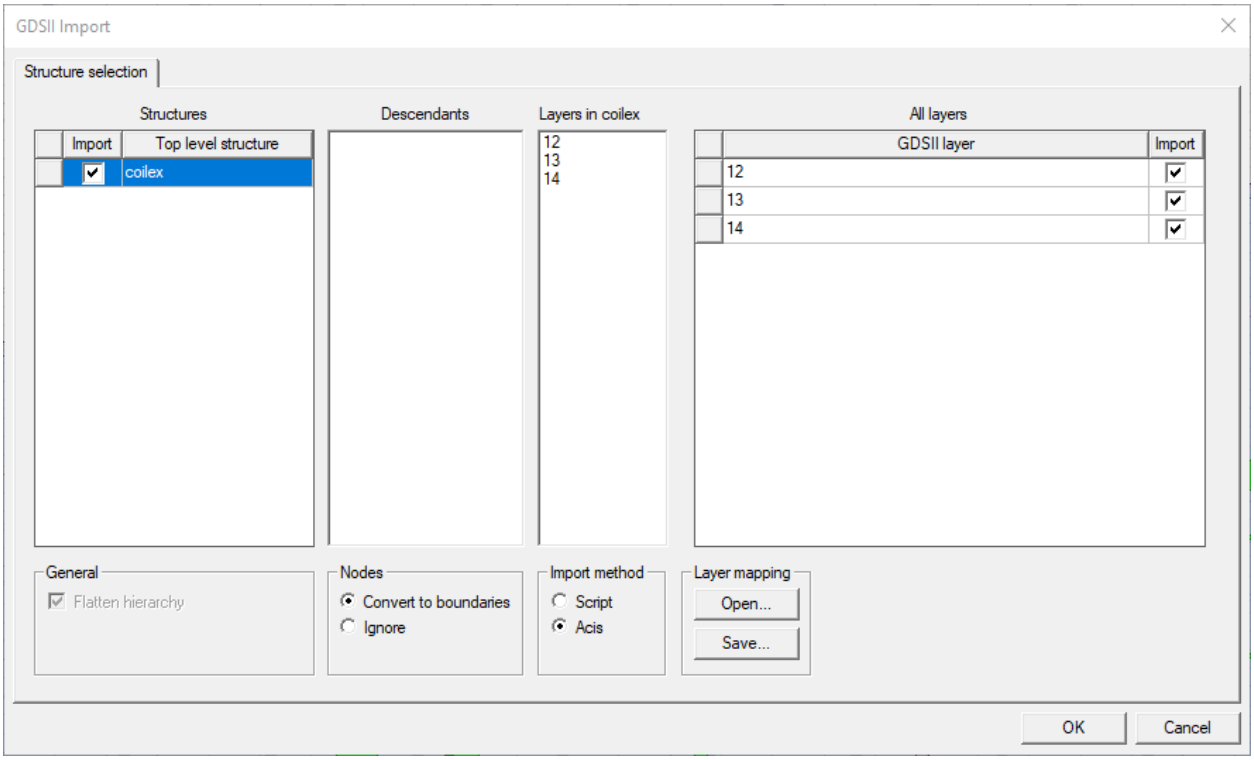
   The **Import File** dialog box appears.



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 2026 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, Icepak 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. Icepak always flattens any hierarchical geometry in the GDSII.
    - **Nodes** – GDSII supports nodes and boundaries as separate data types. Normally, boundaries represent polygons. Icepak 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 2026 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.
4. Click **OK**.

   The file is imported into the active **Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
4. Click **OK**.

   The file is imported into the active **Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 }
NavajoWhite1, navajo white	{ 255, 222, 173 }
NavajoWhite2	{ 238, 207, 161 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
MistyRose2	{ 238, 213, 210 }
MistyRose3	{ 205, 183, 181 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
DodgerBlue3	{ 24, 116, 205 }
DodgerBlue4	{ 16, 78, 139 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
dark turquoise	{ 0, 206, 209 }
medium turquoise	{ 72, 209, 204 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
SeaGreen2	{ 78, 238, 148 }
SeaGreen3	{ 67, 205, 128 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
khaki2	{ 238, 230, 133 }
khaki3	{ 205, 198, 115 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
indian red	{ 205, 92, 92 }
IndianRed1	{ 255, 106, 106 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
firebrick3	{ 205, 38, 38 }
firebrick4	{ 139, 26, 26 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
OrangeRed2	{ 238, 64, 0 }
OrangeRed3	{ 205, 55, 0 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
violet red	{ 208, 32, 144 }
VioletRed1	{ 255, 62, 150 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
medium purple	{ 147, 112, 219 }
MediumPurple1	{ 171, 130, 255 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
gray27	{ 69, 69, 69 }
gray28	{ 71, 71, 71 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
gray64	{ 163, 163, 163 }
gray65	{ 166, 166, 166 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
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 }
gray98	{ 250, 250, 250 }
gray99	{ 252, 252, 252 }

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray100, white	{ 255, 255, 255 }

Importing 3D Model Files

3D model files can be imported directly into the active **Modeler** window using the **Modeler** > **Import** menu. Supported 3D model file formats are listed in the table that follows.

With 2026 R1, Hoops Exchange translators will be used to translate following 3D CAD files: ACIS, Inventor, Catia, Creo, IGES, JT, NX, Solidworks, and STEP.

Type	Extension(s)	Supported Version(s)
ACIS SAB	*.sab	Up to 2023
ACIS SAT	*.sat	Up to 2023
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 V2026
CATIA	*.exp, *.model, *.CATPart, *.CATproduct	CATIA V4 – Up to 4.2.5 CATIA V5 – Up to V5_6R2025
Creo Parametric	*.prt, *.asm	Pro/Engineer 19.0 to Creo 12.4.0.0
GL Transmission Format	*.gltf (ASCII), *.glb (binary)	Version 2.0 only
IGES	*.iges, *.igs	5.1, 5.2, 5.3
JT	*.jt	B-rep JT, up to 10.9
NASTRAN	*.nas	CROD, CBEAM, CTRIA3, and CQUAD4 element types
Siemens NX	*.prt	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2506
Parasolid	*.x_t, *.x_b	Up to 38.0
SOLIDWORKS	*.SLDPRT,	From 97 up to 2025

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Extension(s) *.SLDASM	Supported Version(s)
STEP	*.step, *.stp	AP 203 Ed1, Ed2 - AP 214 - AP 242 Ed1, Ed2, Ed3
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 CATIA, STEP and IGES files, specify stitch tolerance and units. The default value (**Auto**) comes from the 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.



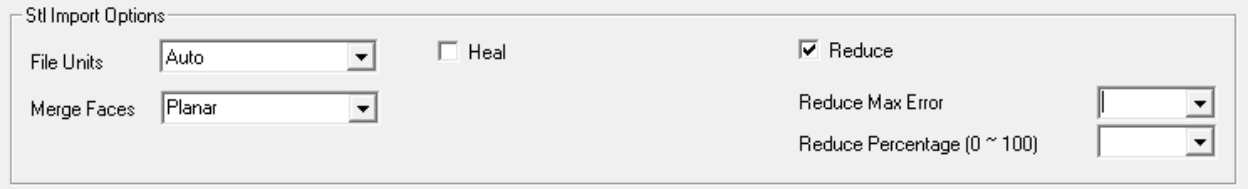
   The generic format specification for STL does not include units. When "**Auto** is chosen for the file units, the current Model units are normally used.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the solution type is HFSS SBR+, the STL import also includes options to **Import as Lightweight Faceted Body** and **Separate Disjoint Lumps**.



For STL files, you can choose to **Reduce** the number of facets in faceted objects while keeping the overall shape of the object.



Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

For details on these features and their use with Icepak, see Lightweight Geometry in Icepak.

- **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

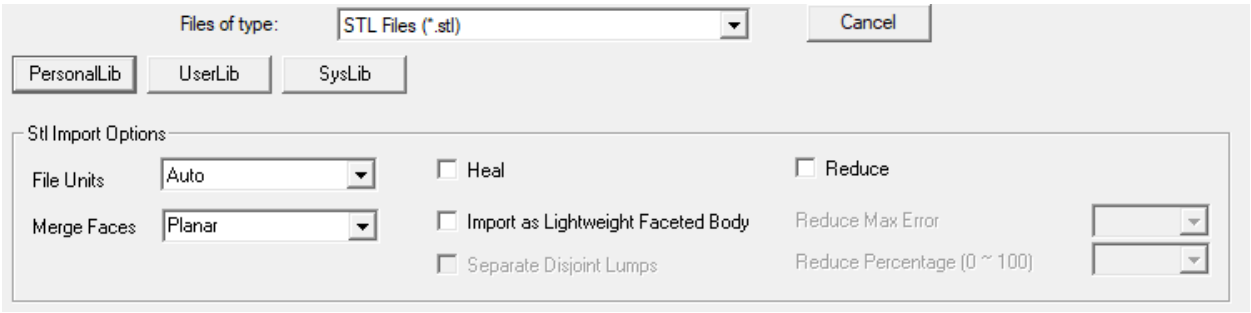
> **Note:**
>
> While objects created in Icepak can always be classed in the history tree as either a solid, sheet, or wire, some imported objects may have mixture of these. Icepak places such objects in an **Unclassified** folder in the history tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lightweight Geometry for Icepak Designs

Importing large stl files into Ansys Electronics Desktop can be time consuming. STL files contain triangulations of objects, and importing them to the modeler creates a face out of every triangle, and an edge out of every triangle edge. This creates a large number of faces and edges. which can slow down operations. The lightweight geometry feature gives you the option to import objects as one face, without edges or vertices in them. Internally, the modeler does not create parasolid bodies for these objects.

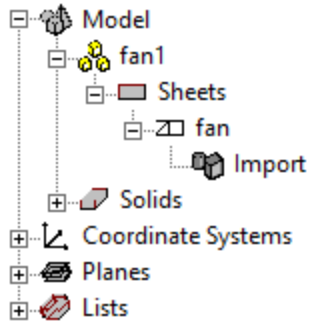
The Import File dialog for STL files contains several options.



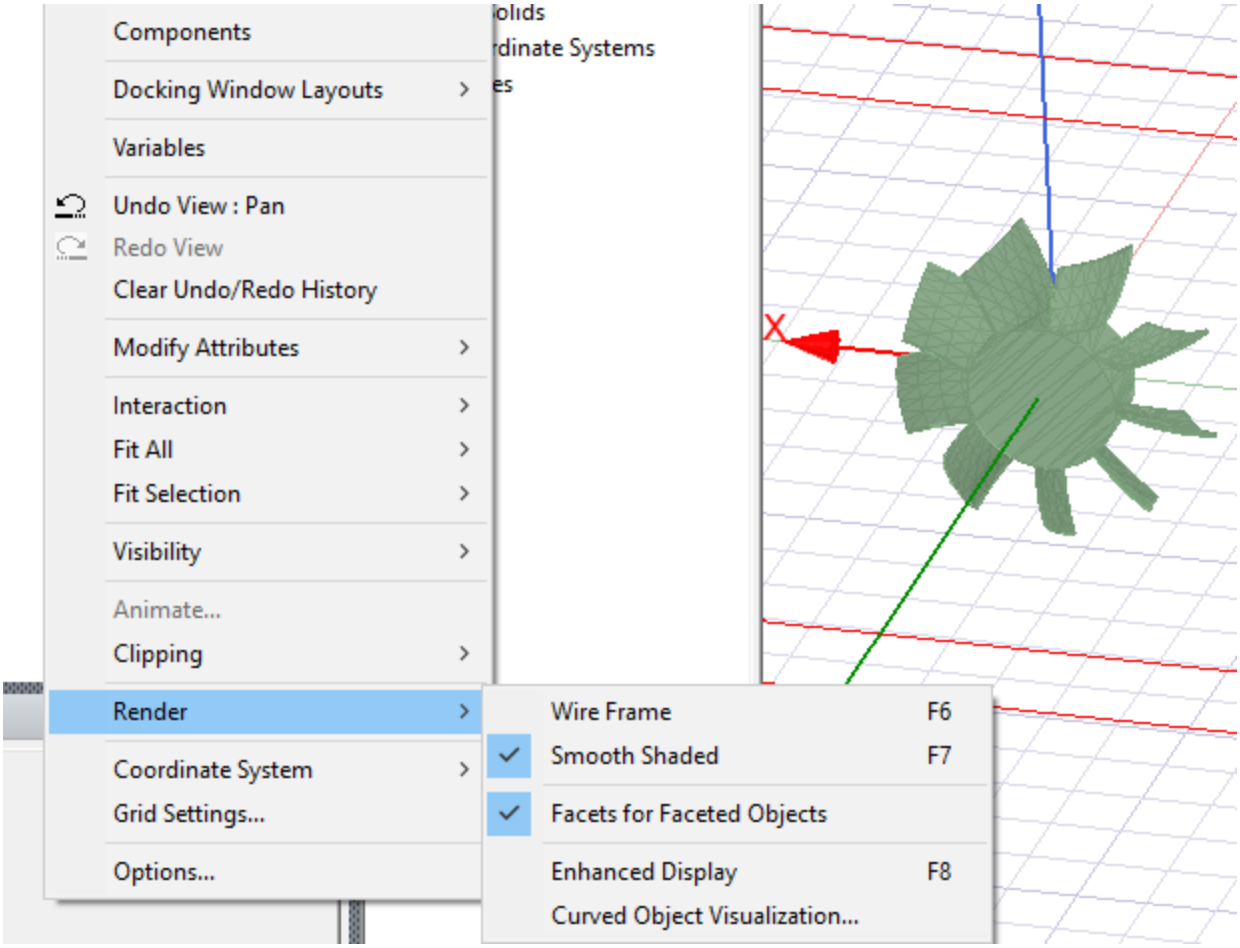
- **Heal** attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.
- **Import as Lightweight Faceted Body**, if checked, creates faceted bodies from imported objects and enables the Separate Disjoint Lumps option.
- **Separate Disjoint Lumps** option, if checked, create multiple parts if the .stl files have objects having disjoint lumps.
- **Reduce** enables lets you reduce the number of facets in faceted objects while keeping the overall shape of the object. Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

Once imported, faceted objects have distinct icons in history tree. They can also be grouped together, as shown in the following figure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The option **Facets for Faceted Objects** is provided in **View>Render** menu. When this option enabled, the modeler renders edges of triangle contained in the faceted bodies. These edges cannot be selected. See the following figure:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permitted Modeler Operations on Faceted Objects

- Arrange operations such as Move, Rotate, Mirror.
- Duplicate along line, around axis, with mirror.
- Scale.
- Copy-Paste.
- Create object CS - you can snap to vertices of facets to create object CS.
- Create face CS - you can pick any point, determine the triangle on which the picked point lies, and store the position relative to the triangle vertices.
- Create 3D components out of faceted objects.
- Selecting faces for assigning boundaries, coordinate systems, excitations or hybrid regions. **Note:** the entire object boundary is selected.

Other Characteristics of Faceted Objects

- Solid and sheet objects are identified in the History tree.
- Object Measure dialog shows the correct volume, and surface area of faceted objects.
- Faceted objects are skipped during intersection checks and containment checks done during geometry validation.
- Faceted objects cannot be exported from the Modeler.

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):

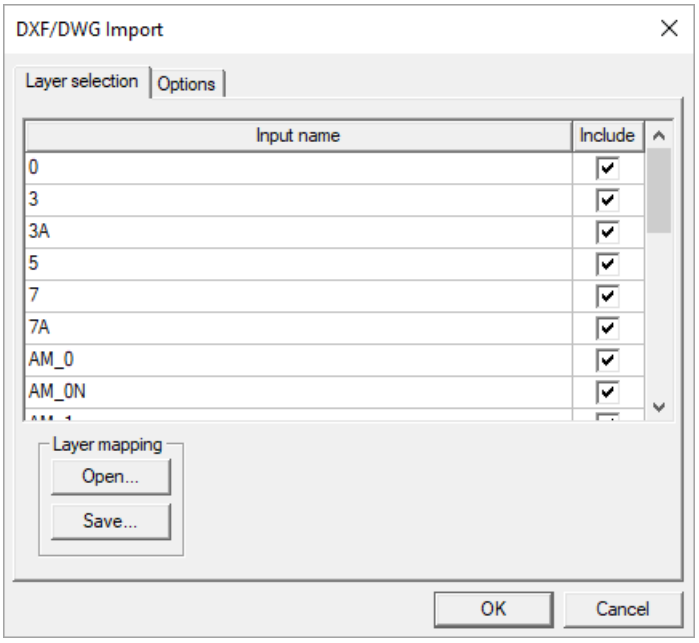
1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



   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

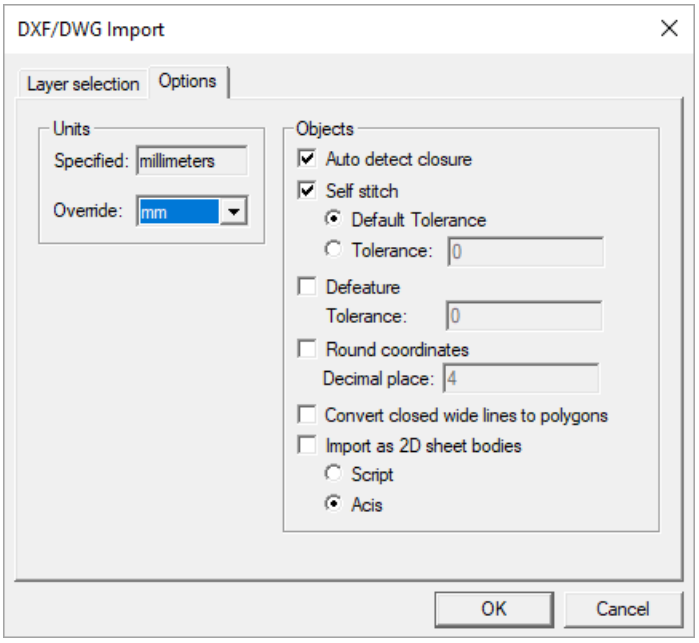
MIDLAYER green 500 200

TOPLAYER blue 1000 200
```

6. Click **Options**.

   The **Options** tab displays.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



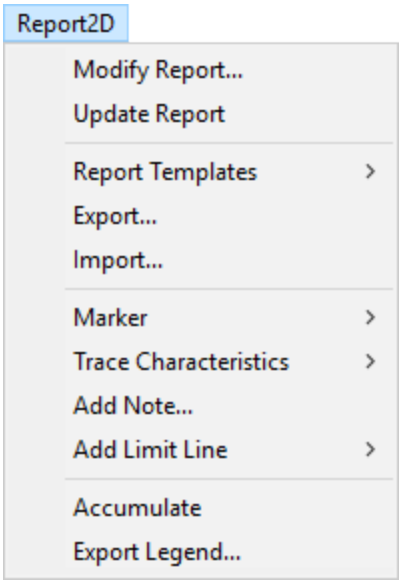
7. In the **Units** area, the **Specified** field displays the layout units in the file for import. Use the **Override** drop-down menu to select a different unit of measure.
8. In the **Objects** area, use the check boxes to fine tune the import:
    - **Auto-detect closure** causes polylines to be 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.
    - **Defeature 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.
    - **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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The file is imported into the active **Layout** window.

Importing Plot Data

When a report is open in Ansys Electronics Desktop, the **Report2D** menu appears.



> **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.

Ansys Electromagnetics Suite 2026 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, 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.

Importing IDF Files

Icepak and HFSS 3D Layout can import Intermediate Data Format (IDF) files that have been exported from an ECAD package. The IDF is a neutral file format that is supported by many ECAD packages (for example, Board Station from Mentor Graphics and Visula from Zuken) and some mechanical CAD packages (for example, PTC Creo, SDRC/I-deas, and Unigraphics). You can import IDF 2.0 and IDF 3.0 files into Icepak or HFSS 3D Layout. Note that IDF 4.0 files cannot be read by either Icepak or HFSS 3D Layout. This is because, currently, ECAD and CAD packages do not export to the IDF 4.0 format, due to the changes from IDF 3.0 to IDF 4.0.

Overview of Importing IDF Files

IDF 2.0 consists of two files, a board file and a library file. IDF 3.0 may consist of two or three files: a board file, a library file, and (in some cases) a control file. The board file contains information about the outline of the board, its thickness, and the positions (but not the sizes) of the components on the board. The library file contains information about the components of the board: their sizes, and their thermal and electrical properties. Icepak and HFSS 3D Layout needs both of these files to obtain complete thermal information about the board and the components. The control file contains information about the placement of boards in a rack.

Note that IDF 2.0 is a simpler format than IDF 3.0. For example, information such as junction-to-case thermal resistances and operating power information can be found only in an IDF 3.0 file.

When importing an IDF file, you can choose to filter out devices by height, power, or type, while optionally choosing to retain the filtered devices as non-model objects. Board conductivity properties, such as board thickness and materials assignment, can be specified and are used to calculate the orthotropic conductivity for the board. The IDF import workflow also allows you to specify a cut-off height for devices when the default height in the IDF file is large. And, if desired, you can select to import the board and devices as simple rectangular objects.

Import an IDF File

Follow this procedure to import an Intermediate Data Format (IDF) files that have been exported from an ECAD package.

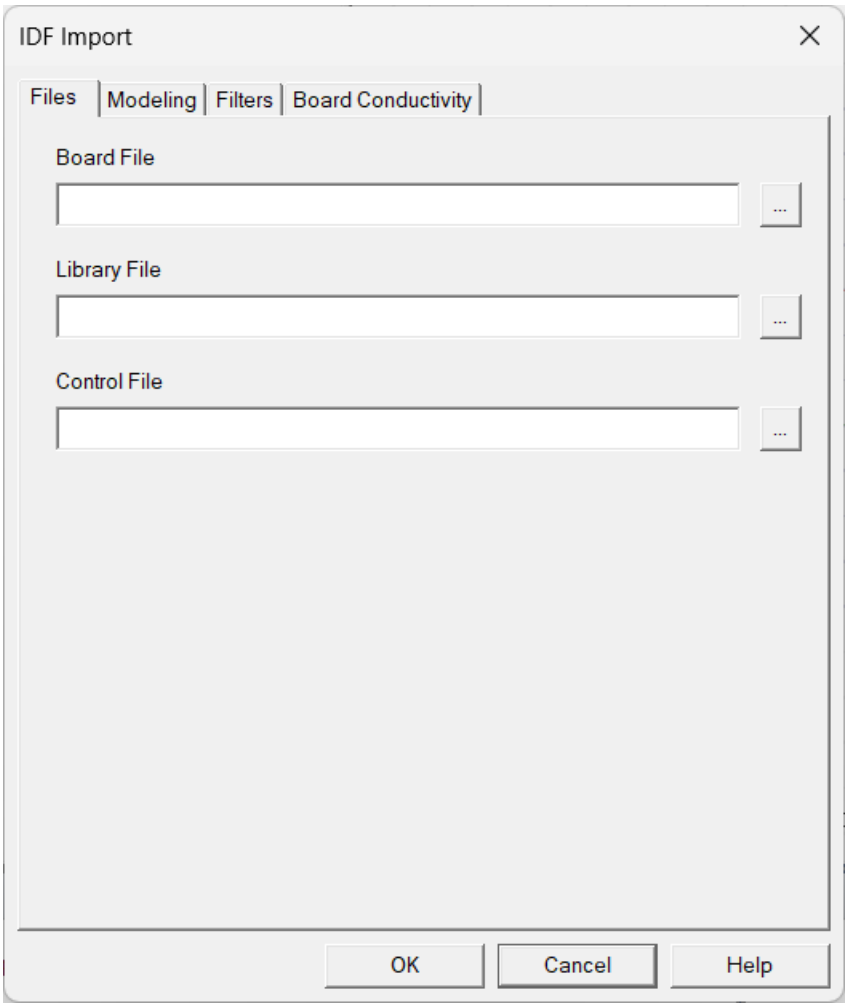
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Icepak** menu, select **Import IDF**.

   Note:

   The **File** > **Import** > **IDF** option creates a new HFSS 3D Layout project and is not relevant in Icepak designs.

2. On the **IDF Import** dialog box **Files** tab, click the button next to the **Board File** field.



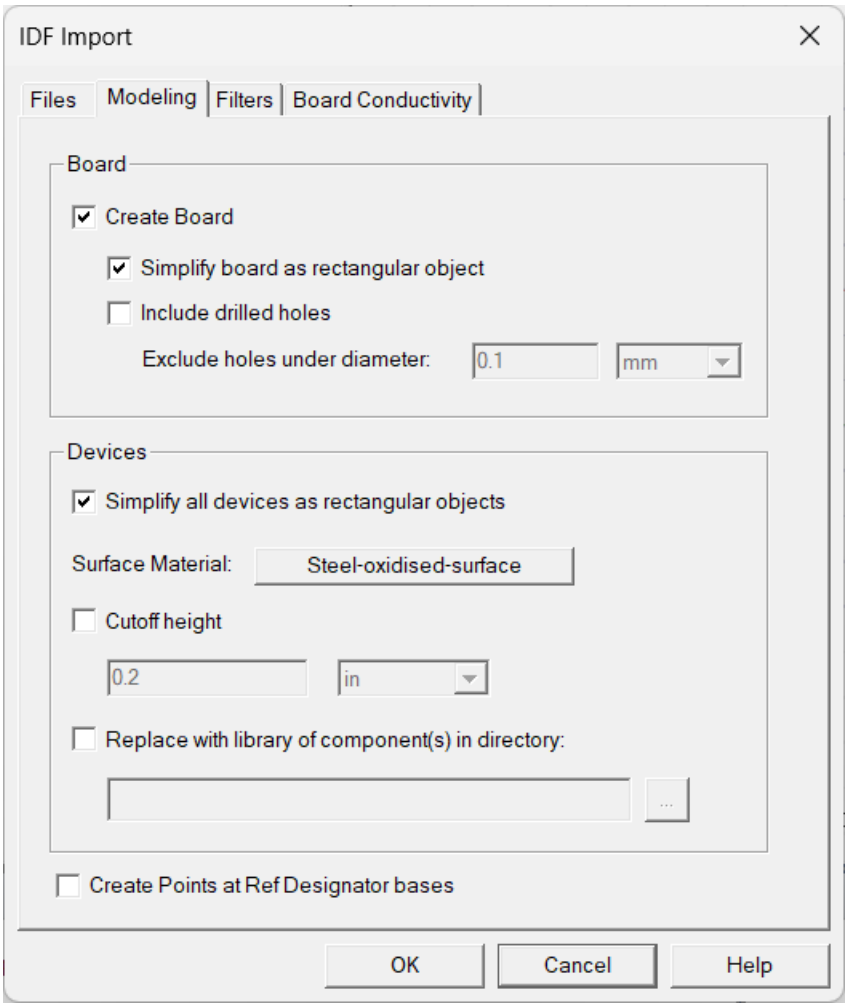
3. In the **Open** dialog box, browse to the location of the IDF files, select the board file, and click **Open**. If available, the **Library File** and **Control File** fields are automatically populated.
4. If needed, click the button next to **Library File** and **Control File** to select these files.
5. On the **Modeling** tab, do the following if needed:
   - Select **Create Board** to include board geometry. To import the board as a simple rectangular geometry, select **Simplify board as a rectangular object**. To import

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the board in its original polygonal shape, leave this field unchecked.

- Select **Include drilled holes** to import the board geometry with the drilled holes with air assigned as the material property. To filter out small holes that are not of interest, enable **Exclude holes under diameter** and specify the threshold diameter value and unit.
- Select **Simplify all devices as rectangular objects** to import all devices as simple rectangular pieces of geometry.
- Click the **Surface Material** button to open the **Select Definition** dialog box and select a surface material to assign to the devices.
- Select **Cutoff height** and enter a maximum height. Components with a greater height are cut off to the specified height.
- If needed, select **Replace with library of component(s) in directory:** to replace the imported model components with those components in the specified directory. A component in the specified directory replaces an imported model component when there is a match between the modified *partNumber* name in the IDF file and the component file name in the specified directory. Supported file types include .a3dcomp, .jptd, and JEP30 .xml. Part number names are modified to an Electronics Desktop group and part name using valid characters (letters, numbers, and underscores). Invalid characters are replaced by underscores.
- Select **Create Points at Ref Designator bases** to generate points in the 3D modeler window and History tree for each reference designator.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

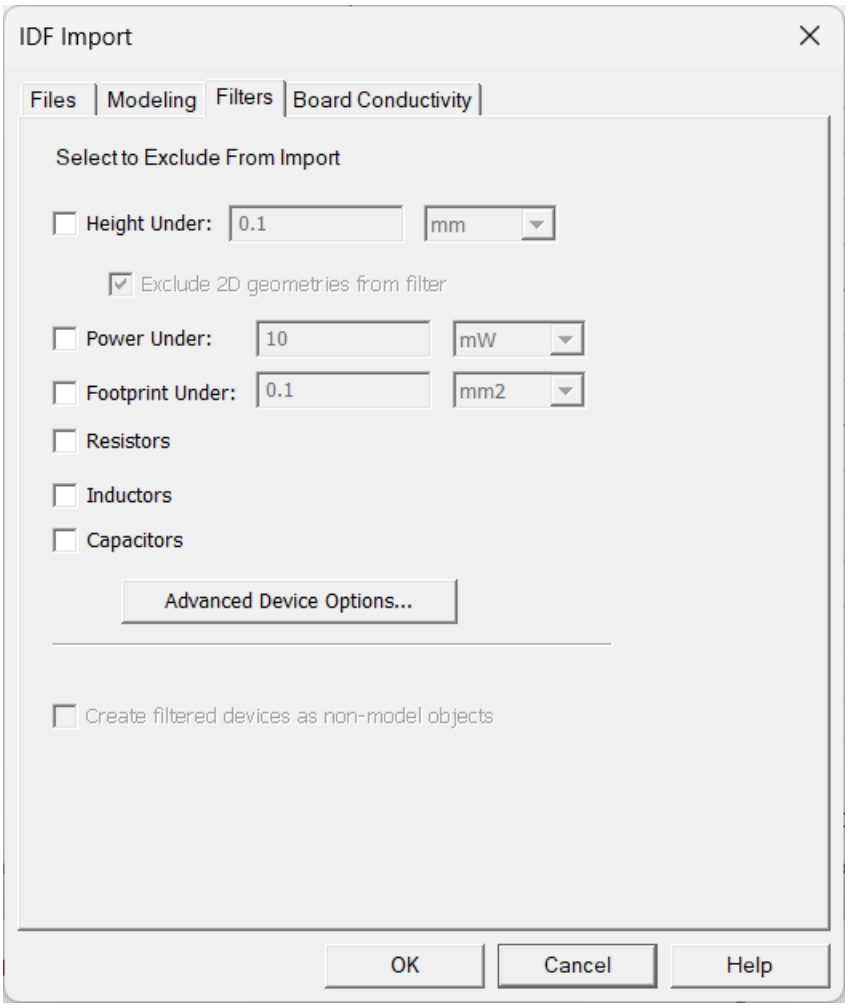


6. On the **Filters** tab, select the check boxes of the filters you want to apply to the import.
   - For **Height Under**, **Power Under**, and **Footprint Under**, specify the minimum height, power, and footprint of the components to be imported.

     > **Note:**
     >
     > If needed, enable **Exclude 2D geometries from filter** to ensure that two-dimensional geometries, such as a 2D source, are imported.

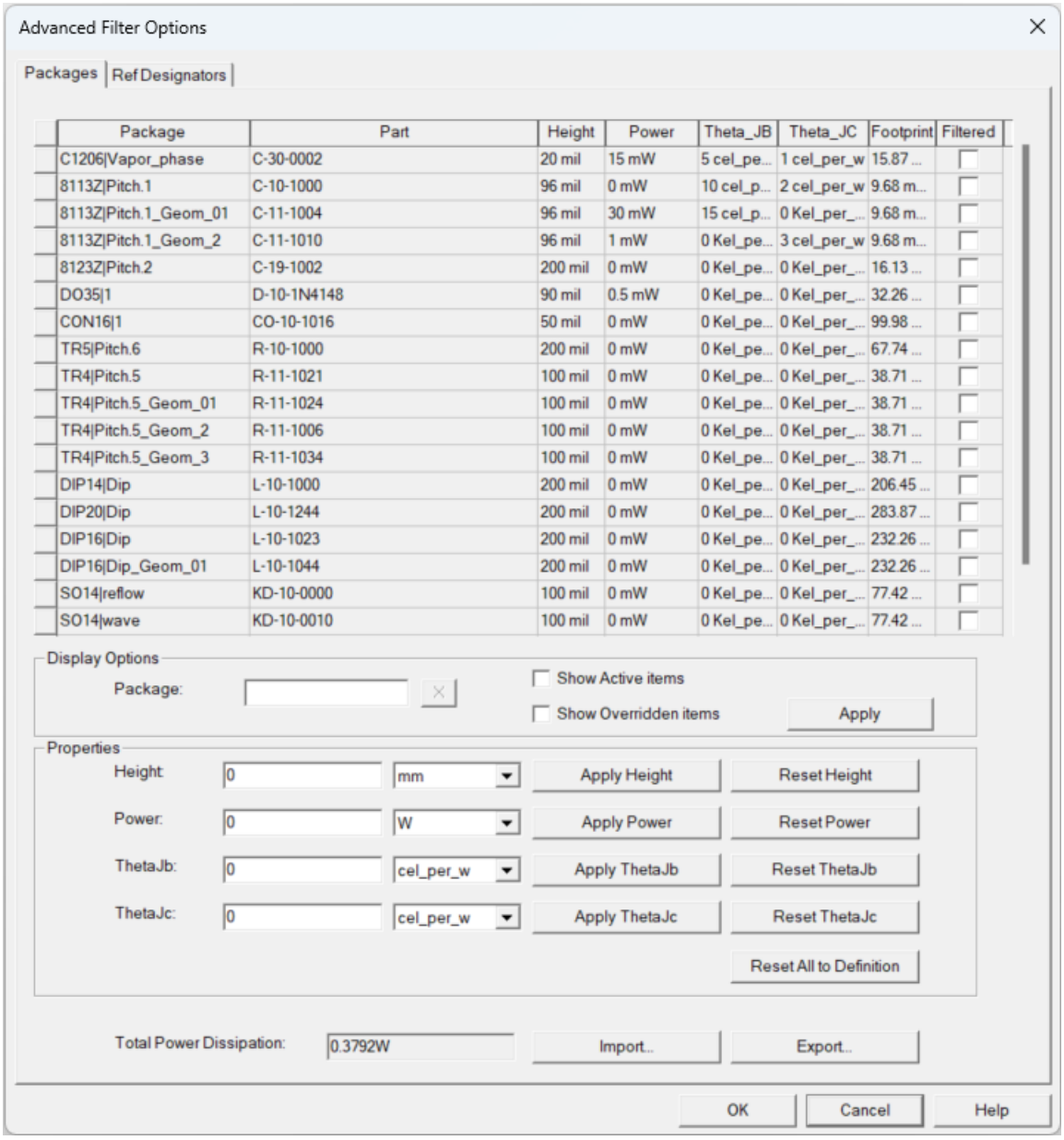
   - If **Resistors**, **Inductors**, or **Capacitors** is selected, the corresponding components are not imported.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. If needed, click **Advanced Device Options** to override the thermal properties of devices and enable advanced filtering options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The table at the top contains geometric and thermal model information of each package as available in the IDF file. The **Packages** tab contains per package data, while the **Ref Designator** tab contains data for all instances of each package. Below the table are the **Display Options** and **Properties** editing area.
- The **Packages** tab lists the **Package** name, the unique **Part** name, and the **Footprint** area, as input in the ECAD tool. These values can only be changed in the ECAD tool. Note that, other than the board, all objects in the IDF file, including logos and mechanical parts, are treated as a package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The remaining fields in the **Packages** tab are editable: **Height**, **Power**, **Theta_JB**, and **Theta_JC**. To edit any of these fields, first select the row corresponding to a package. Then, in the **Properties** area, enter new values for the **Height**, **Power**, **ThetaJb**, or **ThetaJc** fields. Finally, click the corresponding **Apply** button to update the value. The **Reset** buttons will revert the field values to those in the IDF file.
- The table can be sorted in ascending or descending order with any of the column titles as the sorting criterion, except the **Filtered** column. For example, toggle on the **Footprint** column title to sort the entire table in ascending or descending order of the package footprint areas.
- Packages selected in the **Filtered** column will not be imported into Icepak. Toggling the **Filtered** column title selects or deselects all packages.
- The **Ref Designators** tab lists all the instances of each package. This tab contains the additional information: reference designator under **Ref Des**, component **Placement** as the **Top** or **Bottom** side with respect to the board, and **Model** set as solid **Block** or **2R Network**.
- The **Ref Designators** tab allows you to change the power values in any of the instances of a given package, while the unchanged instances follow the inputs given in the **Packages** tab.

  > **Note:** Overridden height, power, theta jb, and theta jc values appear in orange text. To restore all original values, click **Reset All to Definition**.

- In the **Display Options**, the table in the **Packages** tab can be reduced to show only the active (unfiltered) packages. This reduces the list to the active packages in the **Ref Designator** tab as well. The same options also exist in the **Ref Designators** tab and similarly reduce the **Packages** table that is displayed. Additional filter options are listed in the table below.

Display Options	
Package	Enter a string of letters and/or numbers to displays rows containing those letters in the **Package** cell.
Show Active Items	Enable **Show Active Items** to hide rows with **Filter** enabled.
Show Overridden Items	Enable **Show Overridden Items** to show only rows whose height, power, thetajb, or thetajc properties have been overridden in the **Properties** area.

- To filter the items displayed in the Advanced Device Options table by device types, select the device type. The table below shows all the display filtering options by device types.

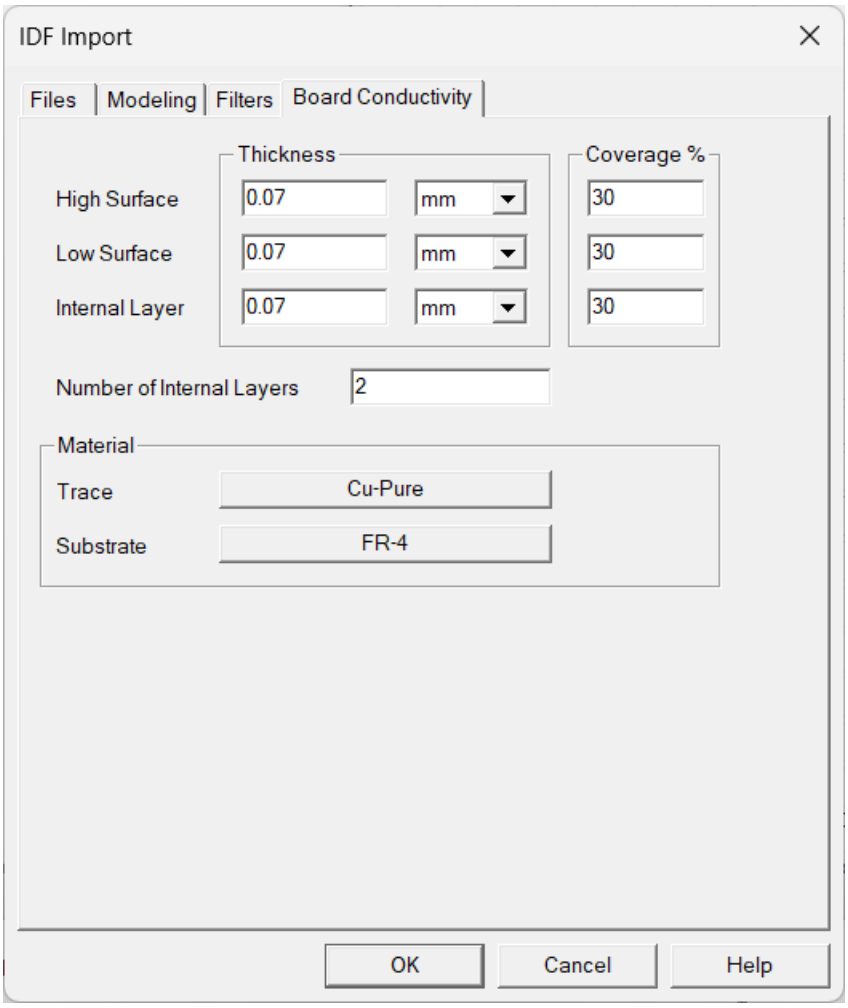
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Options	
Package	Enter a string of letters and/or numbers to displays rows containing those letters in the **Package** cell.
Ref Designator	Enter a string of letters and/or numbers to displays rows containing those letters in the **Ref Des** cell.
R, L, C, Others	Enable **R**, **L**, and/or **C** to display rows beginning with the selected letter in the **Ref Des** cell. Enable **Others** to display rows beginning with letters other than R, L, or C.
Top **Bottom**	Enable **Top** and/or **Bottom** to display rows based on the **Placement** of the device.
Apply	Click **Apply** to filter by the selected display options.
Show Active Items	Enable **Show Active Items** to hide rows with **Filter** enabled.
Show Overridden Items	Enable **Show Overridden Items** to show only rows whose power properties have been overridden.

- The **Total Power Dissipation** field below the **Properties** area shows the sum of power dissipation of all the component instances, which may be verified in the **Ref Designators** tab.
- The **Export** option allows saving all the changes made here into a tab-delimited (.tab) text file or a comma-separated value (.csv) spreadsheet file. If repeating this in future and the same inputs are desired, this file may be re-imported using the **Import** tab. Once exported, the files may be edited as long as the format remains the same, to enable a modified import.
- When finished, click **OK** to close the **Advanced Device Options** dialog box.

8. If you enabled **Create Board**, do the following on the **Board Conductivity** tab:
   - Enter **Thickness** and **Coverage%** for **High Surface**, **Low Surface**, and **Internal Layer**.
   - Enter the **Number of Internal Layers**.
   - Assign materials for **Trace** and **Substrate**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Click **OK** to import the model.

> **Note:**
>
> Icepak thermal boundary conditions are automatically assigned after import based on the IDF data.

Device Properties File Formats

Tab-delimited (.tab) and comma-separated value (.csv) files can be used to import device properties in the **Advanced Device Options** dialog box. The contents of the file formats are identical, but the application in which you can edit them differs.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: When exported, device property values and units are converted to SI units. When imported, they are converted from SI units to the values and units displayed in the **Advanced Device Options** dialog box.

The columns of data are arranged as in the following table.

Column 1	Column 2	Column 3	Column 4	Column 5
Instance Name	Power	Theta_jb	Theta_jc	Height

Tab-Delimited (.tab) Files

Use a plain text editor to create columns of data using the tab key.

```
C1      0.015   5       1       0.000508
C2      0.015   5       1       0.000508
C3      0.015   5       1       5.08e-05
C8      0.015   5       1       0.000508
C22     0.015   5       1       0.000508
C23     0.015   5       1       0.000508
C24     0.015   5       1       0.000508
C25     0.015   5       1       0.000508
C26     0.015   5       1       0.000508
C27     0.015   5       1       0.000508
C28     0.015   5       1       0.000508
C29     0.015   5       1       0.000508
C30     0.015   5       1       0.000508
```

Comma-Separated Value (.csv) Files

Use a spreadsheet-editing application to enter data into the appropriate columns.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B	C	D	E
1	C1	0.015	5	1	0.000508
2	C2	0.015	5	1	0.000508
3	C3	0.015	5	1	5.08E-05
4	C8	0.015	5	1	0.000508
5	C22	0.015	5	1	0.000508
6	C23	0.015	5	1	0.000508
7	C24	0.015	5	1	0.000508
8	C25	0.015	5	1	0.000508
9	C26	0.015	5	1	0.000508
10	C27	0.015	5	1	0.000508
11	C28	0.015	5	1	0.000508
12	C29	0.015	5	1	0.000508
13	C30	0.015	5	1	0.000508

Importing IDX Files

Icepak can import Incremental Design eXchange (IDX) files that have been exported from an MCAD or ECAD package. IDX is an XML-based format containing MCAD and ECAD data that allows you to exchange incremental design revisions between ECAD and MCAD applications. IDX import in Ansys Icepak only supports MCAD data.

Overview of Importing IDX Files

The initial IDX file is referred to as the baseline file. Revisions to the design are included in change files. Currently, Ansys Icepak supports the import of IDX baseline files, which generates the design geometry and assigns boundary conditions.

> **Note:** Board cutouts are not supported.

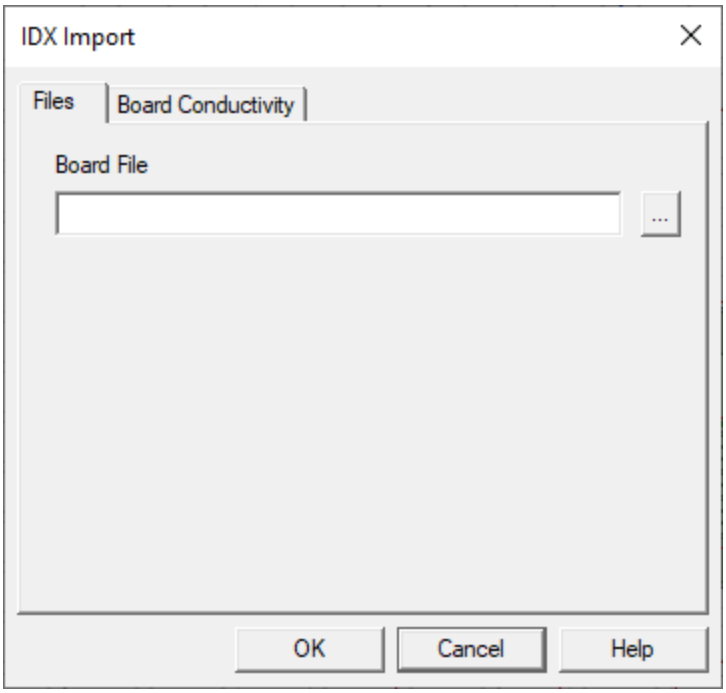
Import an IDX File

Follow this procedure to import an Incremental Design eXchange (IDX) file.

> **Note:** IDX import currently does not support filtering and modeling options.

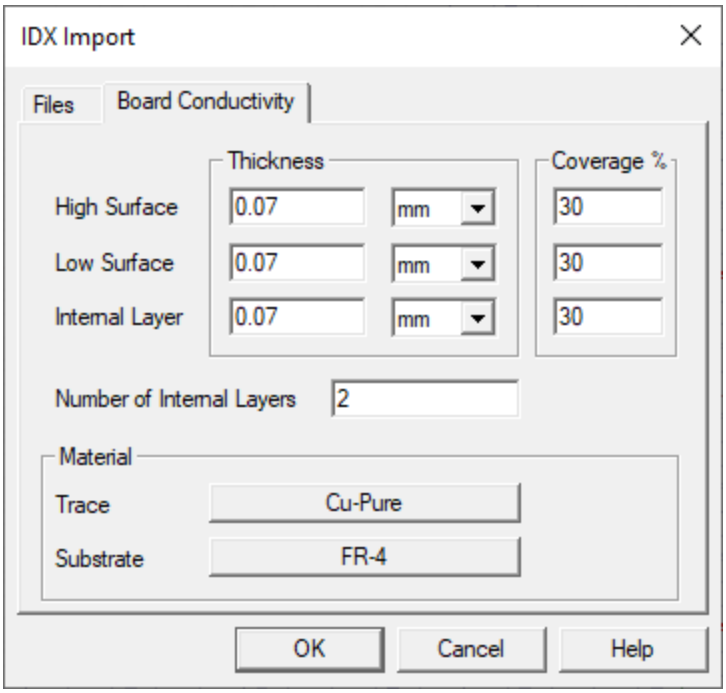
1. From the **Icepak** menu, select **Import IDX**.
2. In the **IDX Import** dialog box **Files** tab, click the button next to the **Board File** field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the **Open** dialog box, browse to the location of the IDX file, select the board file, and click **Open**.
4. Do the following on the **Board Conductivity** tab:
   - Enter **Thickness** and **Coverage%** for **High Surface**, **Low Surface**, and **Internal Layer**.
   - Enter the **Number of Internal Layers**.
   - Assign materials for **Trace** and **Substrate**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



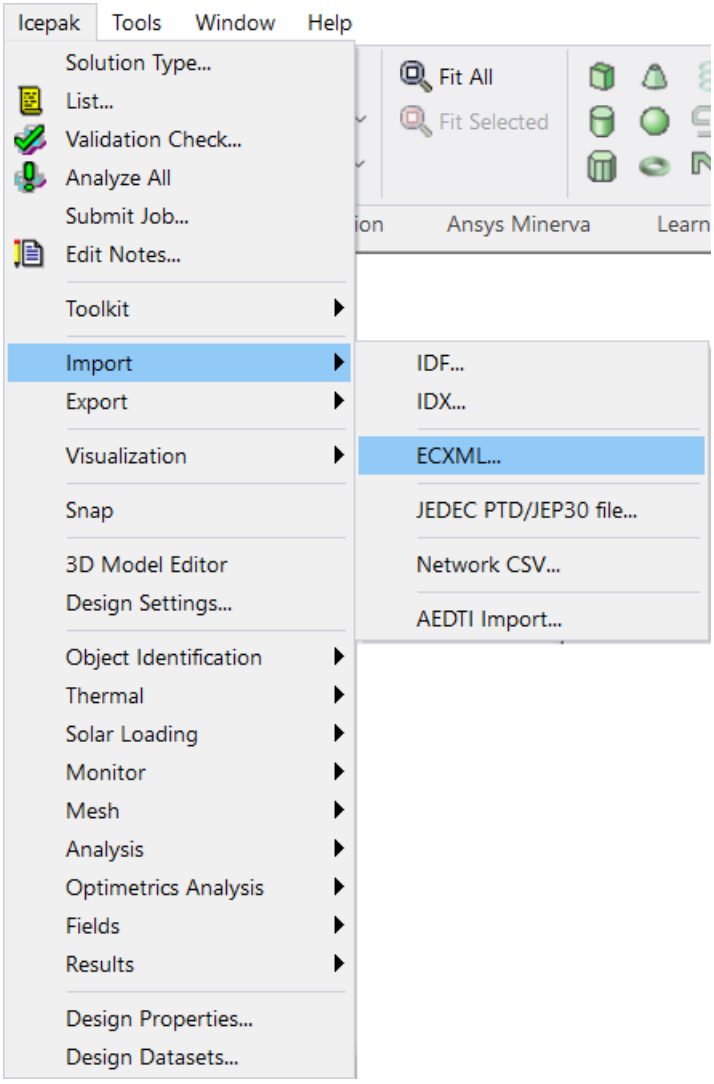
5. Click **OK** to import the model.

> **Note:**
>
> Icepak thermal boundary conditions are automatically assigned after import based on the IDX data.

Importing Electronics Cooling XML Projects

Icepak can import Electronics Cooling XML (EC XML) files. EC XML files are neutral format that can contain geometry, boundary condition, material, and monitor point information. See EC XML File Format for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> You can also access the import menu by right-clicking on the design in the **Project Manager**.

Importing JEDEC JEPJPTD and JEP 30 Files

JEDEC standard JEP30-T100A establishes the requirements for exchanging electronic part thermal data. Two types of thermal Network Models are supported; DELPHI-Model, as defined in JESD15-4 and 2-ResistorModel, as defined in JESD15-3. Both models involve the definition of a nodal thermal network, which can be exported into Icepak using *.xml or *.jpd files, respectively for JEDEC standard JEP30-T100A and for its predecessor known as Package

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Data Interchange File Format (PTD IFF). For thermal Network Models import/export, these files contain similar information formatted in slightly differing schemas.

The *.xml or *.jpd files contain the following information:

- Model ID name, which can be according to ECAD component part name or part number
- Units for thermal data
- Model geometry specifications: package (Package Body Center to the Origin), internal and external nodes definitions (Volume, face areas)
- Network specifications: model type, nodal power and capacitance, resistance interconnection schemas

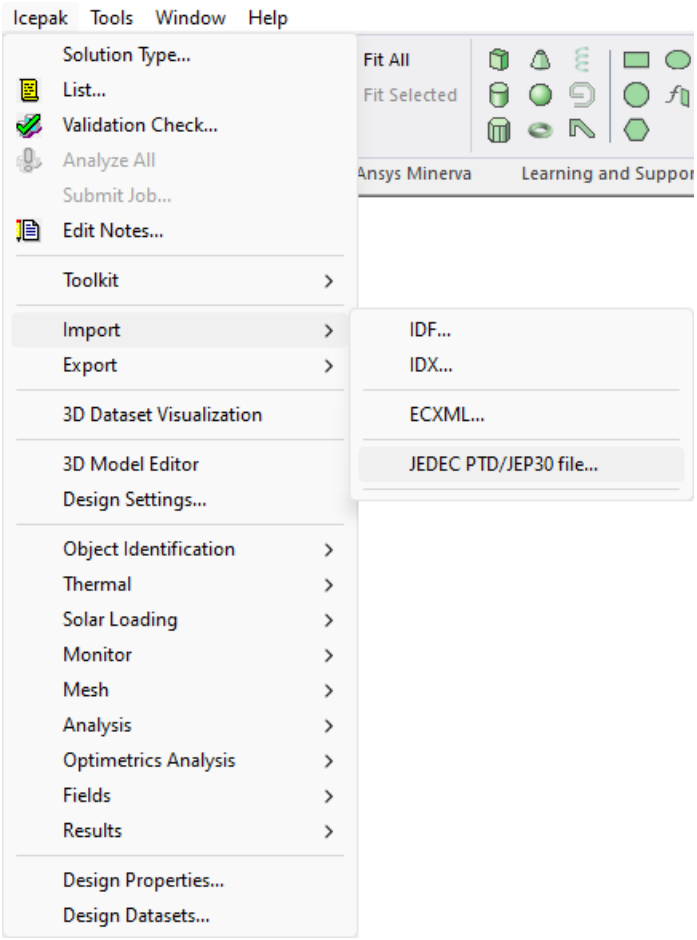
For detailed schema information regarding these file formats, see PTD IFF File Format and JPTD File Format.

Additionally, *.xml or *.jpd files can be capitalized into library and these ones could reuse for thermal simulation at PCB level.

Based on ECAD component part name or part number as model name, *.xml or *.jpd files can be automatically placed on PCB using Icepak IDF file import.

> **Note:**
>
> For more information about JEDEC standards, visit www.jedec.org.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> You can also access the import menu by right-clicking on the design in the **Project Manager**.

PTD IFF File Format

The following schema structure illustrates a truncated definition of a chip thermal model in a JPTD file.

Units and Network Models

The schema defines the part, units, partName, and networkModels. Within networkModels, the Delphi model node geometry and thermal properties are defined.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<JEDEC_PTD_IFF version="1.0">
    <part>
        <units>
            <length>m</length>
            <power>W</power>
            <mass>kg</mass>
            <resistance>K/W</resistance>
            <heat_capacity>J/kg-K</heat_capacity>
            <temperature>K</temperature>
            <heat_transfer_coefficient>W/K-m2</heat_transfer_coefficient>
        </units>
        <partName>CTM network.1</partName>
        <networkModels>
            <DELPHIModel>
                <name>network.1</name>
                <coreNetwork>
                    <nodes>
                        <node name="Junction" power="1.5">
                            <nodeVolumes>
                                <cuboidalNodeVolume>
                                    <position x="0.0875" y="0.0875" z="0.0875"/>
                                    <size x="0.025" y="0.00215" z="0.025"/>
                                </cuboidalNodeVolume>
                            </nodeVolumes>
                            <mass>2.0</mass>
                            <capacity>7.89165E-4</capacity>
                        </node>
```

Resistances

Below the definition of the Delphi model nodes, the thermal resistance properties are defined.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                </nodes>
                <resistances>
                    <resistance fromNode="int1" toNode="Junction" value="80.1777"/>
                    <resistance fromNode="int2" toNode="Junction" value="100000"/>
                    <resistance fromNode="int2" toNode="int1" value="2.0803"/>
                    <resistance fromNode="top_inner" toNode="Junction" value="20.5058"/>
                    <resistance fromNode="top_outer" toNode="Junction" value="65.8042"/>
                    <resistance fromNode="top_outer" toNode="int2" value="23.4659"/>
                    <resistance fromNode="top_outer" toNode="top_inner" value="100000"/>
                    <resistance fromNode="bottom_inner" toNode="int1" value="1.5861"/>
                    <resistance fromNode="bottom_outer" toNode="int2" value="3.4763"/>
                </resistances>
            </coreNetwork>
        </DELPHIModel>
    </networkModels>
  </part>
</JEDEC_PTD_IFF>
```

JEP 30 File Format

The following schema structure illustrates a truncated definition of a chip thermal model in a JEP 30 file.

Units

The schema defines the ThermalSection and ThermalFamily-Array. Within ThermalFamily, the JEP 30 schema defines the model ID and units first.

```
<?xml version="1.0" encoding="UTF-8"?>
<ThermalSection>
  <ThermalFamily-Array>
        <ThermalFamily>
      <ID>CTM MN7_92089118</ID>
      <ThermalData>
        <UnitsForThermalData>
          <DimensionUOM>mm</DimensionUOM>
          <NodalMassUOM>kg</NodalMassUOM>
          <PowerUOM>W</PowerUOM>
          <SpecificHeatCapacityUOM>J/kg-K</SpecificHeatCapacityUOM>
          <ThermalCapacitanceUOM>J/K</ThermalCapacitanceUOM>
          <ThermalResistanceUOM>K/W</ThermalResistanceUOM>
          <TemperatureUOM>K</TemperatureUOM>
        </UnitsForThermalData>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network Models

Below the definition of units, the NetworkModel schema defines the node geometry for face nodes and power for internal nodes.

```
</UnitsForThermalData>
<NetworkModels>
  <DELPHIModel>
    <Name>MN7_92089118</Name>
    <PackageBodyCenterOffset-to-Origin>
      <x>0</x>
      <y>0</y>
      <z>0</z>
    </PackageBodyCenterOffset-to-Origin>
    <CoreNetwork>
      <Node-Array>
        <Node>
          <Name>TOP</Name>
          <NodeFace-Array>
            <RectangularNodeFace>
              <Position>
                <x>0.08625</x>
                <y>0.0112481</y>
                <z>0.001505</z>
              </Position>
              <xyPlane>
                <dx>0.0135</dx>
                <dy>0.0075</dy>
              </xyPlane>
            </RectangularNodeFace>
          </NodeFace-Array>
        </Node>
        <Node>
          <Name>JUNCTION1</Name>
          <Power>1.42</Power>
        </Node>
```

Resistance and Capacitance

Below the definition of the model nodes, the thermal resistance and capacitance properties are defined.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

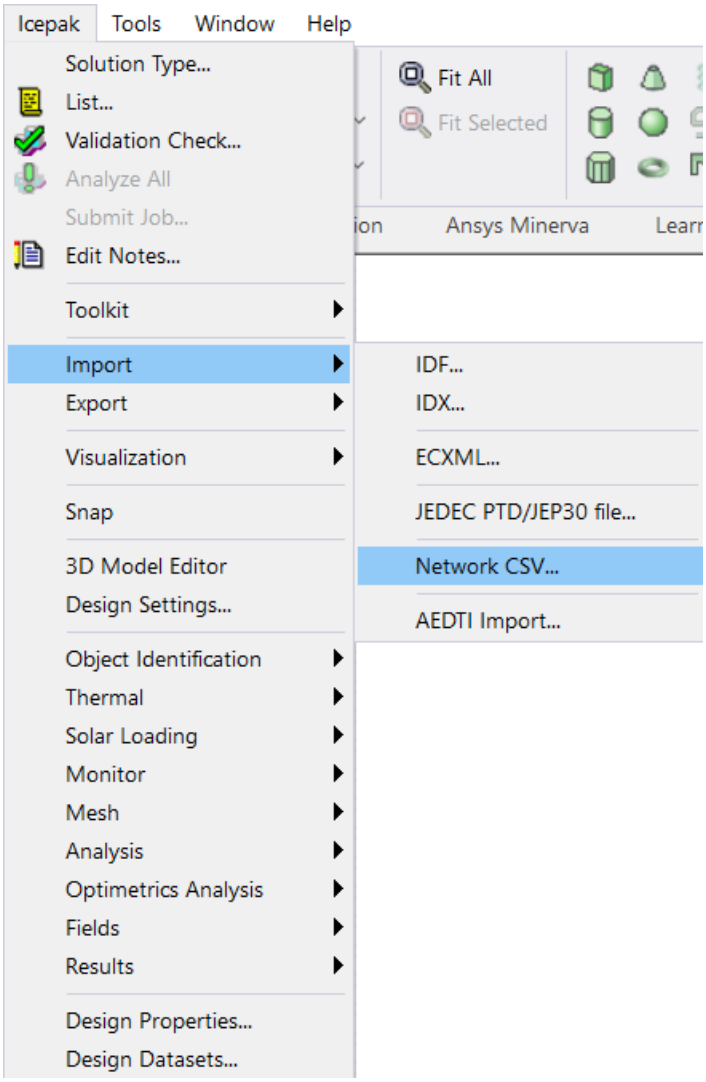
```
            </NodeFace-Array>
          </Node>
        </Node-Array>
        <ThermalResistance-Array>
          <ThermalResistance>
            <ThermalResistance>20.976337468893114</ThermalResistance>
            <FromNode>TOP</FromNode>
            <ToNode>BOTTOM</ToNode>
          </ThermalResistance>
          <ThermalResistance>
            <ThermalResistance>17.061333333333334</ThermalResistance>
            <FromNode>TOP</FromNode>
            <ToNode>BACK</ToNode>
          </ThermalResistance>
        </ThermalResistance-Array>
        <ThermalCapacitance-Array>
          <ThermalCapacitance>
            <FromNode>JUNCTION1</FromNode>
            <NodalMass>1</NodalMass>
            <NodalSpecificHeatCapacity>0.0011409504437841155</NodalSpecificHeatCapacity>
          </ThermalCapacitance>
        </ThermalCapacitance-Array>
      </CoreNetwork>
    </DELPHIModel>
  </NetworkModels>
</ThermalData>
</ThermalFamily>
</ThermalFamily-Array>
</ThermalSection>
```

Importing Network CSV Files

Icepak can import networks that you previously exported in a comma-separated value (.csv) file format. To import a network .csv file, select **Icepak** > **Import** > **Network CSV**. In the **Open** dialog box, navigate to the location of the .csv file, select it, and click **Open** to import the network geometry and boundary condition. See Network CSV File Format for the information included in the exported file.

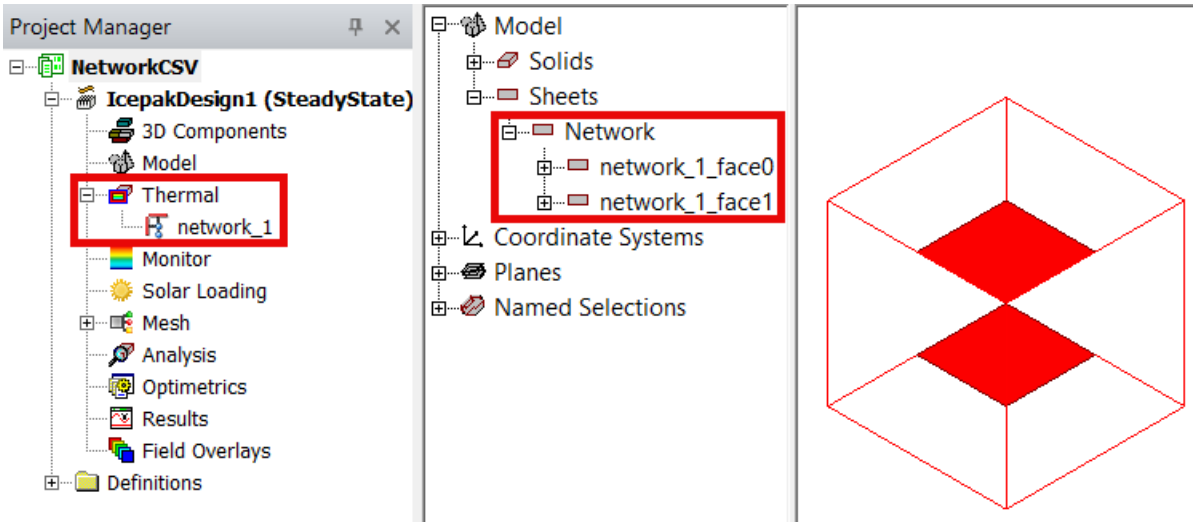
> **Note:**
>
> You can also access the import menu by right-clicking on the design in the **Project Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

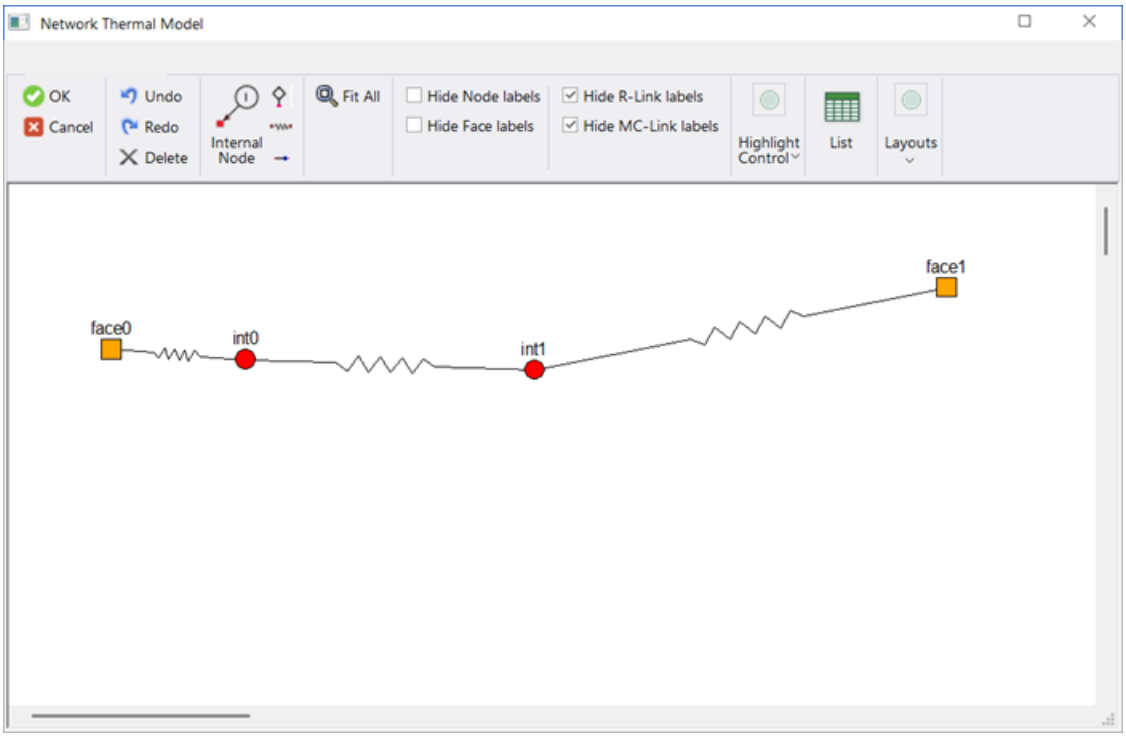


The network boundary condition appears in the **Project Manager** under **Thermal**. The geometry appears in the History Tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double-click the boundary condition to review the network nodes and links in the **Network Thermal Model** dialog box.

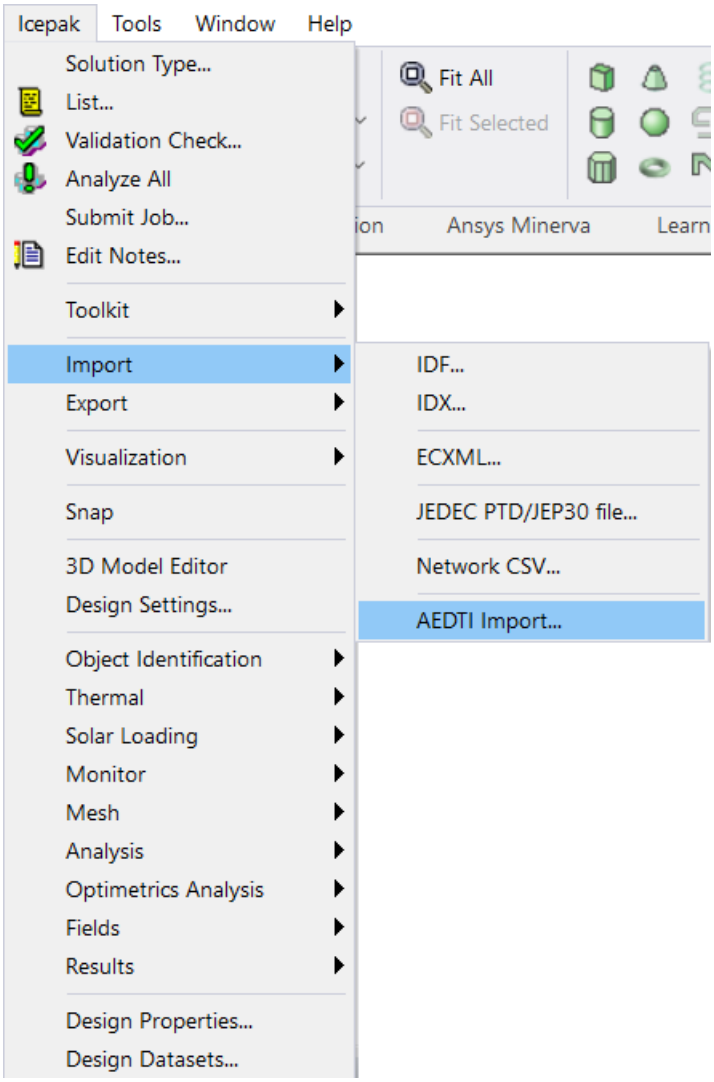


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing AEDTI Files

Icepak can import AEDTI files, which are generated by Ansys Discovery. AEDTI files contain geometry, boundary condition, and components. To import an AEDTI file, select **Icepak > Import > AEDT Import**.

> **Note:**
>
> You can also access the import menu by right-clicking on the design in the **Project Manager**. Alternatively, you can drag an .aedti file into Electronics Desktop's 3D Modeler window to open it.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AEDTI File Object Conversion

The following tables contain details about how Ansys Discovery Fluid Flow and Solid Thermal boundaries are converted in Ansys Electronics Desktop.

> **Important:**
>
> Limitations are displayed in the warning messages when exporting in Discovery.

Fluid Flow Boundary

Discovery	Electronics Desktop
Velocity	Opening
Flow velocity with swirl	2D fan
Fluid fan	3D fan
Fluid fan with constant pressure rise and gauge pressure	Opening
Rotating fluid zone	Fluid block with MRF enabled

Solid Thermal Boundary

Discovery	Electronics Desktop
Insulated	Adiabatic plate with emissivity values assigned to the surface material
Heat	2D source or 3D block
Convection	Wall for 2D assignments and block for 3D assignments
Wall	Stationary wall
Heat flux	Wall
Temperature	Source with fixed temperature
Radiation	Wall

Exporting Files

You can export the following types of files from Ansys Electronics Desktop projects:

- 2D model files
- 3D model files
- LTI ROM files

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- LPV ROM files
- ECXML files
- SIwave temperature files
- Graphics files
- Reports as data or graphics files

Exporting 2D Model Files

Icepak 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)**.
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 are listed below.

Type	**File Extension(s)**
ACIS SAB	*.sab (exported with version 5.0)
ACIS SAT	*.sat (exported with version 5.0)
AutoCAD	*.dxf, *.dwg
GDSII	*.gds
GL Transmission Format	*.gltf or *.glb (exported as version 2.0)
IGES	*.iges, *.igs (exported as version 5.1)
JT	*.jt (exported as version 10.0)
OBJ Files (for importing models	*.obj

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type to Ensight.)	File Extension(s)
Parasolid	*.x_t, *.x_b (exported as any version up to 38.0)
STEP	*.step, *.stp (exported as AP 242 Ed1)
STL	*.stl
VRML	*.wrl

To export a file to a 3D model format:

1. Click **Modeler** > **Export**.

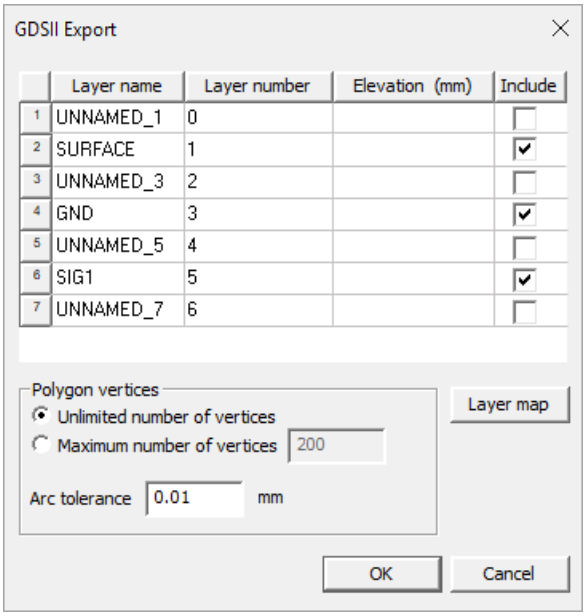
   The **Export File** window appears.

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:

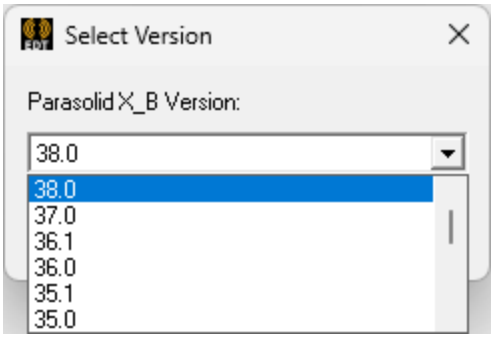


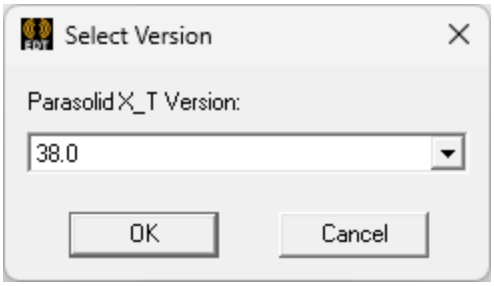
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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:





Use the drop-down menu to select a Parasolid version.

State-Space Characterization

State-space characterization generates linear time-variant reduced order models (LTI ROMs). State-space characterization can significantly reduce the model size and transient simulation time for LTI systems. This approach represents the thermal problem as a system that processes a set of input signals (or simply inputs) yielding a set of output signals (or simply outputs). For a typical thermal problem, the inputs are heat sources at certain locations, and the outputs are temperature rise at specified locations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When to Use State-Space Characterization

The state-space approach is generally valid for linear systems consisting of forced convection flow and insignificant radiative heat transfer.

Overview of State-Space Characterization

Run a Steady-State Analysis

1. Create a steady-state Icepak design.
2. After importing or building model geometry, assign thermal boundary conditions that contain parameters to which you'll assign variables the subsequent transient optimetrics analysis.

   > **Note:**
   >
   > For the steady-state analysis, ensure the **Value** is 0 for the parameters to which you intend to assign variables in the transient design.

3. Create **Temperature** monitor points to record the temperature of the boundary condition geometry.
4. Analyze the steady-state design.

Run a Transient Optimetrics Analysis

1. After the steady-state simulation is finished, copy the steady-state design and paste it in the project.
2. Change the **Solution Type** to **Transient**.
3. Create variables for the boundary condition parameters of interest for the optimetrics analysis.
4. Add a solution setup.
   - On the **General** tab, select **LTI ROM Defaults** from the solutions setup defaults drop-down list.
   - On the **Solver Settings** tab under **Import Options**, select **Start/Continue from a previously solved setup** and click **Setup Link**.
5. In the **Setup Link** dialog box next to **Source Design**, deselect **Use this design** and select the steady-state design you previously analyzed and click **OK**. If the steady-state design solves for flow and temperature, select **Frozen flow simulation**.
6. Click **OK** in the **Icepak Solve Setup** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. From **Optimetrics**, create a parametric setup using all of the variables you've created. Each row in the parametric table should have one variable with the same nonzero value and all other variables with a value of 0. The example below features a nonzero value of 5W.

*	FET1	FET2	FET3	FET4
1	0W	0W	0W	5W
2	0W	5W	0W	0W
3	0W	0W	5W	0W
4	5W	0W	0W	0W

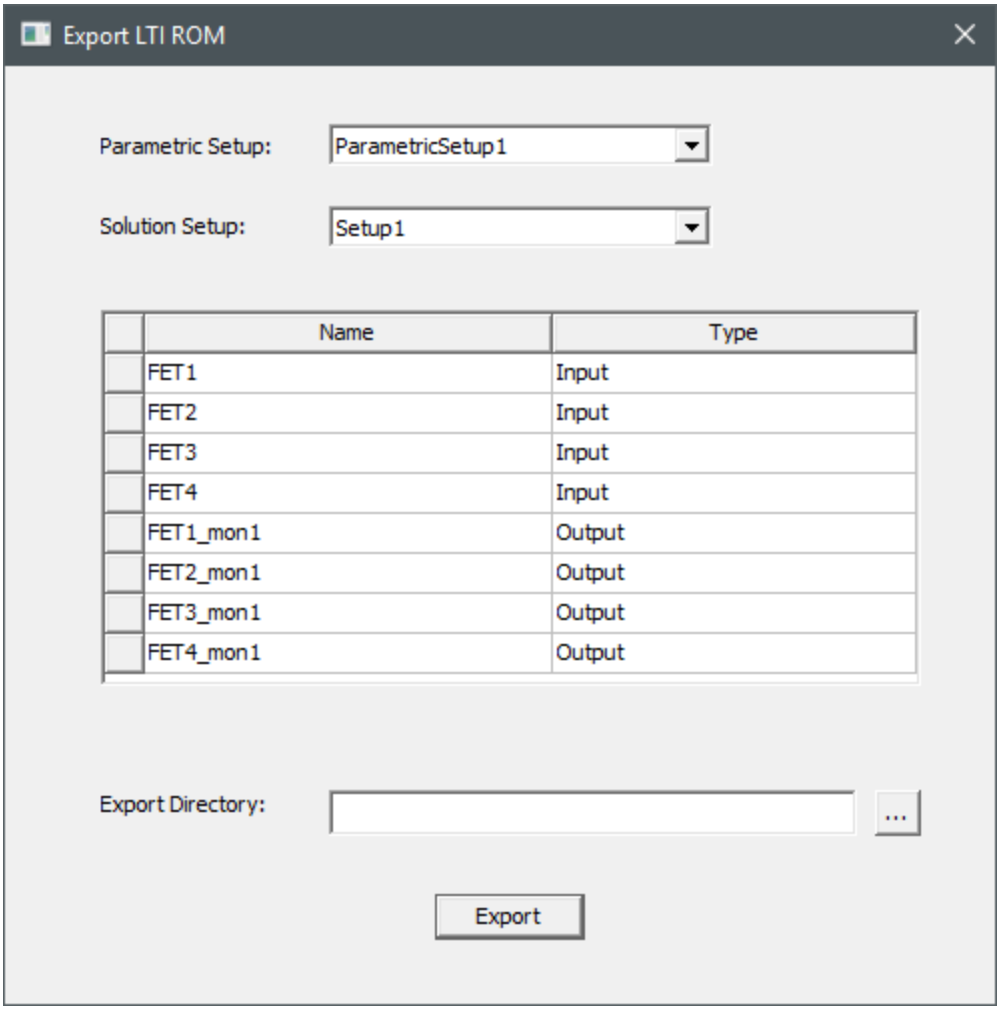
> **Note:**
>
> Use the **LTI ROM Parametric Setup** toolkit to quickly create a parametric setup for the transient design. See LTI ROM Parametric Setup for more information.

8. Analyze the transient parametric setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export LTI ROM

1. From the Icepak menu, select **Export ROM** > **LTI ROM**.



2. In the **Export LTI ROM** dialog box, ensure that the variables are set as inputs and the monitor points as outputs.

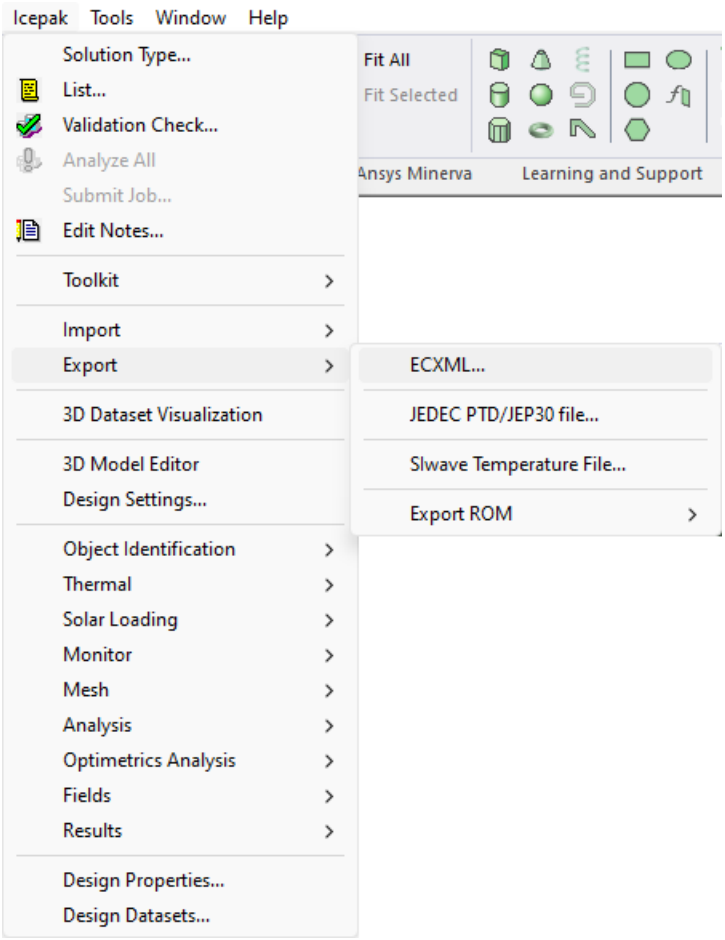
> **Note:**
>
> If a variable was not used in the setup, select **Unused** for its **Type**.

3. Select an **Export Directory**.
4. Click **Export** to generate the files required for TwinBuilder ROM setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting Electronics Cooling XML Files

Icepak enables you to export models in Electronics Cooling XML (EC XML) format that can be read by applications with EC XML compatibility. Native 3D Components (e.g., a fan) and geometry with a thermal boundary condition assigned are exported. See EC XML File Format for the information included in the exported file.



> **Note:**
>
> You can also access the export menu by right-clicking on the design in the **Project Manager**.

EC XML File Format

> **Note:** Only axis-aligned (orthogonal) geometries are exported.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Region

The following is the EC XML format for the region (computational domain).

```
<solutionDomain>
  <location>
    <x>0</x>
    <y>0</y>
    <z>0</z>
  </location>
  <size>
    <x>1</x>
    <y>1</y>
    <z>1</z>
  </size>
</solutionDomain>
```

Blower

Blower objects are not exported.

Plate

Note the following regarding exported plate objects:

- Adiabatic plates are exported with a thickness of 1 mm.
- Conducting plates with a **Thermal Specification** of **Conductance**, **Thermal Resistance**, or **Thermal Impedance** are exported with a specification of **Thickness** and given a 1 mm thickness.

The following is the EC XML format for a plate boundary condition and its geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <solid2dBlock>
    <name>ConductingPlate1_Rectangle1</name>
    <active>true</active>
    <location>
      <x>0</x>
      <y>-0.2</y>
      <z>0</z>
    </location>
    <size>
      <x>0.8</x>
      <y>0.8</y>
      <z>0.001</z>
    </size>
    <plane>+xy</plane>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </solid2dBlock>
</geometry>
```

Wall

Note the following regarding exported wall objects:

- Symmetry walls are not exported.
- Walls are exported with a thickness of 1 mm.

The following is the EC XML format for a wall boundary condition and its geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <solid2dBlock>
    <name>StationaryWall1_Box1</name>
    <active>true</active>
    <location>
      <x>0.6</x>
      <y>0</y>
      <z>0</z>
    </location>
    <size>
      <x>0</x>
      <y>0.4</y>
      <z>0.4</z>
    </size>
    <plane>+yz</plane>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </solid2dBlock>
</geometry>
```

Block (Box)

Note the following regarding exported block objects:

- All 3D shapes except for cylinders are exported as block objects.
- Hollow and fluid blocks are not exported.

The following is the EC XML format for a block boundary condition assigned to box geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <solid3dBlock>
    <name>Block1_Box1</name>
    <active>true</active>
    <location>
      <x>0.2</x>
      <y>0</y>
      <z>0</z>
    </location>
    <size>
      <x>0.2</x>
      <y>0.2</y>
      <z>0.2</z>
    </size>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </solid3dBlock>
</geometry>
```

Block (Cylinder)

Hollow and fluid blocks are not exported.

The following is the EC XML format for a block boundary condition assigned to cylinder geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <solidCylinder>
    <name>Block1_Cylinder1</name>
    <active>true</active>
    <location>
      <x>0.117157287525</x>
      <y>-0.0828427124746</y>
      <z>0</z>
    </location>
    <size>
      <x>0.565685424949</x>
      <y>0.565685424949</y>
      <z>0.4</z>
    </size>
    <plane>+xy</plane>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </solidCylinder>
</geometry>
```

Source (3D Box)

Note the following regarding exported 3D source objects:

- 3D sources are exported as source block objects.
- Sources with a **Thermal Condition** of **Surface Flux** or **Fixed Temperature** are exported with a power specification of 0.

The following is the EC XML format for a source boundary condition assigned to 3D Box geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <sourceBlock>
    <name>Source1_Box1</name>
    <active>true</active>
    <location>
      <x>0.6</x>
      <y>0.4</y>
      <z>0</z>
    </location>
    <size>
      <x>0.4</x>
      <y>0.1</y>
      <z>0.2</z>
    </size>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </sourceBlock>
</geometry>
```

Source (2D Rectangle)

Note the following regarding exported 2D source objects:

- 2D sources are exported as 2D source block objects.
- Sources with a **Thermal Condition** of **Surface Flux** or **Fixed Temperature** are exported with a specification of power of 0.

The following is the EC XML format for a source boundary condition assigned to 2D rectangle geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <source2dBlock>
    <name>Source1_Rectangle1</name>
    <active>true</active>
    <location>
      <x>-0.2</x>
      <y>0.2</y>
      <z>0</z>
    </location>
    <size>
      <x>0.8</x>
      <y>0.2</y>
      <z>0.001</z>
    </size>
    <plane>+xy</plane>
    <material>Al-Extruded</material>
    <powerDissipation>0</powerDissipation>
  </source2dBlock>
</geometry>
```

Grille

Note the following regarding exported grille objects:

- The **Pressure Loss Type Loss Curve** is not supported. It is exported with a loss coefficient of 0 and a free area ratio of 1.
- The **Pressure Loss Type Loss Coefficient** is calculated based on the **Resistance Type**. For the given **Resistance Type**, the loss coefficient is computed as follow, where A is the free area ratio:
  - **Perforated Thin Vent**:

$$l_c = \frac{1}{A^2}\left[0.707(1-A)^{0.375} + 1 - A\right]^2$$

  - **Circular Metal Wire Screen**:

$$l_c = 1.3\,(1-A) + \left(\frac{1}{A} - 1\right)^2$$

  - **Two-Plane Screen Cyl. Bars**:

$$l_c = \frac{1.28\,(1-A)}{A^2}$$

The following is the EC XML format for a grille boundary condition and its geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <grille>
    <name>Grille1_Box1</name>
    <active>true</active>
    <location>
      <x>0.6</x>
      <y>0.2</y>
      <z>0</z>
    </location>
    <size>
      <x>0.001</x>
      <y>0.2</y>
      <z>0.2</z>
    </size>
    <plane>+yz</plane>
    <freeAreaRatio>1</freeAreaRatio>
    <lossCoefficient>0</lossCoefficient>
  </grille>
</geometry>
```

Resistance

Note the following regarding exported resistance objects:

- The **Pressure Loss Types Power Law** and **Loss Curve** are not exported.
- **Pressure Loss Types** of **Power Law** and **Loss Curve** are exported with a loss coefficient of 0 and a free area ratio of 1.
- For resistances with a **Pressure Loss Type** of **Device/Approach**, only the linear coefficient and free area ratio are exported.

The following is the EC XML format for a resistance boundary condition and its geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <flowResistance>
    <name>Resistance1_Box1</name>
    <active>true</active>
    <location>
      <x>0.2</x>
      <y>0.2</y>
      <z>0</z>
    </location>
    <size>
      <x>0.4</x>
      <y>0.2</y>
      <z>0.2</z>
    </size>
    <freeAreaRatio>
      <x>1</x>
      <y>1</y>
      <z>1</z>
    </freeAreaRatio>
    <lossCoefficient>
      <x>0</x>
      <y>0</y>
      <z>0</z>
    </lossCoefficient>
  </flowResistance>
</geometry>
```

Opening

Note the following regarding exported opening objects:

- Recirculating openings are not exported.
- Free openings are exported as grille objects with a loss coefficient of 0 and a free area ratio of 1.

Network

Note the following regarding exported network objects:

- Only one and two-resistor networks are exported.
- Network faces must be parallel and part of the same 3D geometry.

The following is the EC XML format for a network boundary condition and its geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <twoResistorModel>
    <name>Network1_Box1</name>
    <active>true</active>
    <location>
      <x>0.03</x>
      <y>0.02</y>
      <z>0.02</z>
    </location>
    <size>
      <x>40</x>
      <y>60</y>
      <z>20</z>
    </size>
    <material>Al-Extruded</material>
    <powerDissipation>5</powerDissipation>
    <thetaJC>5</thetaJC>
    <thetaJB>1</thetaJB>
  </twoResistorModel>
</geometry>
```

PCB

Note the following regarding exported PCB objects:

- Only PCB substrate layers and exported. Devices and solderballs are not exported.
- The default assigned material is FR-4.

The following is the EC XML format for a PCB component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <printedCircuitBoard>
    <name>pcb0</name>
    <active>true</active>
    <location>
      <x>-17.8308</x>
      <y>-16.51</y>
      <z>0</z>
    </location>
    <size>
      <x>35.6616</x>
      <y>33.02</y>
      <z>0.001</z>
    </size>
    <material>FR-4</material>
  </printedCircuitBoard>
</geometry>
```

Heat sink

Heat sink bases and pins are exported as individual solid 3D blocks with its bounding box size and location data.

The following is the EC XML format for a heat sink component.

> **Note:** The heat sink definition contains one solid 3D block entry for the base and each pin.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <heatsink>
    <name>heatsink4</name>
    <active>true</active>
    <parts>
      <solid3dBlock>
        <name>base</name>
        <active>true</active>
        <location>
          <x>-0.1</x>
          <y>-0.1</y>
          <z>0</z>
        </location>
        <size>
          <x>0</x>
          <y>0</y>
          <z>0</z>
        </size>
        <material>Al-Extruded</material>
        <powerDissipation>0</powerDissipation>
      </solid3dBlock>
    </parts>
  </heatsink>
</geometry>
```

Fan (2D)

Note the following regarding exported 2D fan objects:

- Fans with **Flow Type** of **Fixed Volumetric** and **Curve** are exported.
- The **Flow Type Fixed Mass Flow** is not exported.
- Fans with a **Flow Type** of **Fixed Mass Flow**are exported with a fixed flow specification of 0.
- The **Flow Direction** is exported as *<plane></plane>* in the exported xml schema.

The following is the EC XML format for a 2D fan component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <rectangular2dFan>
    <name>fan0</name>
    <active>true</active>
    <location>
      <x>-0.15</x>
      <y>-0.15</y>
      <z>0</z>
    </location>
    <size>
      <x>0.3</x>
      <y>0.3</y>
      <z>0.001</z>
    </size>
    <plane>+xy</plane>
    <flowDefinition>
      <fanCurve>
        <fanCurvePoint>
          <pressure>0</pressure>
          <volumeFlow>30</volumeFlow>
        </fanCurvePoint>
        <fanCurvePoint>
          <pressure>0.01</pressure>
          <volumeFlow>0</volumeFlow>
        </fanCurvePoint>
      </fanCurve>
    </flowDefinition>
  </rectangular2dFan>
</geometry>
```

Fan (3D)

Note the following regarding exported 3D fan objects:

- Fans with **Flow Type** of **Fixed Volumetric** and **Curve** are exported.
- The **Flow Type Fixed Mass Flow** is not exported.
- Fans with a **Flow Type** of **Fixed Mass Flow**are exported with a fixed flow specification of 0.
- The **Flow Direction** is exported as *<plane></plane>* in the exported xml schema.
- For 3D fans with a hub size specified as 0, the exported value is calculated as 1/10th the size of the fan dimensions, where the fan dimension used is the diameter (circular fans) or the smaller of width or length (rectangular fans).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is the EC XML format for a 3D fan component with a **Curve** specified as the **Flow Type**.

```
<geometry>
  <axial3dFan>
    <name>fan0</name>
    <active>true</active>
    <location>
      <x>-0.15</x>
      <y>-0.15</y>
      <z>0</z>
    </location>
    <size>
      <x>0.3</x>
      <y>0.3</y>
      <z>0.05</z>
    </size>
    <plane>+yz</plane>
    <hubSize>0</hubSize>
    <flowDefinition>
      <fanCurve>
        <fanCurvePoint>
          <pressure>0</pressure>
          <volumeFlow>30</volumeFlow>
        </fanCurvePoint>
        <fanCurvePoint>
          <pressure>0.01</pressure>
          <volumeFlow>0</volumeFlow>
        </fanCurvePoint>
      </fanCurve>
    </flowDefinition>
  </axial3dFan>
</geometry>
```

The following is the EC XML format for a 3D fan component with a **Fixed Volumetric** specified as the **Flow Type**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <axial3dFan>
    <name>Fan1_1</name>
    <active>true</active>
    <location>
      <x>-0.15</x>
      <y>-0.15</y>
      <z>0</z>
    </location>
    <size>
      <x>0.3</x>
      <y>0.3</y>
      <z>0.05</z>
    </size>
    <plane>+xy</plane>
    <hubSize>0.02</hubSize>
    <flowDefinition>
      <fixedFlowRate>20</fixedFlowRate>
    </flowDefinition>
  </axial3dFan>
</geometry>
```

Mesh Region

Note the following regarding exported mesh regions:

- A mesh region is exported as an assembly.
- Mesh regions that are assigned to geometries are associated with those geometries when exported.

The following is the EC XML format for a mesh region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<geometry>
  <assembly>
    <name>MeshRegion1</name>
    <active>true</active>
    <geometry>
      <solid3dBlock>
        <name>Box1</name>
        <active>true</active>
        <location>
          <x>1</x>
          <y>0.5</y>
          <z>0.5</z>
        </location>
        <size>
          <x>0.2</x>
          <y>0.3</y>
          <z>0.3</z>
        </size>
        <material>Al-Extruded</material>
        <powerDissipation>0</powerDissipation>
      </solid3dBlock>
    </geometry>
  </assembly>
</geometry>
```

Material

Note the following regarding exported materials:

- Only solid materials are exported.
- Temperature-dependent information is exported for materials assigned a dataset variable.
- Thermal modifiers are not exported for **Mass Density** and **Specific Heat**.
- Thermal Conductivity is supported for **Simple** and **Anisotropic** types.
    - For **Simple** with a **Quadratic** thermal modifier where C2 = 0, it is exported as linear temperature-dependent where the coefficient is equal to $C1 * solidCond$ and TempRef is defined in kelvins.
    - For **Anisotropic**, the thermal conductivity is exported as orthotropic.

The following is the EC XML format for a material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<materials>
  <material>
    <name>Al-Extruded</name>
      <density>2800</density>
      <specific_heat>900</specific_heat>
      <surfaceEmissivity>0</surfaceEmissivity>
      <thermalConductivity>
        <isotropic>
          <conductivity>205</conductivity>
        </isotropic>
      </thermalConductivity>
  </material>
  <material>
    <name>Au-Typical</name>
      <density>19281</density>
      <specific_heat>131</specific_heat>
      <surfaceEmissivity>0</surfaceEmissivity>
      <thermalConductivity>
        <linearTemperatureDependant>
          <conductivity>313</conductivity>
          <coeff>2504</coeff>
          <tref>297.15</tref>
        </linearTemperatureDependant>
       </thermalConductivity>
  </material>
  <material>
    <name>Al-Extrudedtmp</name>
    <density>2800</density>
    <specific_heat>900</specific_heat>
    <surfaceEmissivity>0</surfaceEmissivity>
    <thermalConductivity>
      <nonLinearTemperatureDependant>
        <tempCurvePoint>
          <temperature>10</temperature>
          <conductivity>205</conductivity>
        </tempCurvePoint>
        <tempCurvePoint>
          <temperature>20</temperature>
          <conductivity>512.5</conductivity>
        </tempCurvePoint>
        <tempCurvePoint>
          <temperature>40</temperature>
          <conductivity>820</conductivity>
        </tempCurvePoint>
      </nonLinearTemperatureDependant>
    </thermalConductivity>
  </material>
</materials>
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor Point

The following is the EC XML format for a monitor point.

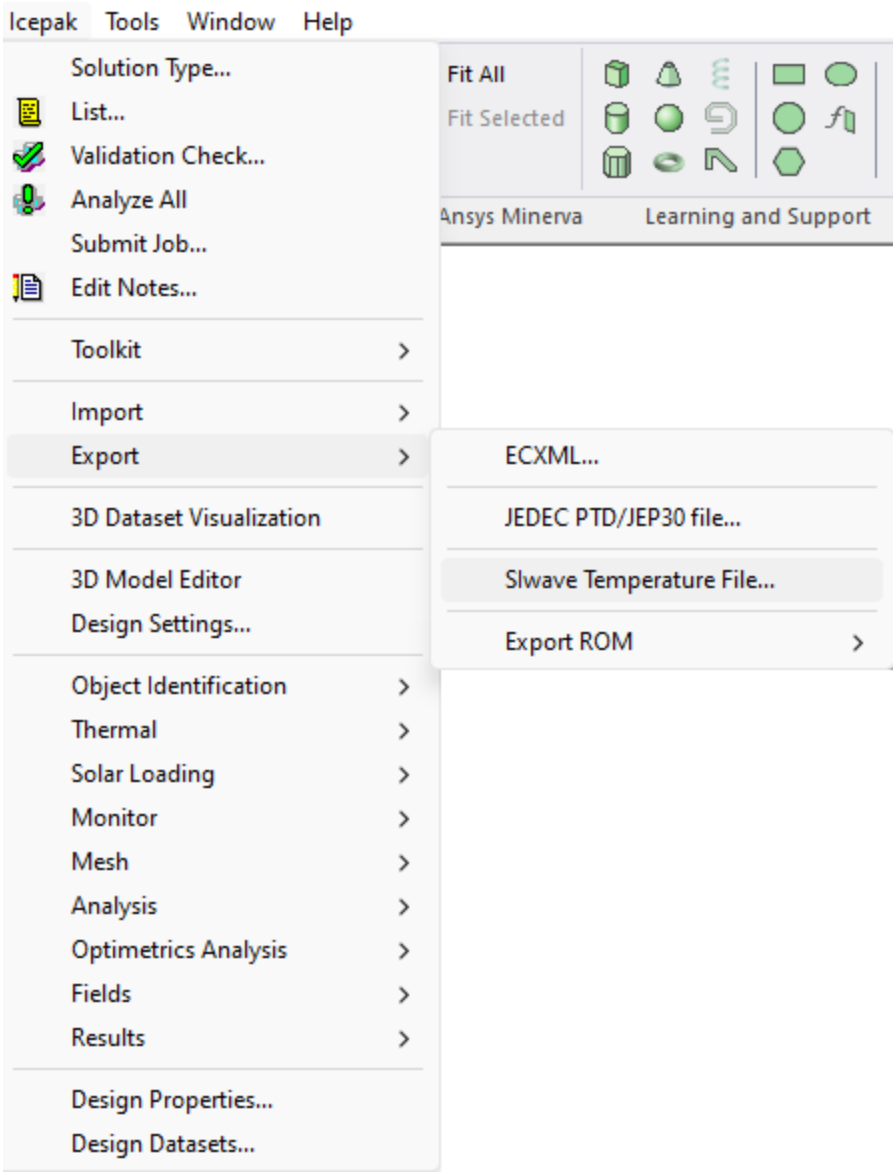
```
<monitorPoint>
  <name>pmon_mon_pt_1_1</name>
  <active>true</active>
  <location>
    <x>0.4</x>
    <y>0.5</y>
    <z>0.5</z>
  </location>
</monitorPoint>
```

Exporting SIwave Temperature Files

Icepak enables you to export temperature data for use in SIwave. The Icepak design must contain a PCB linked to an HFSS 3D Layout design with a DC IR Solution setup. See Coupling SIwave DC IR Loss for information on DC IR loss coupling between SIwave and Icepak.

> **Note:**
>
> Temperature data files from projects containing an Icepak PCB native 3D component can be exported. Layout components are not currently supported.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can also access the export menu by right-clicking on the design in the **Project Manager**.

The temperature data (SIwave.sitemp) file is exported to a *Project_Name*.aedtexport directory created within the project folder. The SIwave.sitemp file contains nodal temperatures for import into SIwave.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
B_NODES 2057
1 -0.010759 -0.009312 0.002035
2 0.094149 -0.009312 0.001674
3 0.012554 0.015318 0.001674
4 0.012554 0.056367 0.000000
5 0.047523 0.015318 0.000828
6 0.094149 0.031737 0.000000
7 0.000897 -0.001102 0.000000
8 0.094149 0.072786 0.002035
9 0.024210 -0.009312 0.000828
10 0.059180 -0.009312 0.001674
11 0.000897 0.064576 0.001039
12 -0.010759 -0.009312 0.000000
13 0.024210 0.023527 0.000048
14 0.059180 0.048157 0.001886
15 0.035867 0.015318 0.002035
```

Exporting JEDEC JEP 30 - PTD Files

JEDEC standard JEP30-T100A establishes the requirements for exchanging electronic part thermal data. Two types of thermal Network Models are supported; DELPHI-Model, as defined in JESD15-4 and 2-ResistorModel, as defined in JESD15-3. Both models involve the definition of a nodal thermal network, which can be exported into Icepak using *.xml or *.jpd files, respectively for JEDEC standard JEP30-T100A and for its predecessor known as Package Thermal Data Interchange File Format (PTD IFF). For thermal Network Models import/export, these files contain similar information formatted in slightly differing schemas.

The *.xml or *.jpd files contain the following information:

- Model ID name
- Units for thermal data
- Model geometry specifications: package (Package Body Center to the Origin), internal and external nodes definitions (Volume, face areas)
- Network specifications: model type, nodal power and capacitance, resistance interconnection schemas

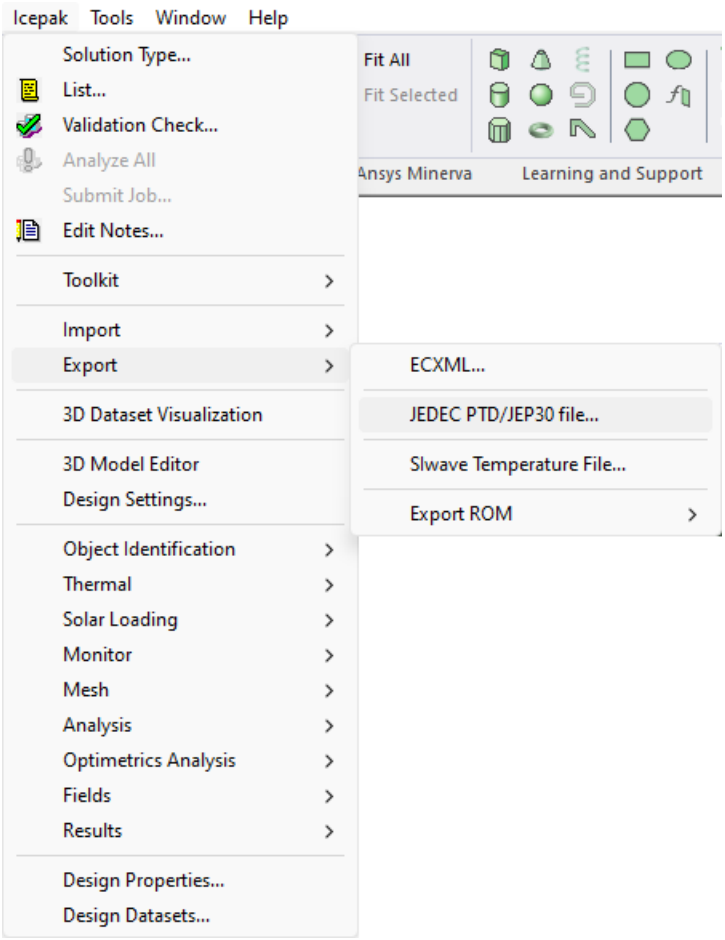
For detailed schema information regarding these file formats, see PTD IFF File Format and JEP 30 File Format.

Additionally, *.xml or *.jpd files enable multiple network export information of any design, so all thermal model's alignments are exported.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For more information about JEDEC standards, visit www.jedec.org.



Note:

You can also access the export menu by right-clicking on the design in the **Project Manager**.

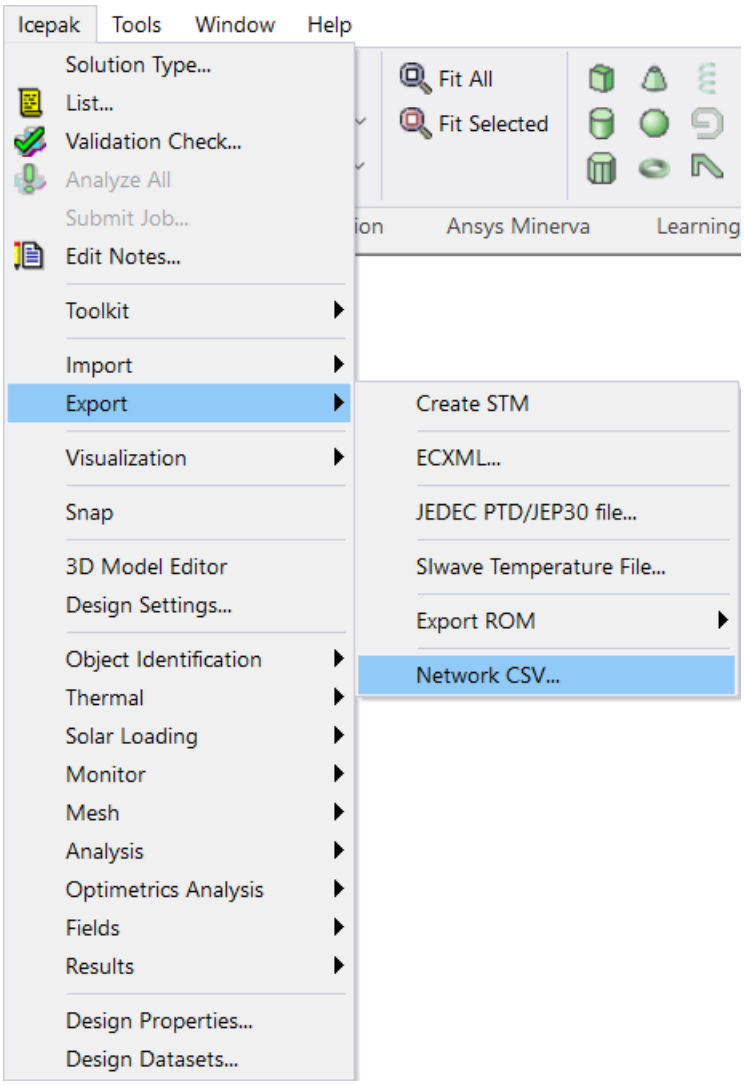
Exporting Network CSV Files

Icepak can export networks as comma-separated value (.csv) files. To export a network .csv file, select **Icepak** > **Export** > **Network CSV**. In the **Save As** dialog box, navigate to the desired location, enter a file name, and click **Save** to save the network geometry and boundary condition information in a .csv file. See Network CSV File Format for the information included in the exported file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You can also access the export menu by right-clicking on the design in the **Project Manager**.



Network CSV File Format

The following image is an example of the network .csv file format.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# network starts
# network name,num_bound,value,num_int,value,num_face,value
network.1,num_bound,0,num_int,2,num_face,2

# boundary nodes
# node name,prop,value,prop,value,...

# internal nodes
# node name,prop,value,prop,value,...
int0,Power,1 W,Mass,0 kg,Capacity,0 J/kg-K,C (W/K),1.0,Low T,ambient,High
T,ambient,TransientPower,0,x,220,y,160
int1,Power,1 W,Mass,0 kg,Capacity,0 J/kg-K,C (W/K),1.0,Low T,30 K,High T,40
C,TransientPower,0,x,500,y,170

# surface nodes
# node name,prop,value,prop,value,...
face0,Effective thickness,1 m,x,90,y,150
face1,x,900,y,90

# thermal links
# node1 name,node2 name,type,prop,value,prop,value,...
int0,face0,R,Resistance,1 C/W
int1,int0,R,Resistance,0 C/W
int1,face1,R,Resistance,0 C/W

# curve data
# node name,point1,point2,point3,...

# plates
# face name,face type,prop,value,prop,value,...
face0,quad,copyspace,0,diff_flag,0,iradius,0,numcopies,1,plane,2,plate_flag,
0,split_flag,0,thickness,0,volume_flag,0,xd,0.2,xe,0.6,xoff,0,xs,0.4,yd,0.2,
ye,0.6,yoff,0,ys,0.4,zd,0,ze,0.4,zoff,0,zs,0.4
face1,quad,copyspace,0,diff_flag,0,iradius,0,numcopies,1,plane,2,plate_flag,
0,split_flag,0,thickness,0,volume_flag,0,xd,0.2,xe,0.6,xoff,0,xs,0.4,yd,0.2,
ye,0.6,yoff,0,ys,0.4,zd,0,ze,0.6,zoff,0,zs,0.6

# network ends
```

Creating and Exporting LPV ROM Files

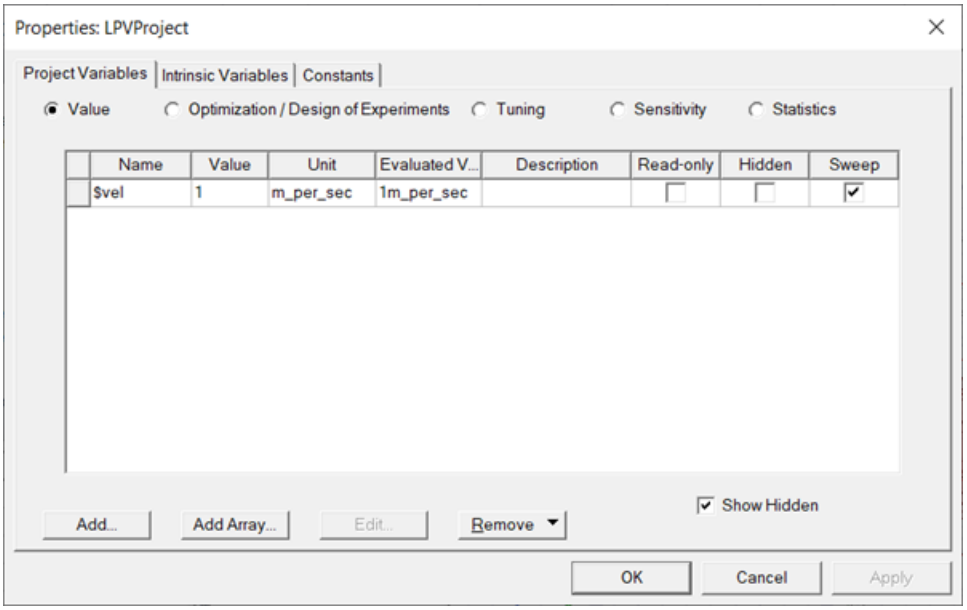
Modeling with Linear Parameter Varying Reduced Order Models (LPV ROMs) allows you to use a variable to simulate varying flow rates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overview of LPV ROM Creation and Export

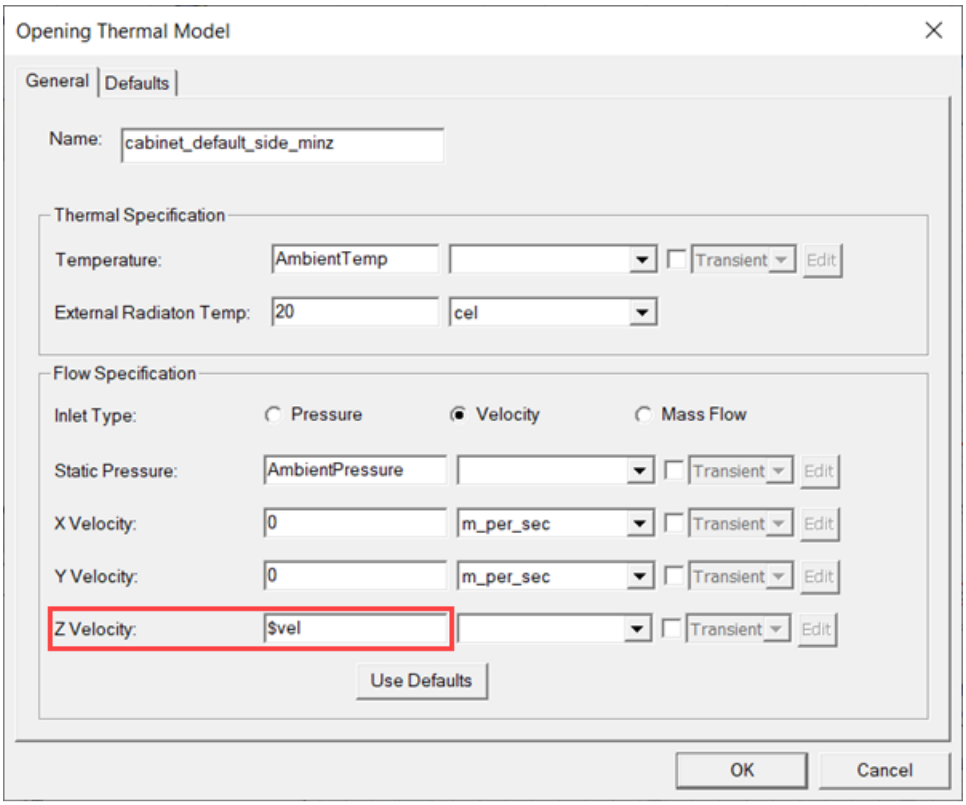
Create a Steady-State Analysis

1. Create a steady-state Icepak design.
2. Create a project variable for velocity (for example, $vel as displayed in the following image).



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

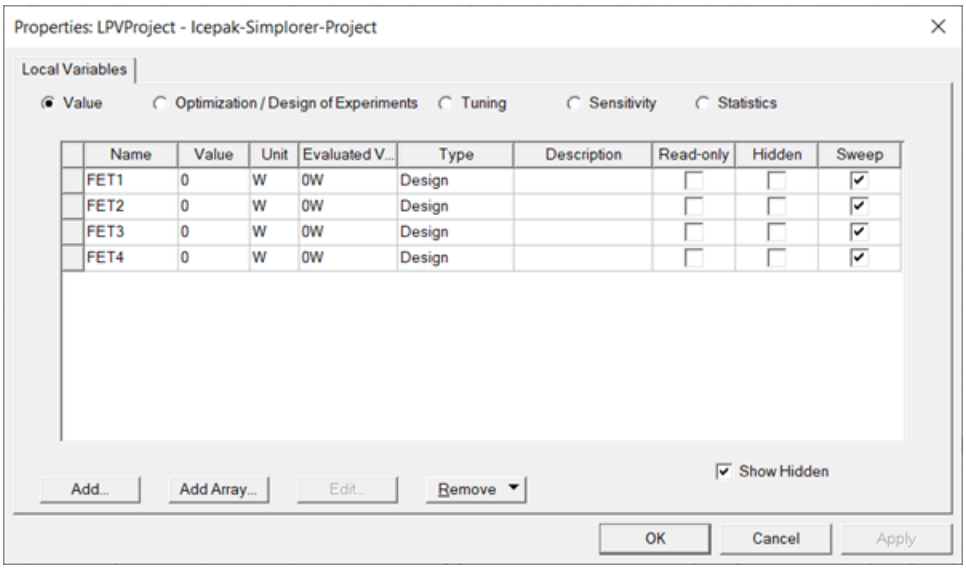
3. After assigning boundary conditions, assign the project variable to the opening boundary condition serving as the inlet.



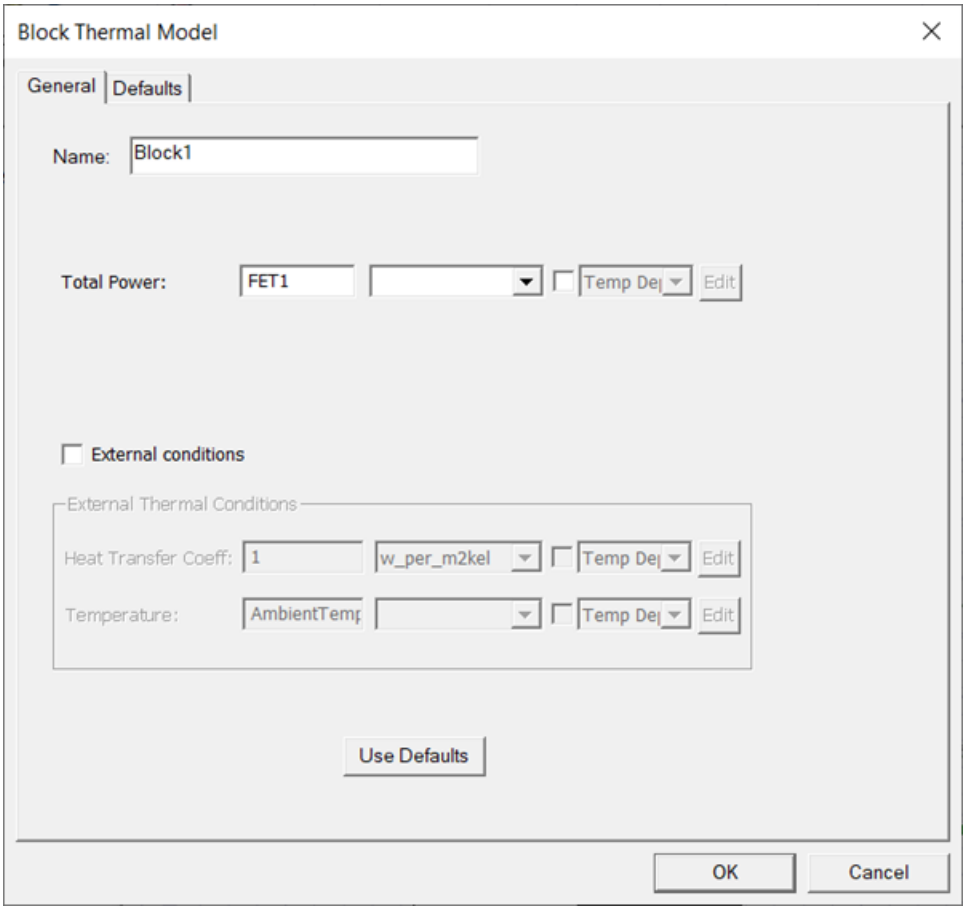
Run a Transient Optimetrics Analysis

1. Copy the steady-state design and paste it in the project.
2. Change the **Solution Type** to **Transient**.
3. Define local design variables to be used in the parametric analysis. For example, create variables to be used to define the power for block boundary conditions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

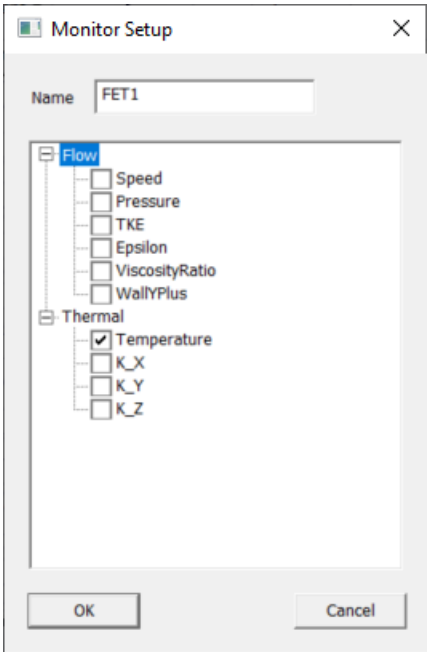


4. Assign the design variables to the boundary condition parameters (e.g., block power as displayed in the following image).



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Add a solution setup.
   - On the **General** tab, select **LTI ROM Defaults** from the solutions setup defaults drop-down list.
   - On the **Solver Settings** tab under **Import Options**, select **Start/Continue from a previously solved setup** and click **Setup Link**.
6. In the **Setup Link** dialog box next to **Source Design**, deselect **Use this design** and select the steady-state design in your project.
7. On the **Variable Mapping** tab, click **Map Variable by Name** to sync the velocity variable in the steady-state and transient designs
8. Click **OK** to close the **Setup Link** dialog box.
9. Click **OK** in the **Icepak Solve Setup** dialog box.
10. Create temperature monitor points for each **Total Power** variable using the same name as the variable. In this example, assign a temperature monitor point named FET1 to the geometry associated with the block boundary condition whose **Total Power** is defined as FET1. Repeat for FET2, FET3, and FET4.



11. From **Optimetrics**, create a parametric setup using all of the variables you've created. Define a nonzero value for the velocity variable. For the power variables, each row in the parametric table should have one variable with the same nonzero value and all other variables with a value of 0. The example below features velocity values of 1 m_per_sec and 2 m_per_sec and a power value of 5W.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

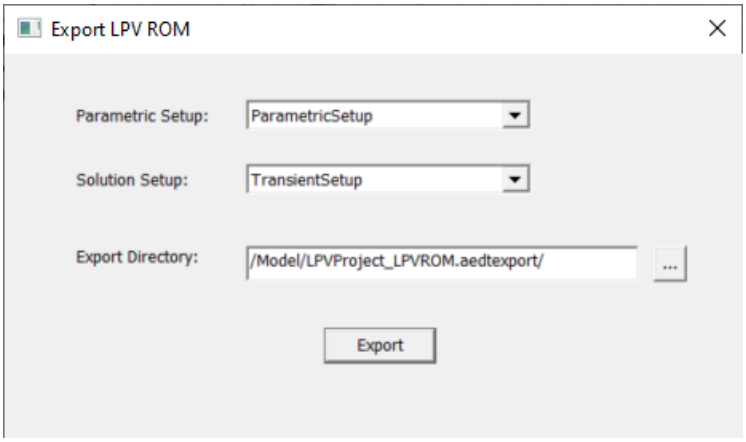
> **Note:**
>
> The unit for the variables must be consistent with the variable type (e.g., power units for power variables and speed unit for velocity variable). Otherwise, the export will not be performed.

*	$vel	FET1	FET2	FET3	FET4
1	1m_per_sec	5W	0W	0W	0W
2	1m_per_sec	0W	5W	0W	0W
3	1m_per_sec	0W	0W	5W	0W
4	1m_per_sec	0W	0W	0W	5W
5	2m_per_sec	5W	0W	0W	0W
6	2m_per_sec	0W	5W	0W	0W
7	2m_per_sec	0W	0W	5W	0W
8	2m_per_sec	0W	0W	0W	5W

12. Analyze the transient parametric setup.

Export LPV ROM

1. From the Icepak menu, select **Export ROM** > **LPV ROM**.



2. Select an **Export Directory**.
3. Click **Export** to generate the files required for TwinBuilder ROM setup.

Exporting Graphics Files

You can export to the following graphics formats:

Extension	Contents
.bmp	Bitmap files
.gif	Graphics Interchange Format files

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extension	Contents
.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

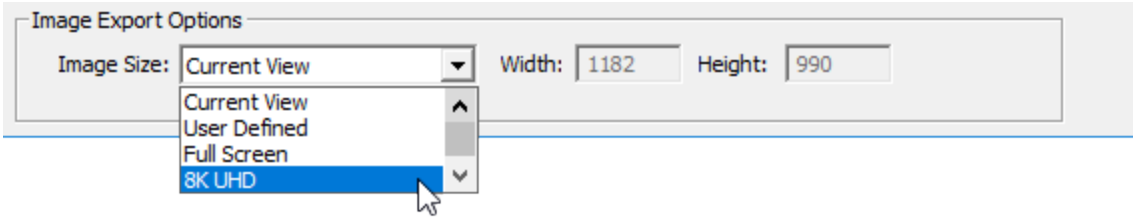
To export a Modeler image to a graphics format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

2. Browse to and select a location to save your file.
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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
oEditor.ExportModelImageToFile("C:/Users/Documents/highresexample_
image.jpg", 7680, 4320,
  ["NAME:SaveImageParams",
    "ShowAxis:=" , "True",
    "ShowGrid:=" , "True",
    "ShowRuler:=" , "True",
    "ShowRegion:=" , "Default",
    "Selections:=" , ""
  ])
```

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>
Python Example	`oDesktop.ExportOptionsFiles ('D:/test/export')`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Validating Projects

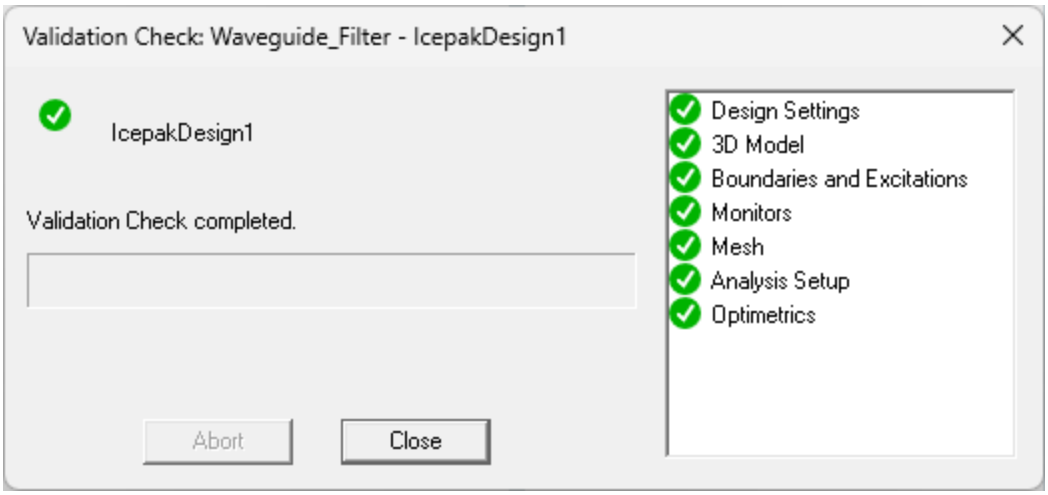
Before running an analysis on a model, it is important to first perform a validation check on the project. During a validation check, the solver examines the setup details of the active project to verify that all the necessary steps have been completed and that their parameters are reasonable.

To perform a validation check on the active project:

1. Click **Icepak** > **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.

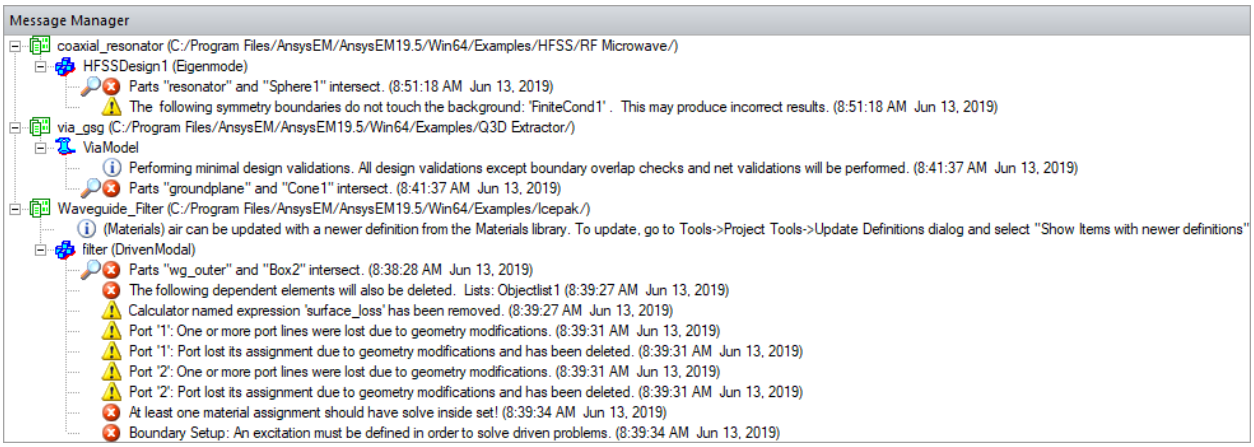
   The following icons can appear next to an item:

	Indicates the step is complete.
	Indicates the step is incomplete.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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 **Icepak** > **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 four levels of model validation for a given design: **None**, **Warning Only**, **Basic**, and **Strict**. The default is **Basic**. The options are explained below.

> **Note:**
>
> This setting only affects the "3D Model" stage of a design validation. To further investigate geometry issues, use the Analyze Objects functionality.

To set the Model Validation level:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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. The default Entity Check Level is **Basic**.

   - **None** – Validation checks are ignored.
   - **Warning Only** – Gives warnings for all issues, but does not fail validation. These issues are posted in the message window as warnings.
   - **Basic** – Only severe problems that are expected to cause missing facets on faces and/or preclude any kind of modeling will be regarded as errors, which will stop the simulation. All other problems will be considered warnings, allowing users to proceed to meshing. Errors must be corrected before attempting to analyze the design under the **Basic** setting; otherwise, you must change the **Model Validation** level to **Warning Only** or **None**.

     > **Note:** Basic checks do not indicate a completely valid model for geometric modeling, but rather the model is in a reasonable state to pursue meshing of the model. Users should carefully review mesh on parts where warnings were detected with the Basic level.

   - **Strict** – Treats almost all geometry/topology issues as errors, which will stop the simulation. 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**.

   > **Note:**
   >
   > Model validations set an entity check of **Warning Only** or higher will issue a warning message if the extents of the model are less than one in all x, y, and z directions. It is recommended that you select units such that the model extents are greater than one in at least one direction.

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.

Copying and Pasting a Project or Design

To **copy** a project or design:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

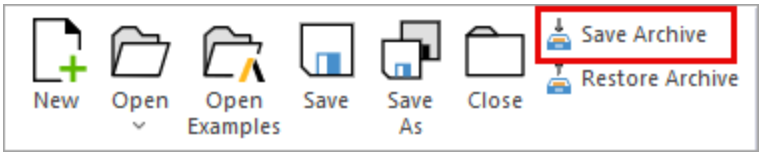
Internally, project archive files are .zip files, and are compatible with any program that can read .zip files. Project archive files have an extension that is unique for each product, and is generated by adding a 'z' to the project file extension (for example, *.aedtz, *.adsnz). This extension displays as the default when saving and restoring archive files.

Archive Preview

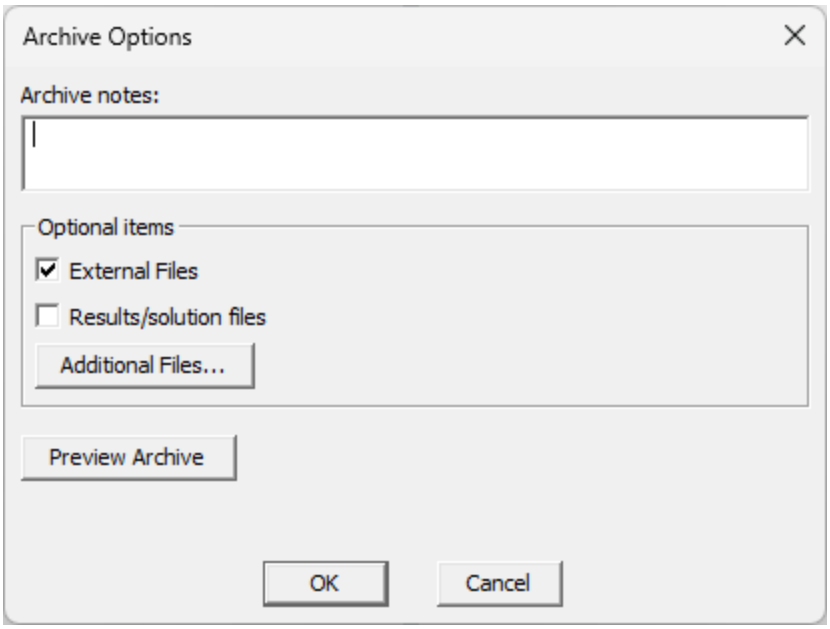
The **Archive** command includes a preview feature that lets you review the contents of a planned archive.

To archive the current project:

1. Click **File** > **Archive** or select the **Desktop** tab of the ribbon and click **Save Archive**.



The **Archive Options** dialog box opens.

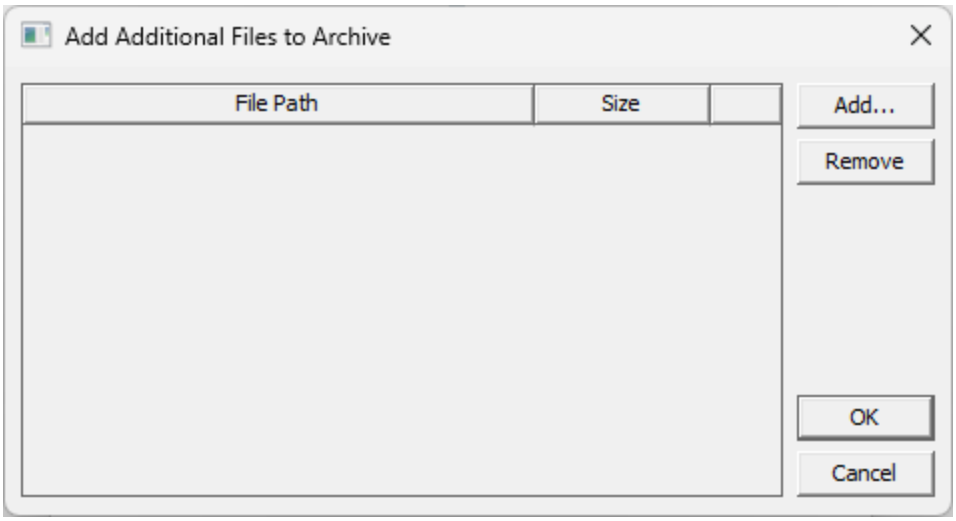


- **Archive Notes** – in this field, you can specify notes that will be visible when previewing the archive. These notes can be viewed from the preview dialog box without restoring the archive.
- **External Files** – selecting this check box causes all external files to be included in the archive. These include any existing files associated with the project, such as

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

linked files, or files added through the **Project** > **Insert Documentation File** command or **Project** > **Data Set** command.

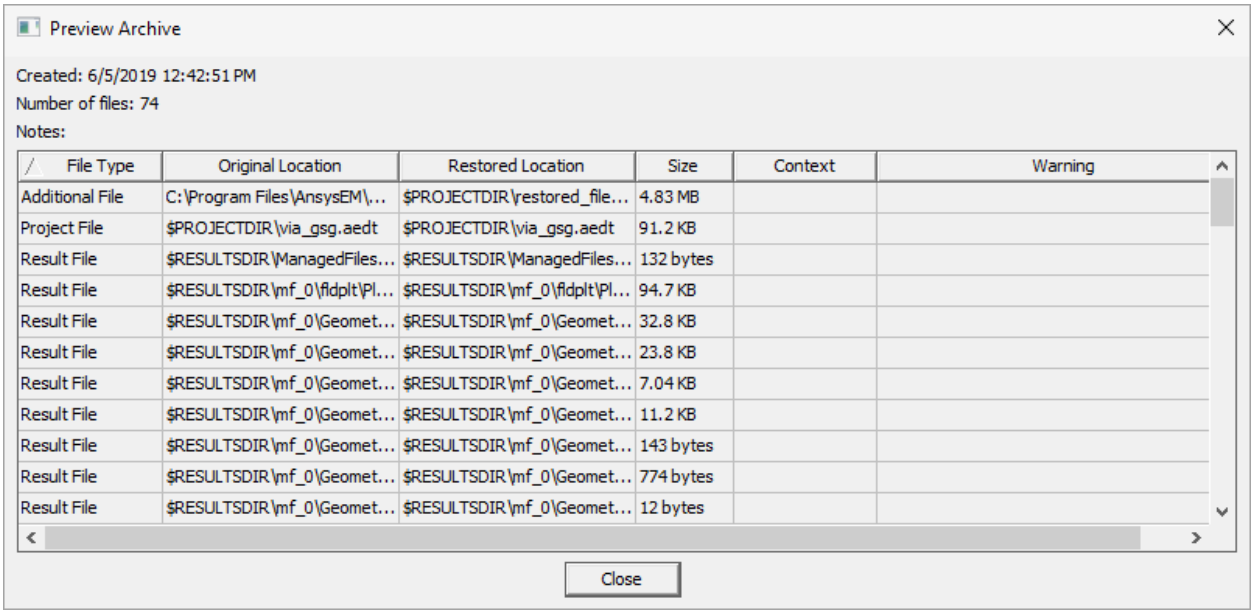
- **Results/Solution Files** – selecting this check box causes the entire results directory to be included in the archive. This may greatly increase the size of the archive file.
- **Additional Files** – clicking this opens the **Add Additional Files to Archive** dialog box.



  - Click **Add** to browse and locate additional files you want to include in the archive.
  - Select and the **Remove** any files listed.
  - 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 the **Preview Archive** button in the Archive Options dialog box 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To read longer locations, you can drag the column header to expand them.

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.

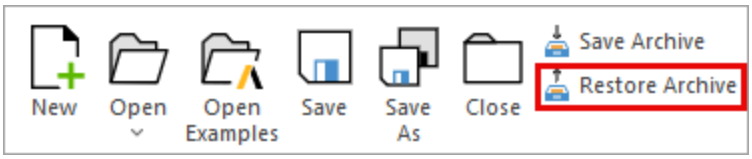
To achieve this, it is sometimes necessary to change the location of files in the archived project so that the external files are located in the project directory. At archive time, any external files not located in the project directory are relocated to the restored_files subdirectory of the project 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

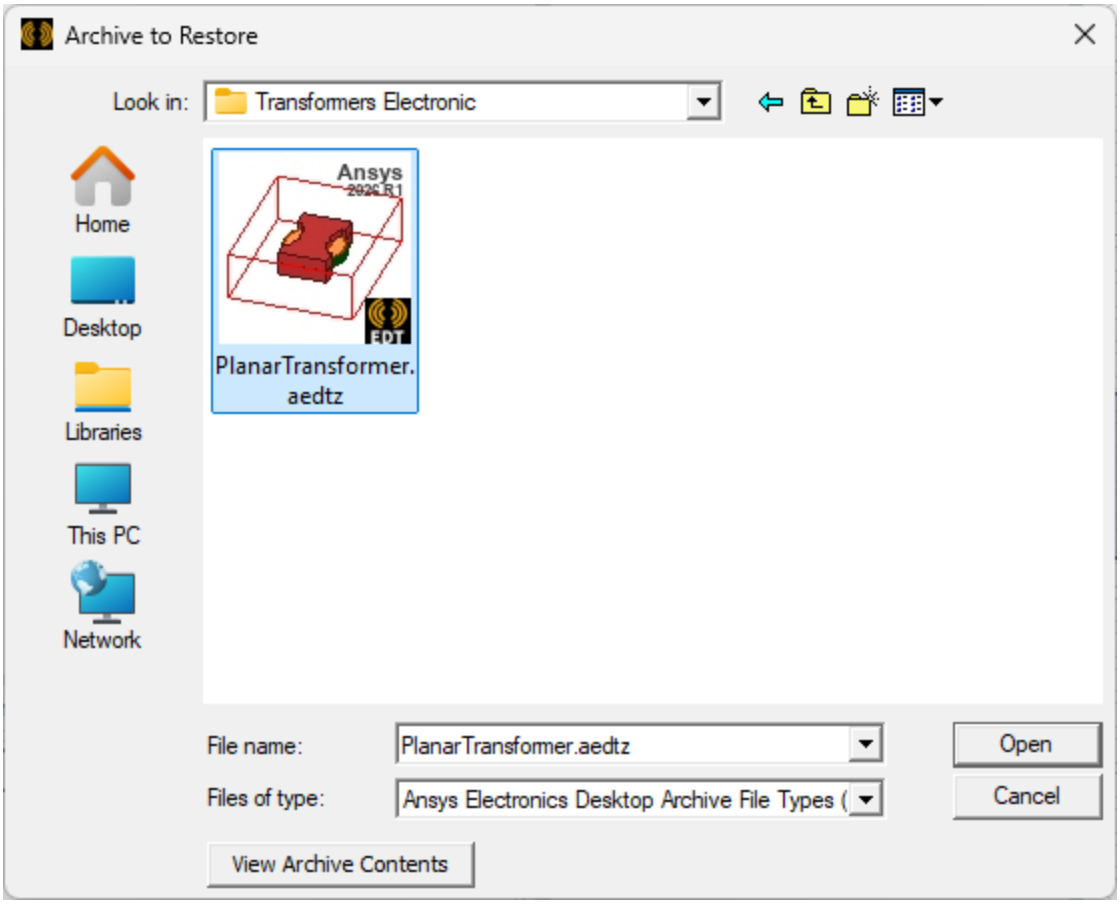
Restoring an Archived Project

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**.



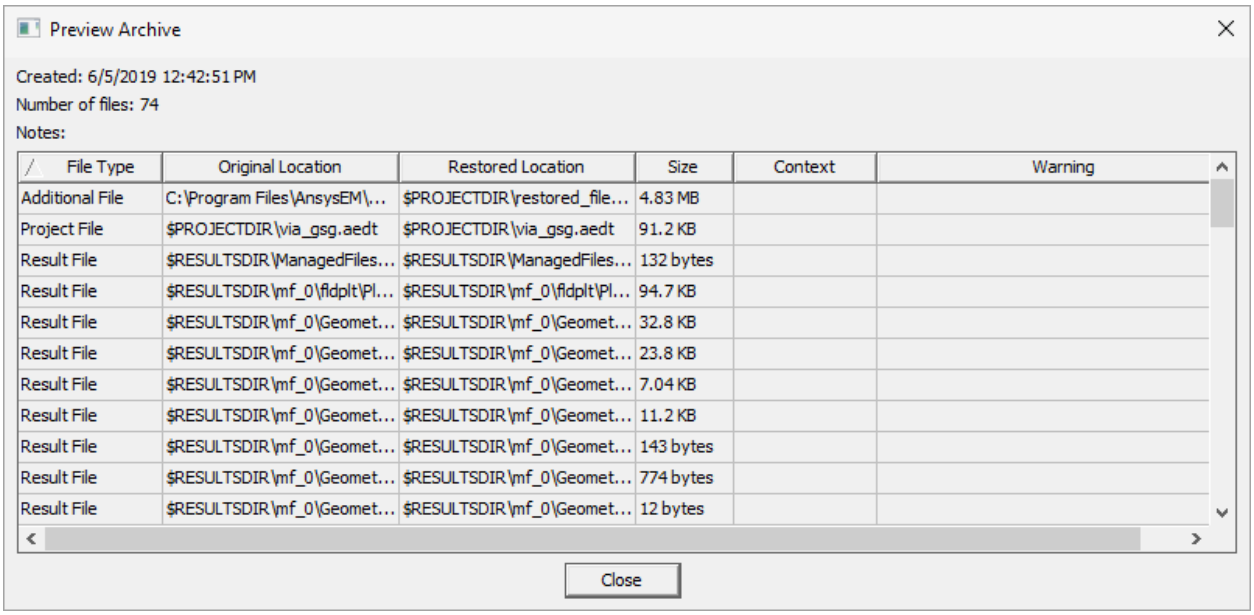
This displays an **Archive to Restore** browser window that lets you navigate your file system for *.aedtz, legacy *.<*solversuffix*>*z* or *.zip archive files.



2. If you select a valid archive file, you can click **View Archive Contents** to preview the contents.

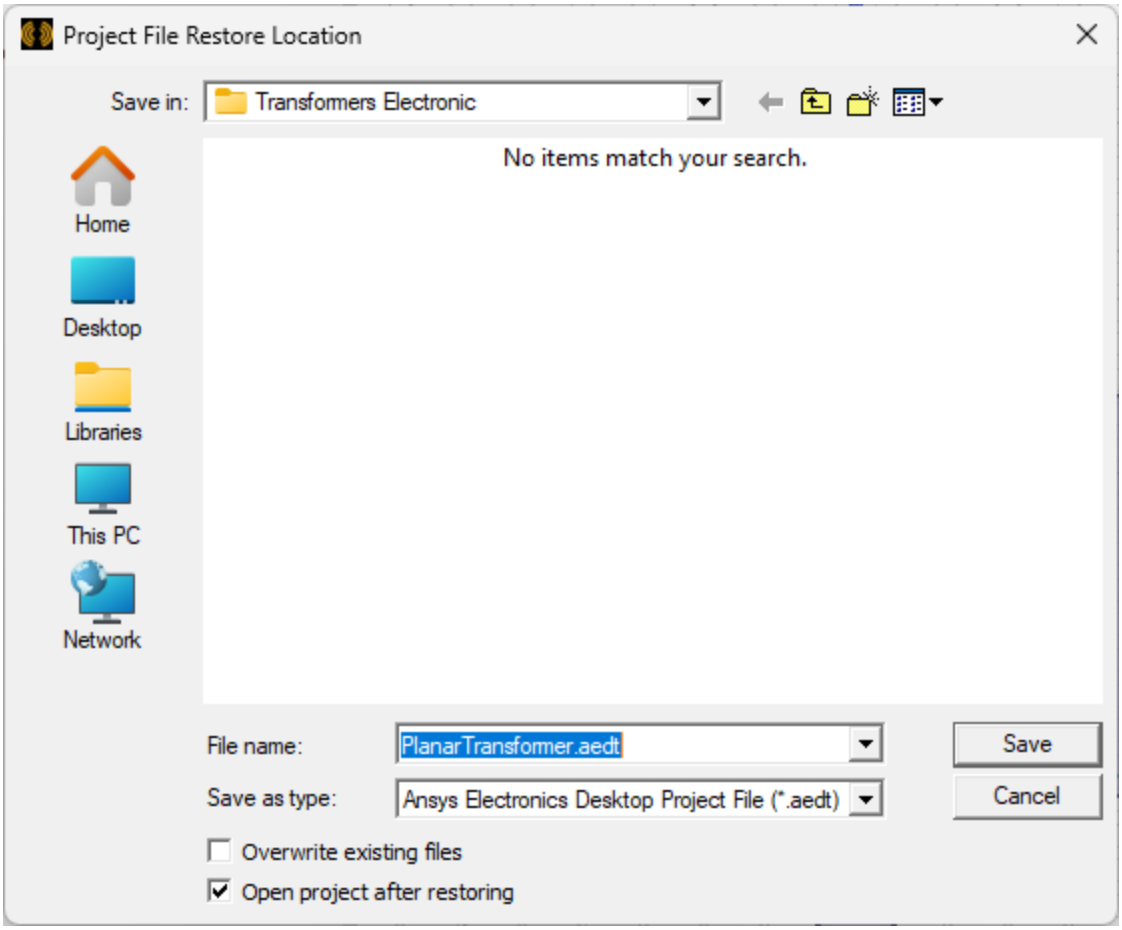
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



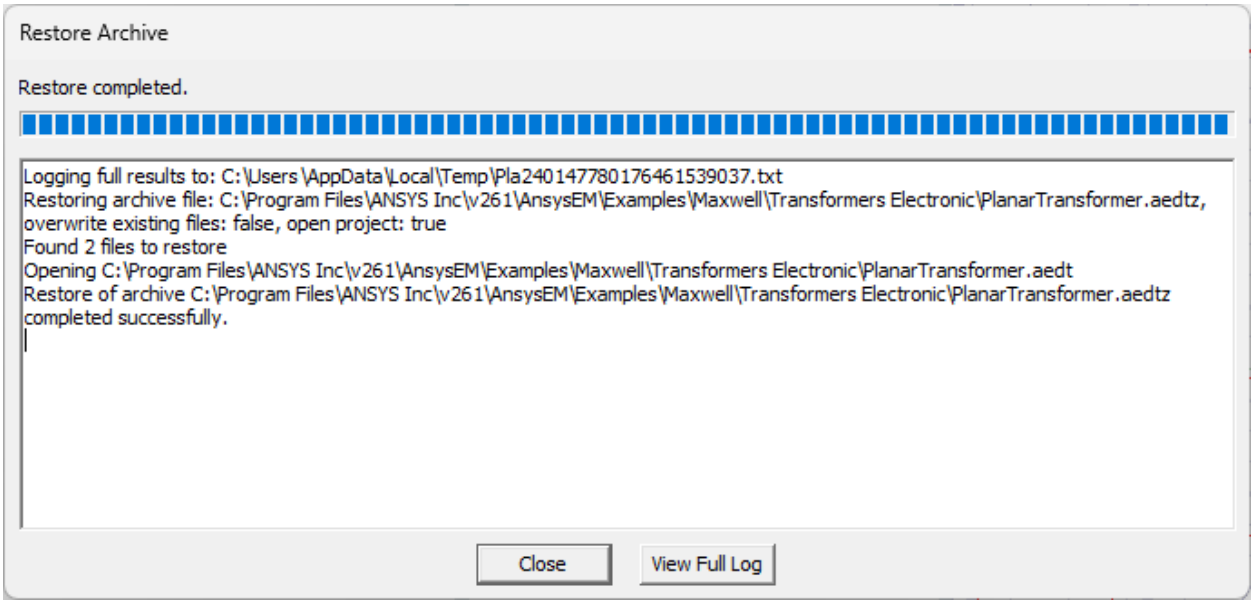
You can edit the file name, and select options:

- **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 2026 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 2026 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**.

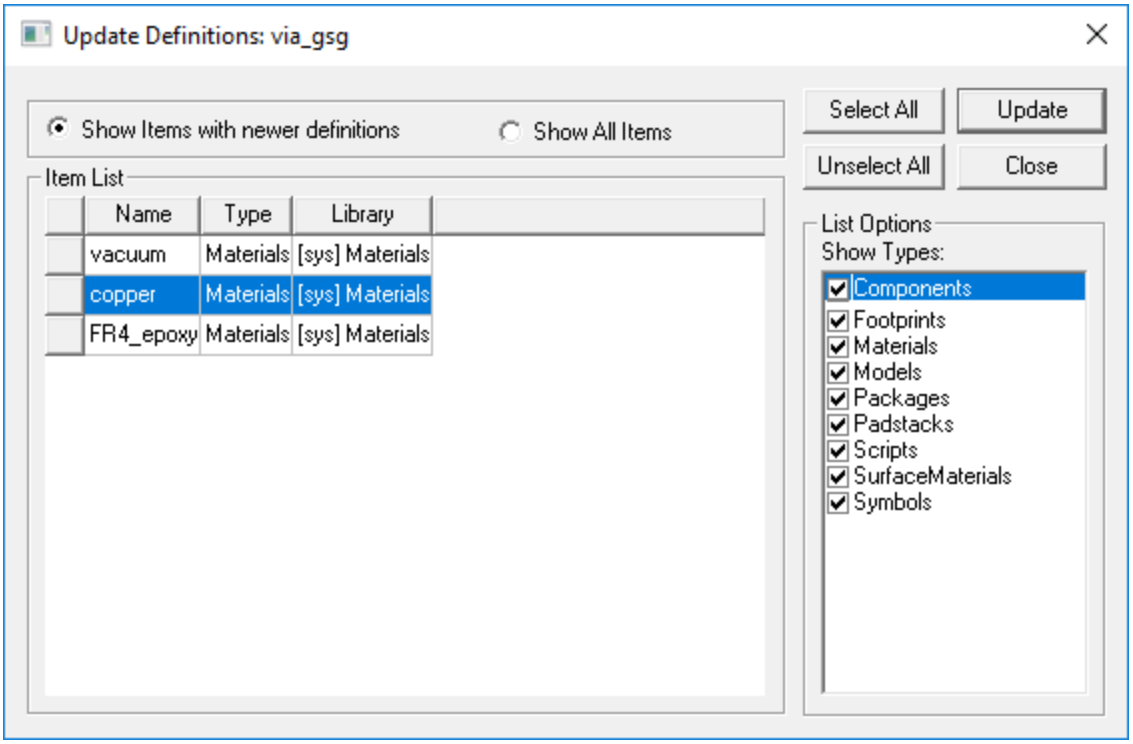
> **Note:**
>
> Read Only settings are not saved when you save the project file.

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 2026 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.

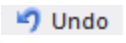
Use the **Redo** command to reperform an action you previously undid.

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 **Icepak** > **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 2026 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.

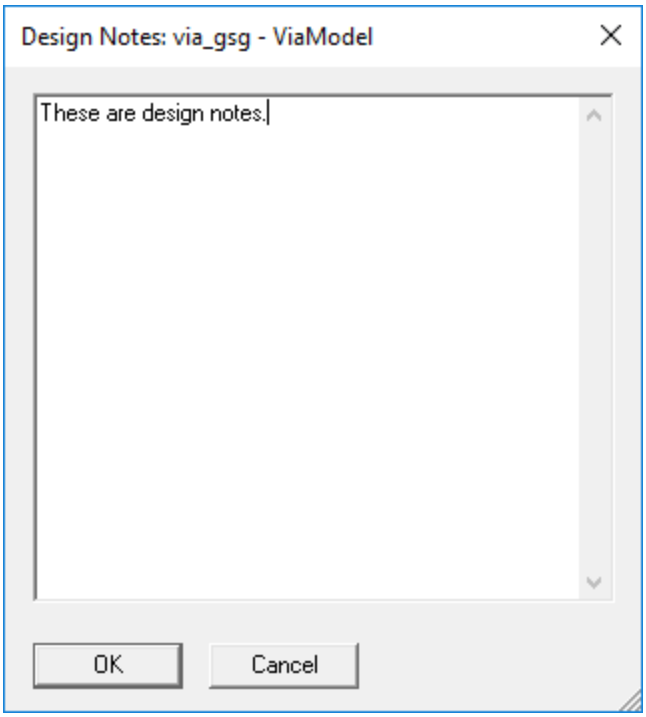
Working with Design Notes

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 **Icepak** > **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.

To edit notes, use the same window or double-click the **Notes** icon (Notes) in the Project Tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

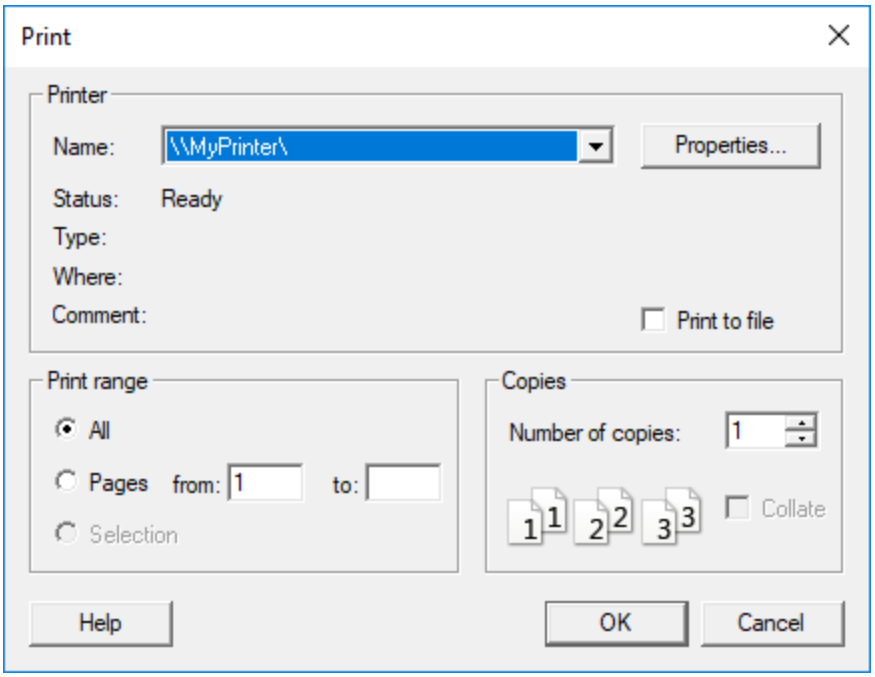
Printing

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 2026 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.

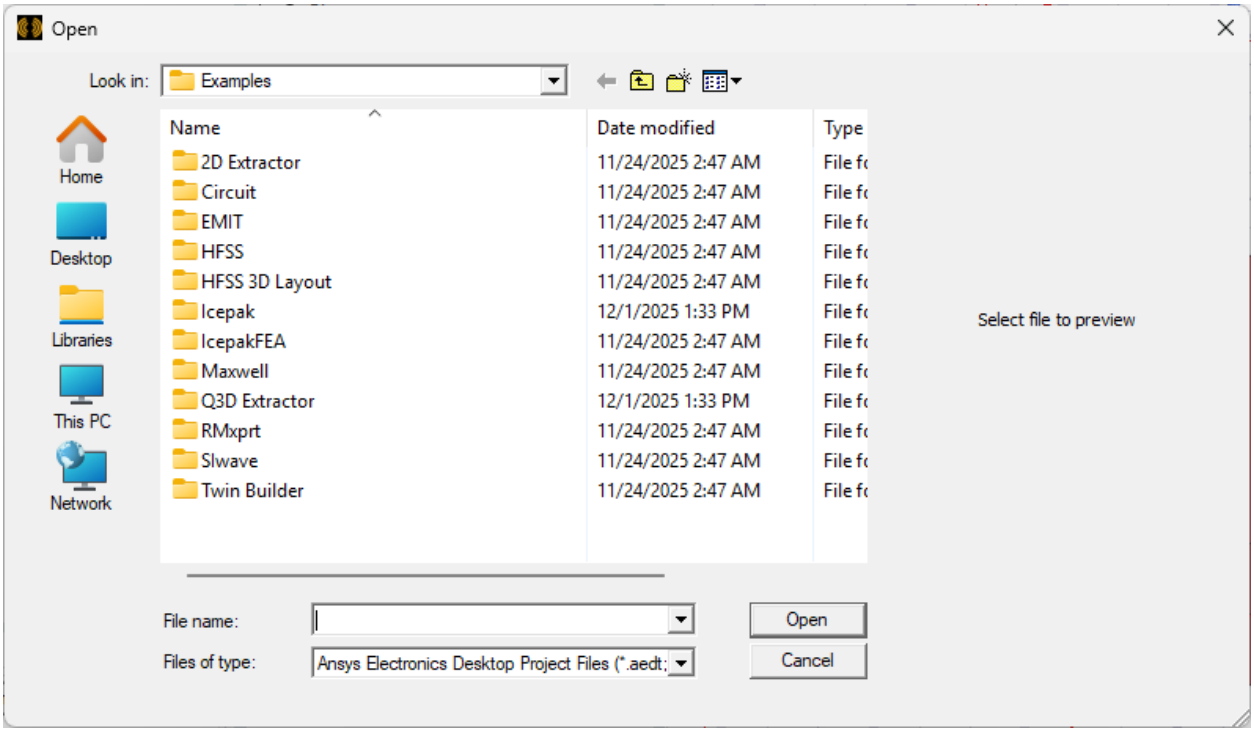
To change additional print settings, click **File** > **Page Setup**.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in **Icepak**. 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.

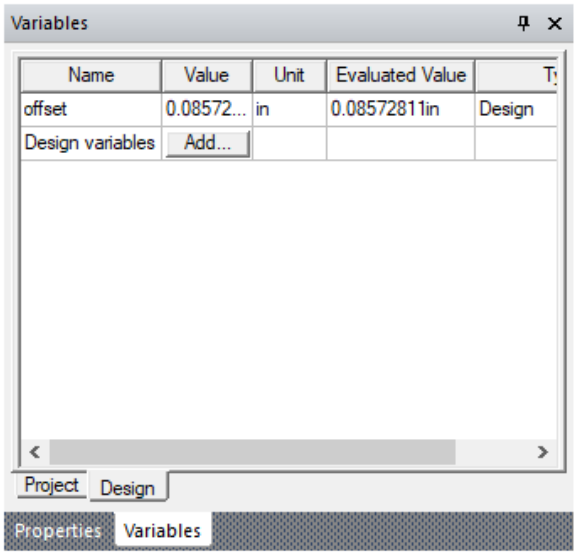
Variables can be regular variables, array index variables, or arrays. A project variable's value can reference other project variables or intrinsics. Values can be a number, variable, or mathematical expression.

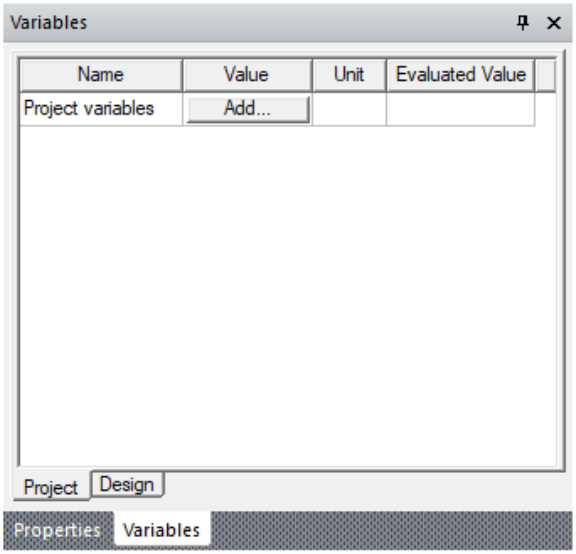
There are two types of variables in Icepak:

Project Variables	A project variable can be assigned to any parameter value in the Icepak project in which it was created. Icepak 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 Icepak will automatically append the project variable's name after you define the variable. Project variables can be designated as Design, ArrayIndex or Separator variables but not as Post-Processing Variables.
Design (Local) Variables	A design variable can be assigned to any parameter value in the Icepak 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 and as Post-Processing 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





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 Icepak project in which it was created. Icepak 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 Icepak 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** – predefined variables that cannot be changed. See: List of Intrinsic Variables.
- **Constants** – predefined constants that cannot be changed. See: List of Constants.

Adding a Project Variable

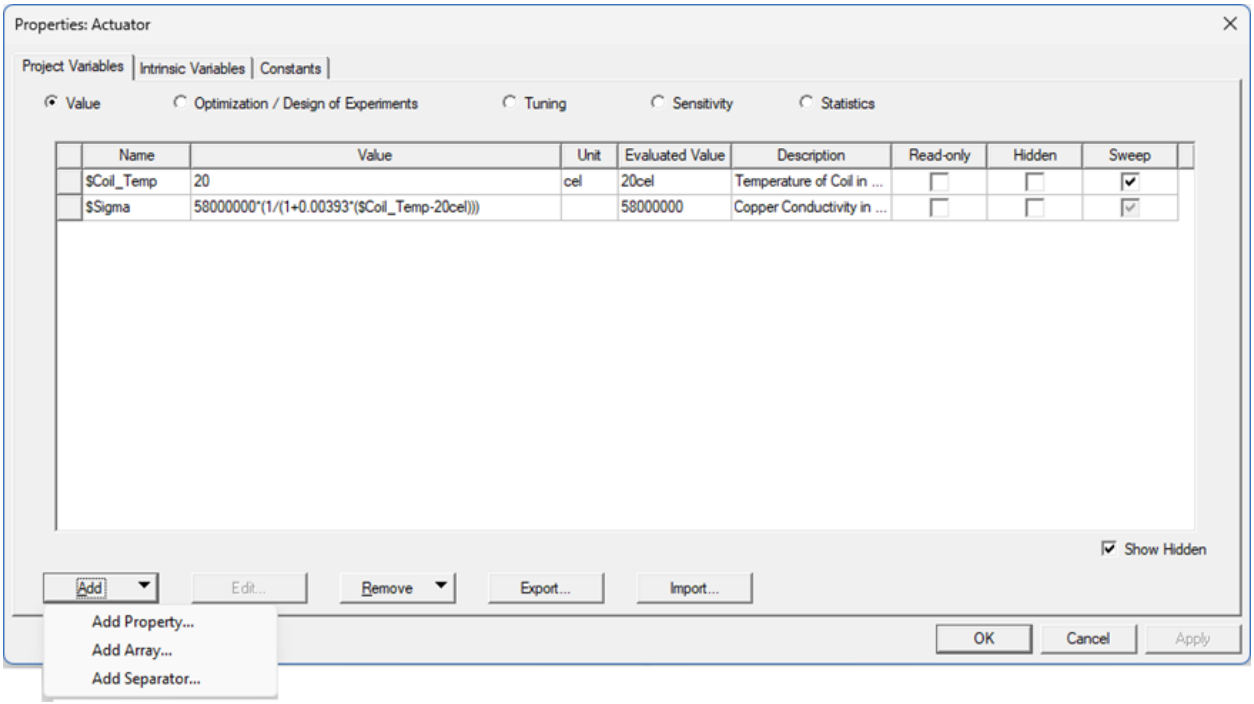
To define a project variable:

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

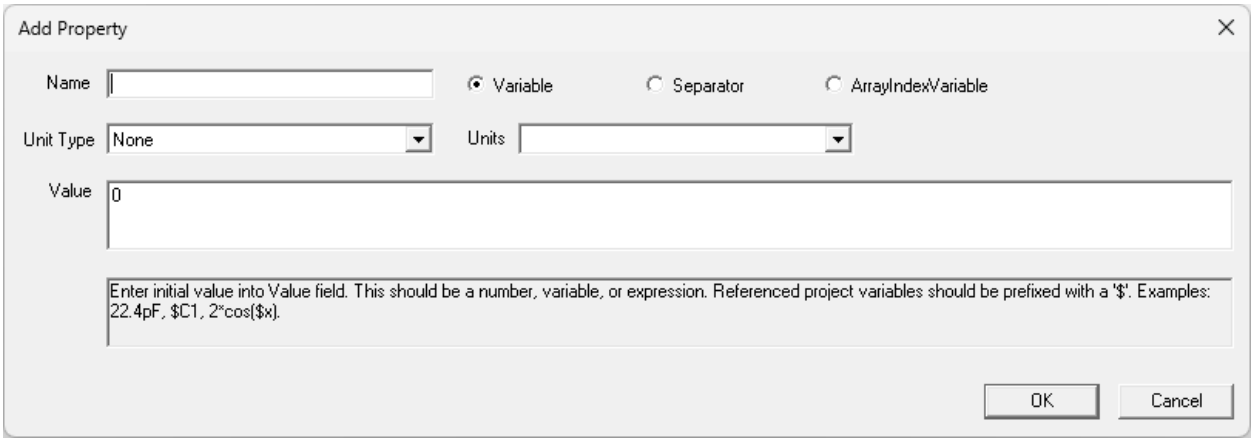
Design Xplorer Setup. Click **Edit Variables** and from the menu select **Edit Project Variables**.

The **Properties** window appears, on the **Project Variables** tab.



2. From the **Project Variables** tab, click **Add Property**.

   The **Add Property** dialog box appears.



Ansys Electromagnetics Suite 2026 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 predefined 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 array index variables, 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 array index variables, 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 2026 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 array index variables, 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.

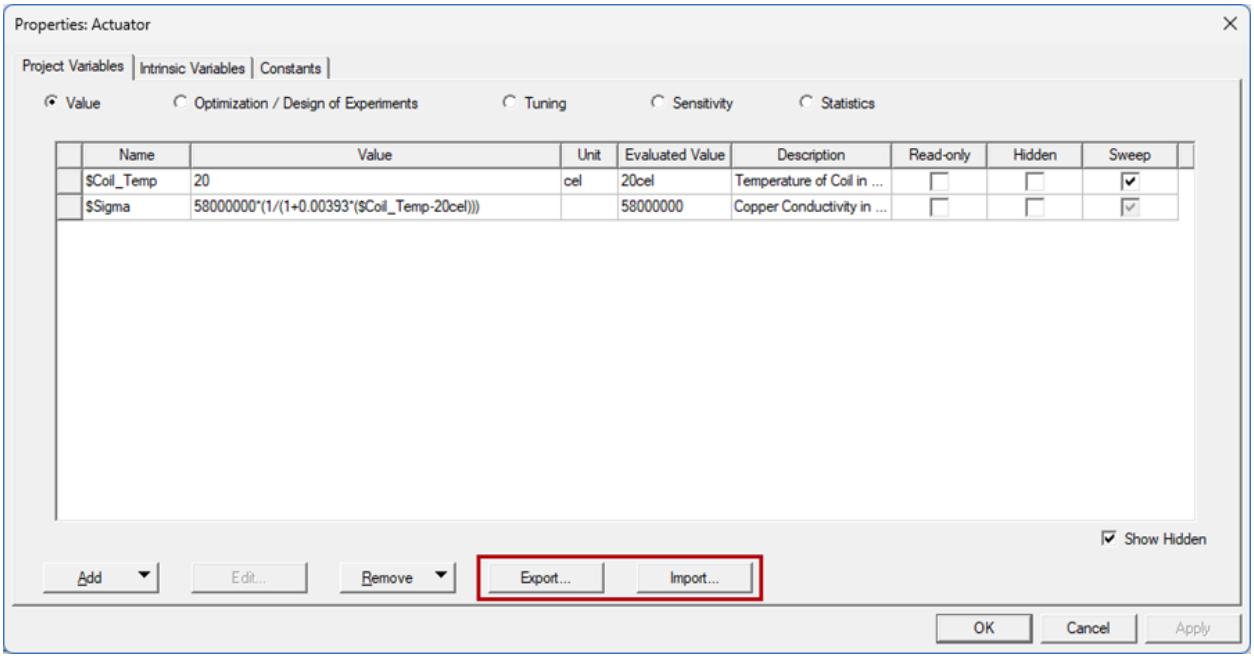
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show 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.

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** dialog box for project variables include **Import...** and **Export...** buttons.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you click **Export...** a dialog box opens that 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.

2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

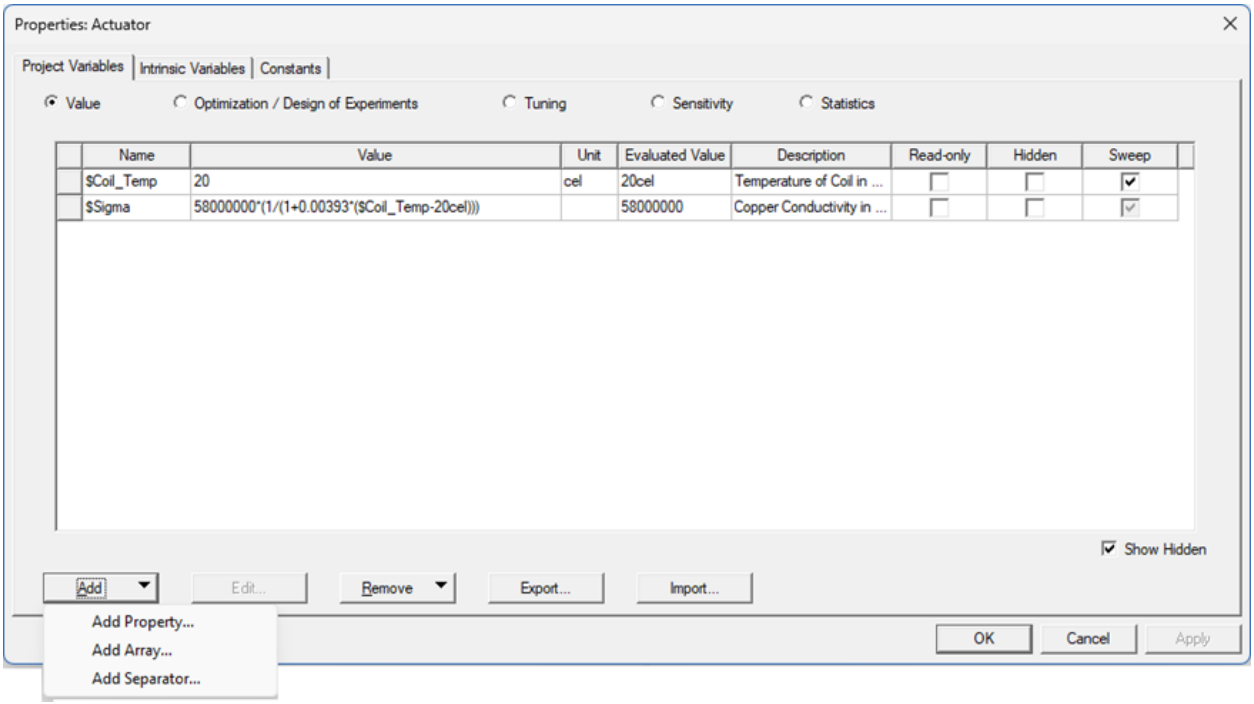
> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 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 dialog boxes: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, or **Design Xplorer Setup**, click **Edit Variables** and from the menu select **Edit Project Variables**.

   The **Properties** window appears, on the **Project Variables** tab.

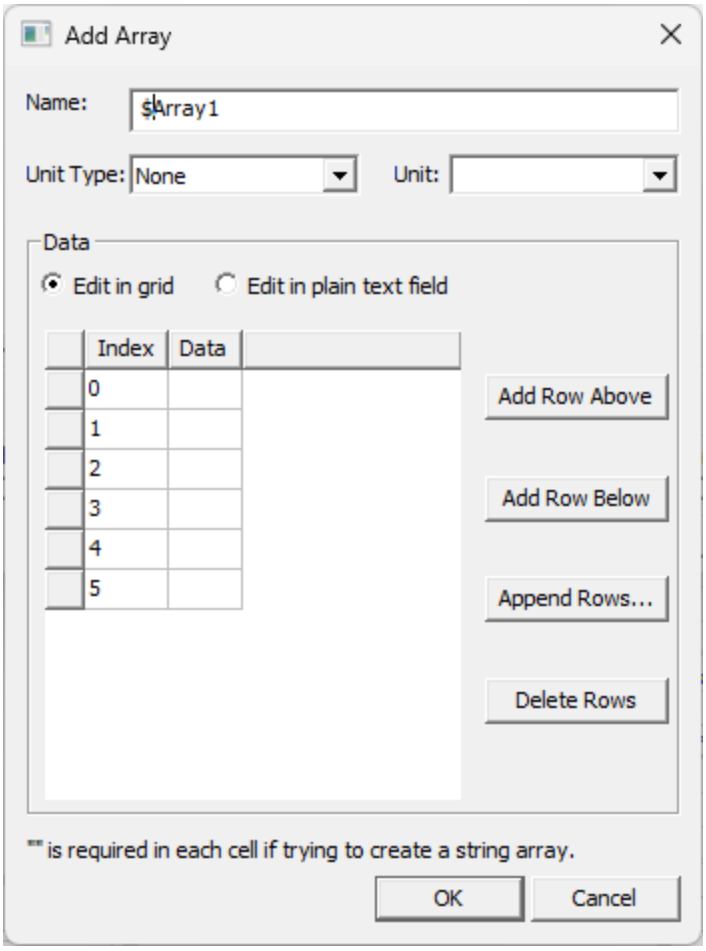
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add Array** > **Add Array...**.

   The **Add Array** window appears.

Ansys Electromagnetics Suite 2026 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 delimiters 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.

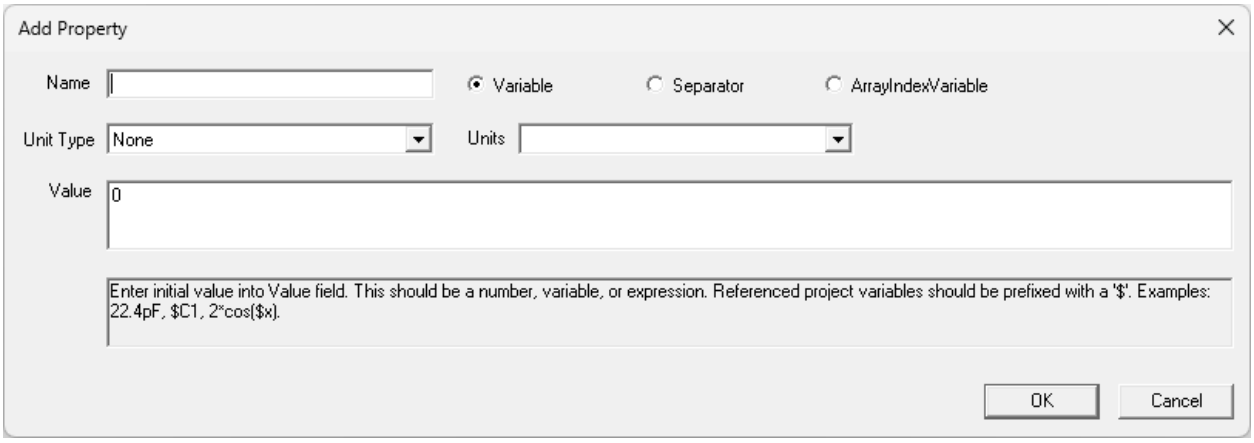
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 predefined 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

List of Intrinsic Variables

The Ansys Electronics Desktop software 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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Unit	Description
FNoi	GHz	Offset noise frequency in harmonic balance noise analysis.
Freq	Hz	Frequency for analysis and postprocessing; cannot be set by the user
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)
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 Icepak 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. Choosing **Add Property** pops up an **Add Property** dialog with default radio button on Variable; choosing **Add Array** pops up an **Add Array** dialog; choosing **Add Separator** pops up an **Add Property** dialog with default radio button on Separator.

The **Design Properties** menu item allows you to access up to three tabs, depending on your design:

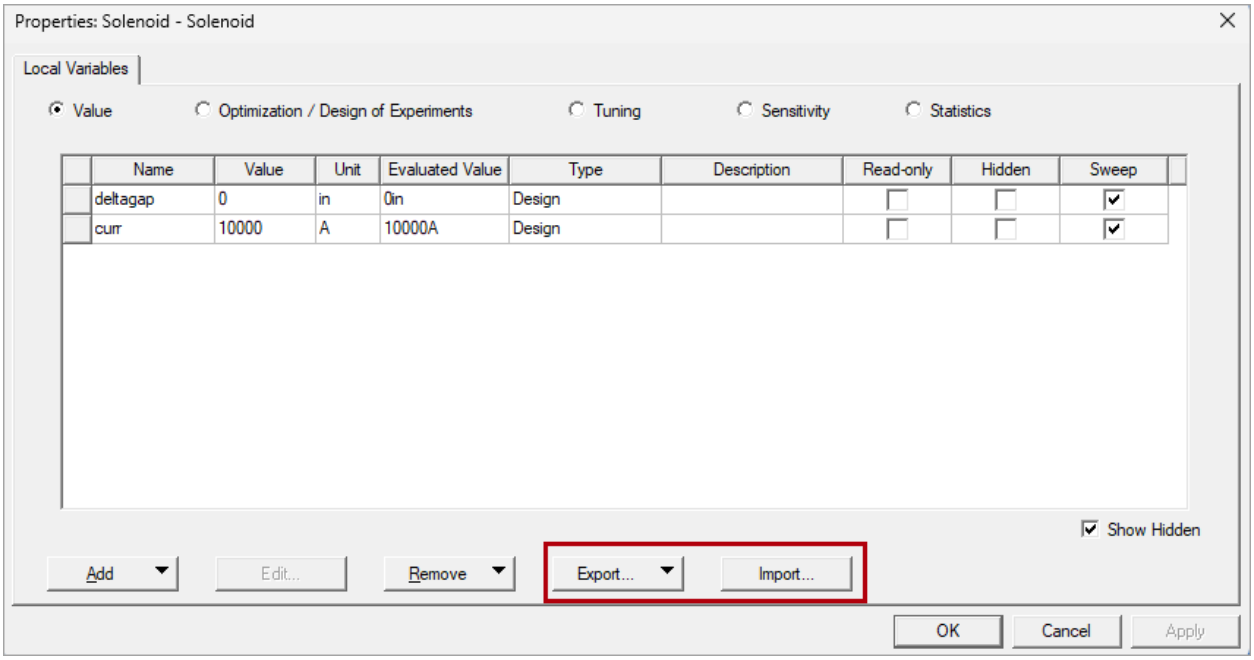
- **Local Variables** – variables, which you define, that apply to the current design only. Local variables can be defined at the design level.
- **General** – predefined design variables that cannot be changed. You cannot use the names of these variables when creating a new variable.

> **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 2026 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

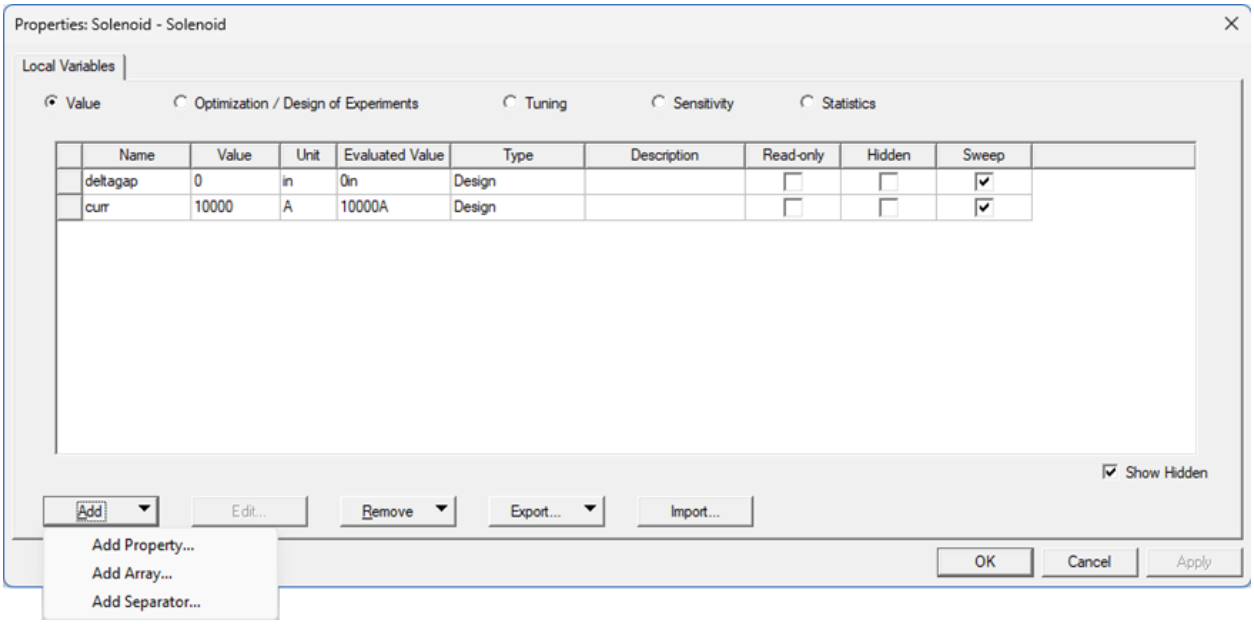
Adding a Local Variable

To define a local variable:

1. Access **Design Properties** one of three ways:
   - Click **Icepak** > **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 windows: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, or **Design Xplorer Setup**, click **Edit Variables** and from the menu select **Edit Design Variables**.

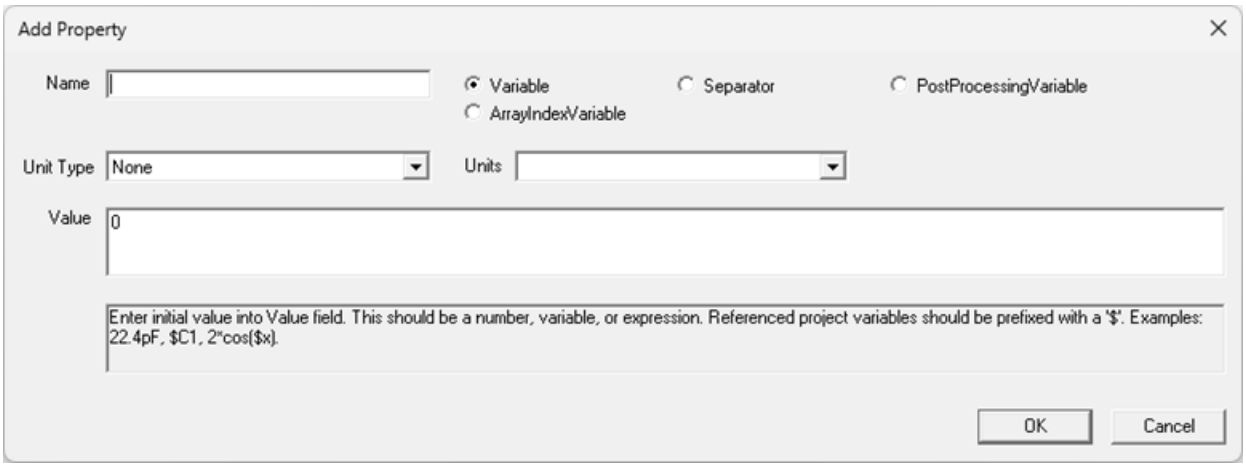
   The **Properties** window appears with the **Local Variables** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add** > **Add Property**.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Not all variable types are available for all designs.

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 postprocessing variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for variables and postprocessing variables, use the drop-down menu to select a unit of measure.
- **Value** – for variables, array index variables, and postprocessing variables, 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 array index variables, 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 array index variables, select an associated array from the drop-down list. You must have previously created an array.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

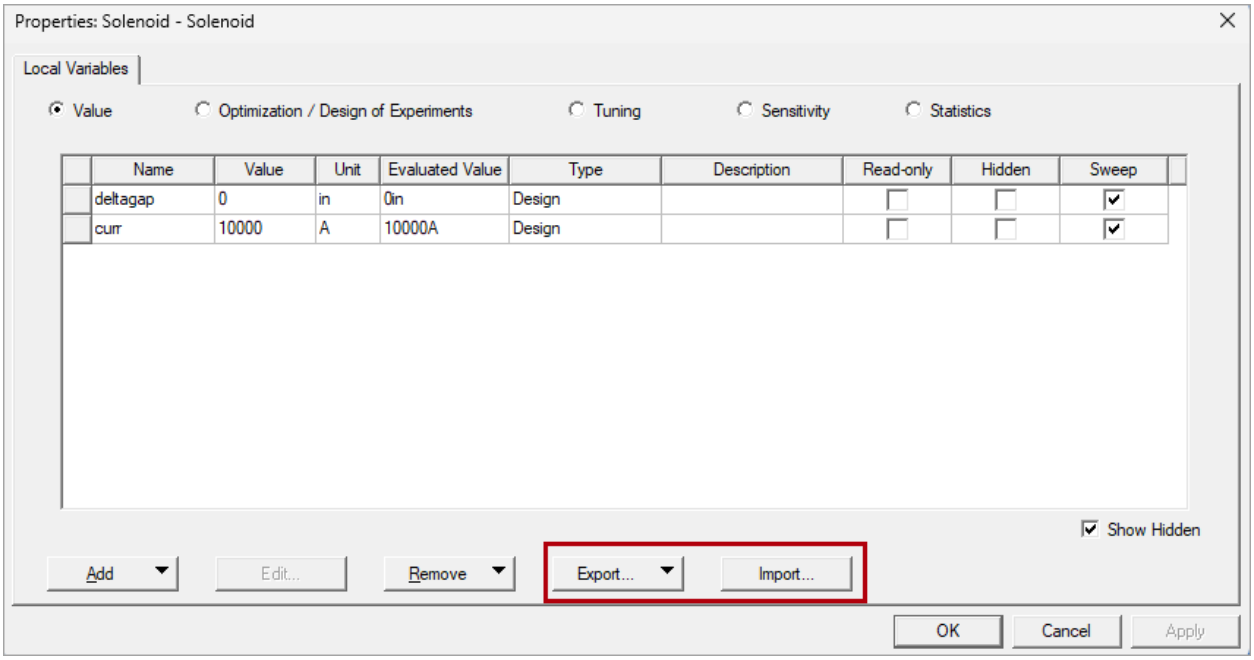
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
   - **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 settings 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Deleting a Local Variable

To remove a Local Variable:

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 optimetric 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Icepak 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:

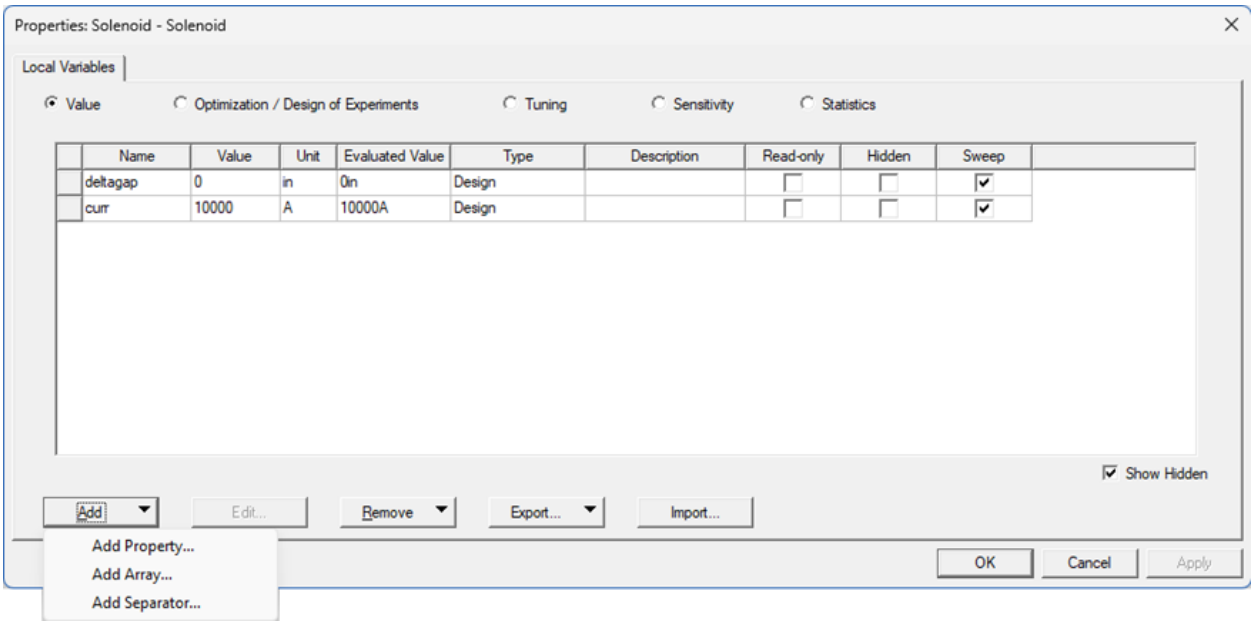
1. Access **Local Variables** one of three ways:
   - Click **Icepak** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, or **Design Xplorer Setup**, click **Edit Variables** and from the menu select **Edit Design Variables**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Properties** window appears.

2. Select the **Local Variables** tab.

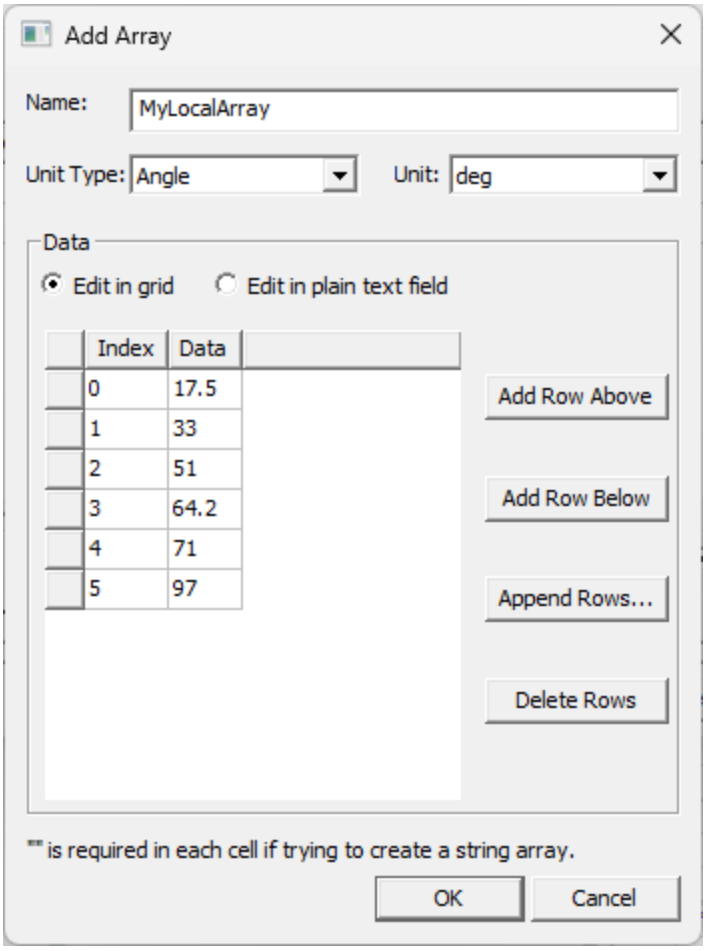
   The **Local Variables** tab displays.



3. Click **Add** > **Add Array**.

   The **Add Array** window appears.

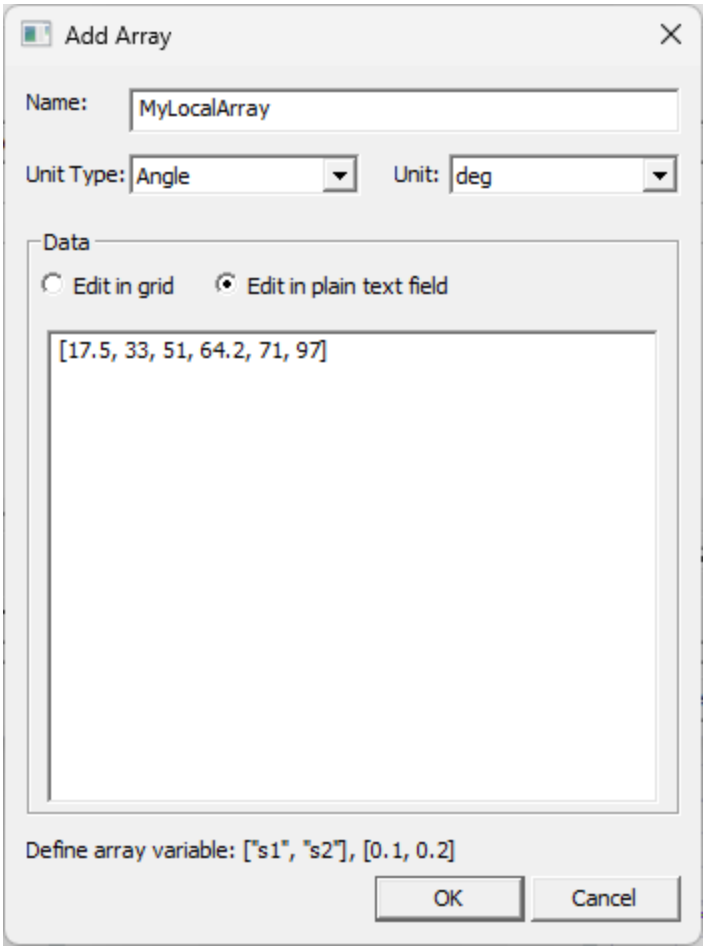
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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 delimiters when array values are entered in either the grid or text field.

6. When you have finished entering values, click **OK**.

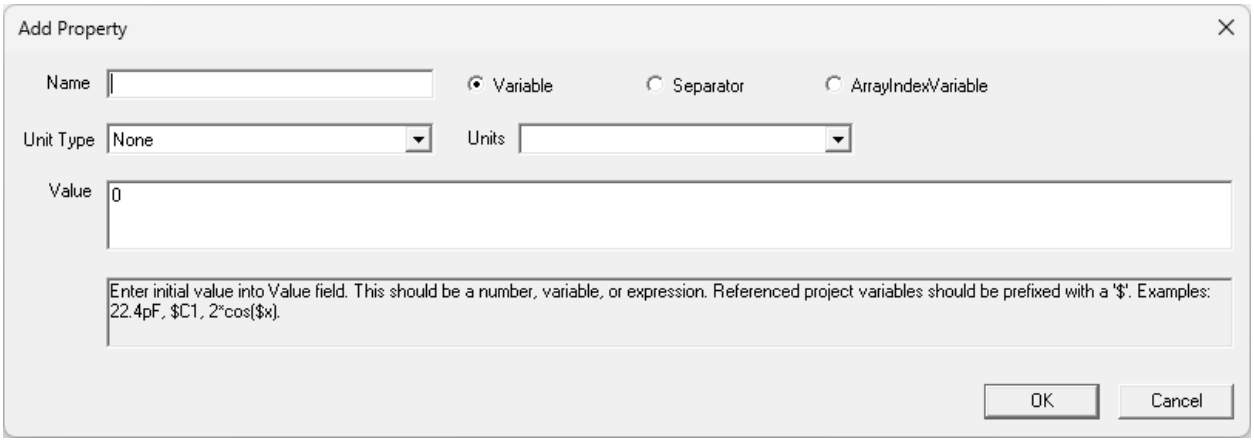
   The array appears in the **Local Variables** list.

You can add a variable to the array by adding an ArrayIndexVariable on the **Local Variables** tab.

Ansys Electromagnetics Suite 2026 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 **Icepak** > **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 predefined 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 2026 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**.

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.

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

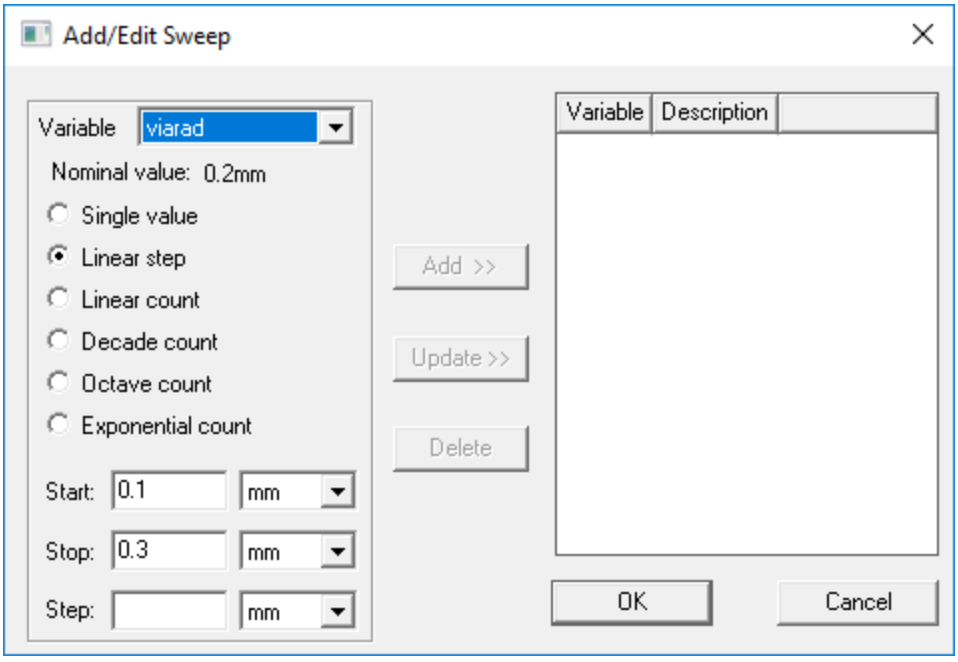
When setting up a sweep, the **Setup** window contains a list of variables being swept.

To add a non-fixed variable sweep:

1. Click **Add**.

   The **Add/Edit Sweep** window appears.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choosing a Variable to Optimize

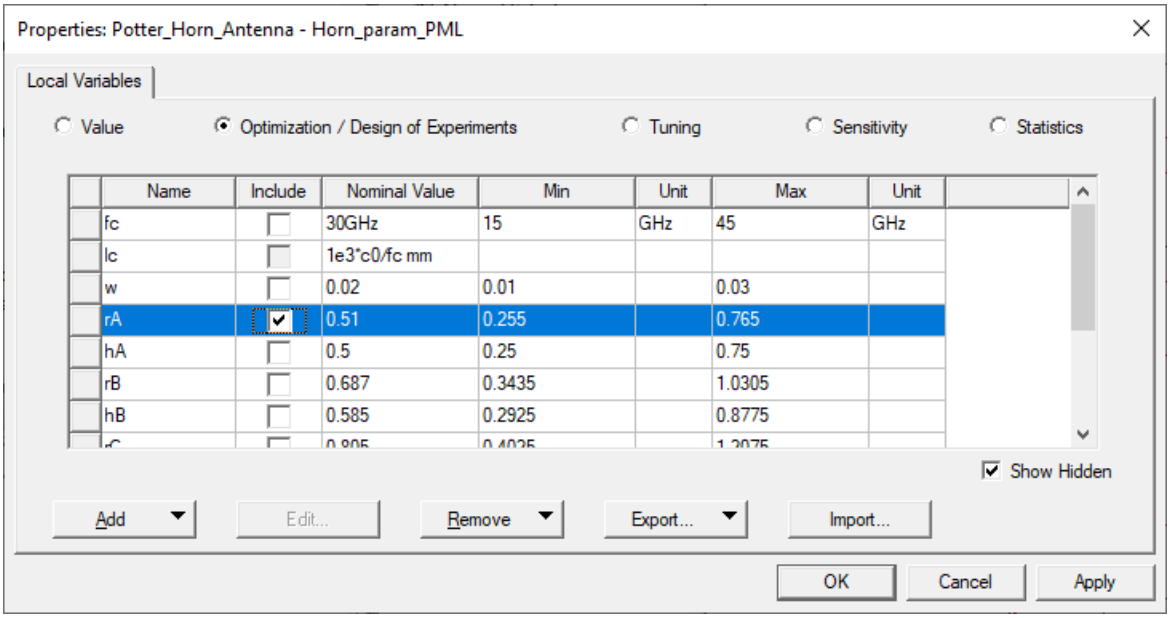
Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** window.

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 (**Icepak** > **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.
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 2026 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.

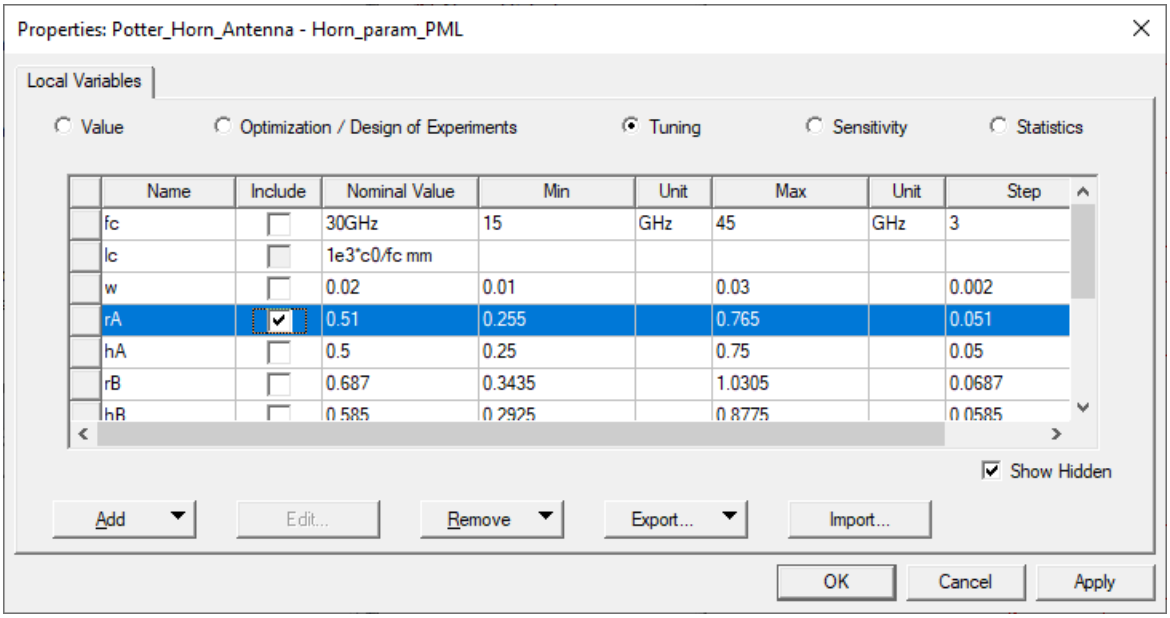
To specify that a variable is to be tuned:

1. Navigate either to the project variables **Properties** window (**Project** > **Project Variables**), or to the design variables **Properties** window (**Icepak** > **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.



Ansys Electromagnetics Suite 2026 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 tuning.

Including a Variable in a Sensitivity Analysis

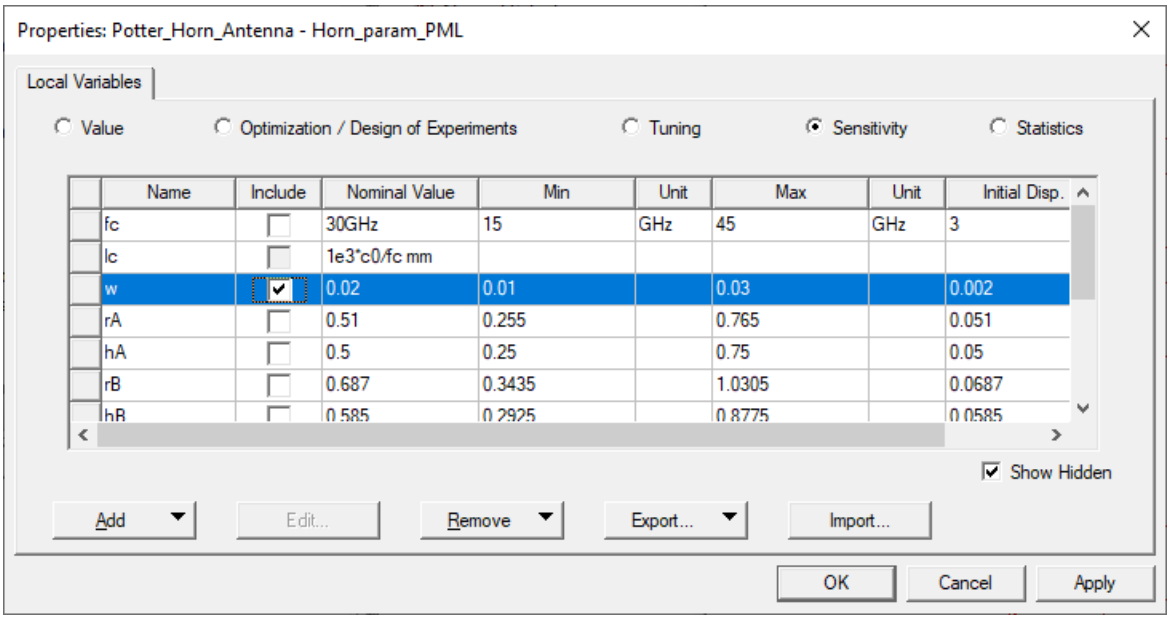
Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** window.

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 (**Icepak** > **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.

Ansys Electromagnetics Suite 2026 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 sensitivity analysis in the current design or project.

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:

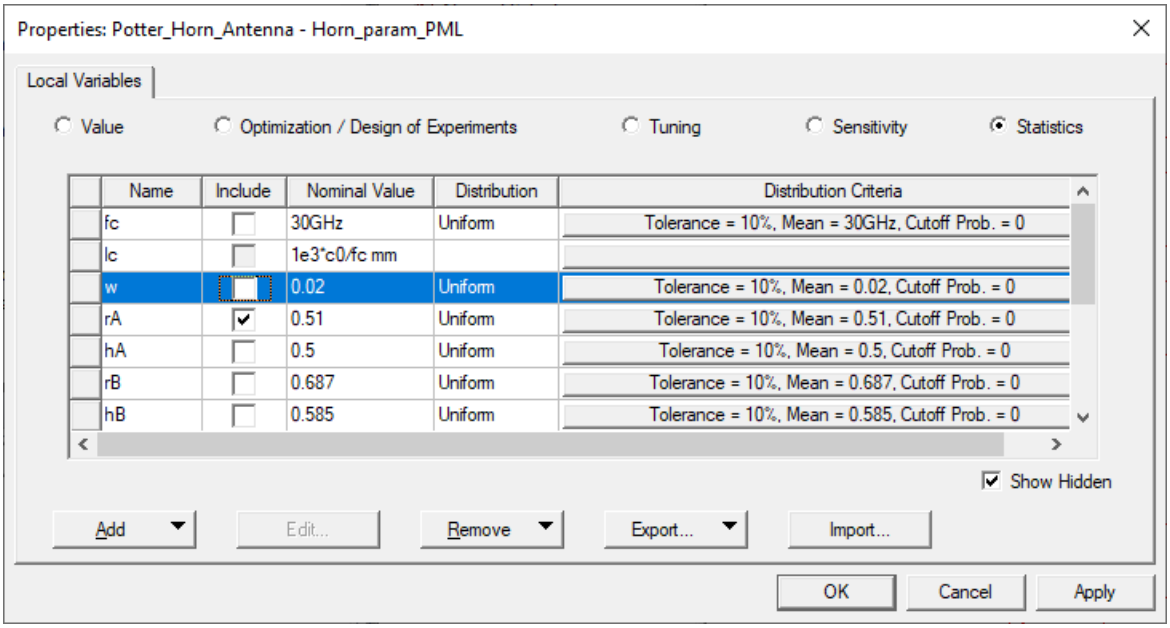
1. Navigate either to the project variables **Properties** window (**Project** > **Project Variables**), or to the design variables **Properties** window (**Icepak** > **Design Properties**).

   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 2026 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 2026 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 Icepak 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 **Icepak** > **Design Datasets**.

Adding Datasets

The following procedure describes how to add a dataset manually.

You can also add datasets:

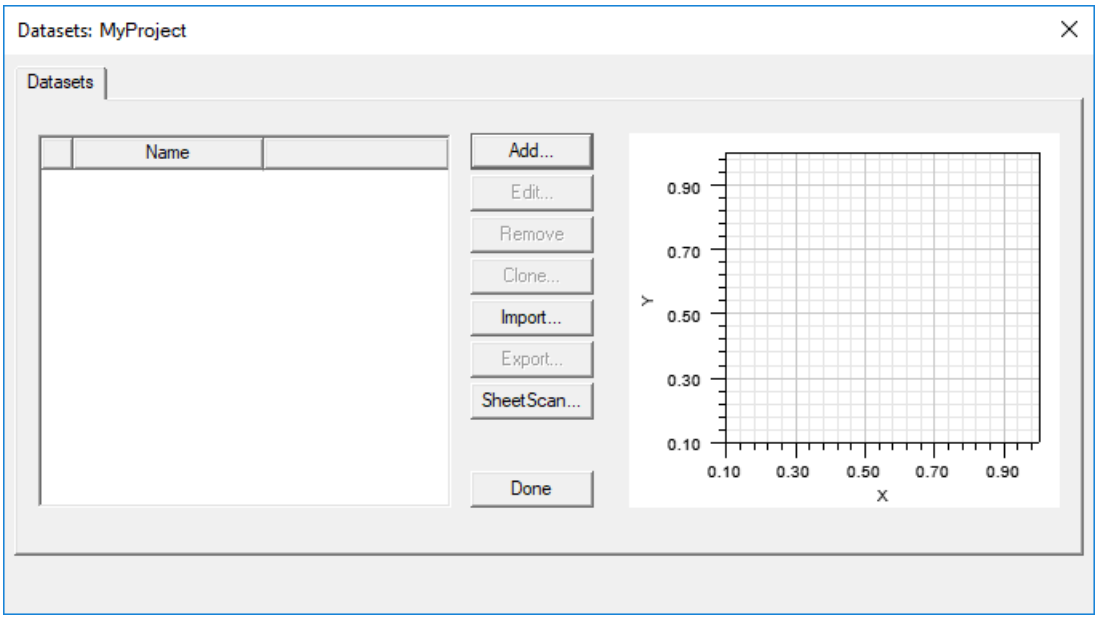
- By importing data from an external tab-delimited file using the **Import...** button.
- Using the Sheetscan tool.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

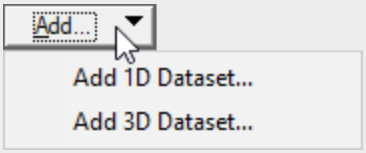
To add a dataset manually:

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 **Icepak** > **Design Datasets** from the menu bar.

   The **Datasets** window appears, listing any project or design-level datasets.

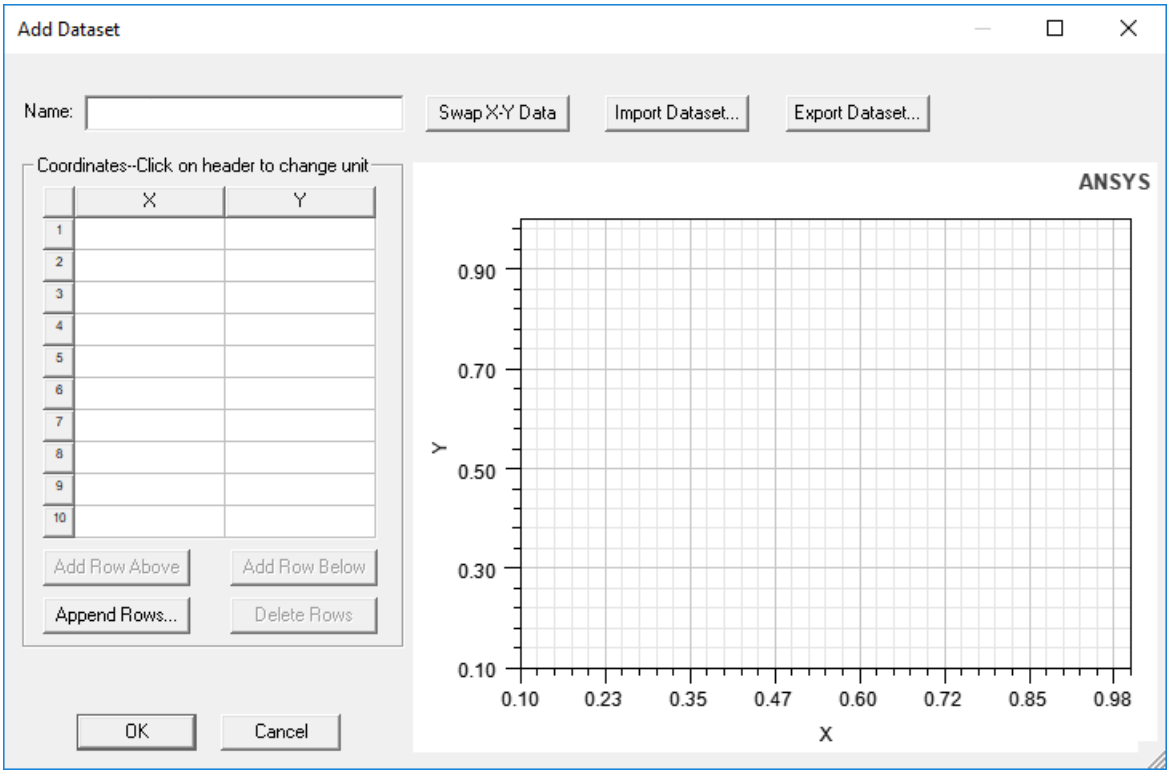


2. Click **Add**.

> **Note:**
>
> If you are adding a design-level dataset in Icepak, you will be presented the option to add either a **1D Dataset** or **3D Dataset**.
>
> 
>
>
> See: Adding a 3D Dataset.

The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2026 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.

> **Warning:**
>
> The imported dataset can only contain data; if it includes any text, such as comments, the data import will fail.

- **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the table.

- **Delete Row** – deletes the selected row or rows.

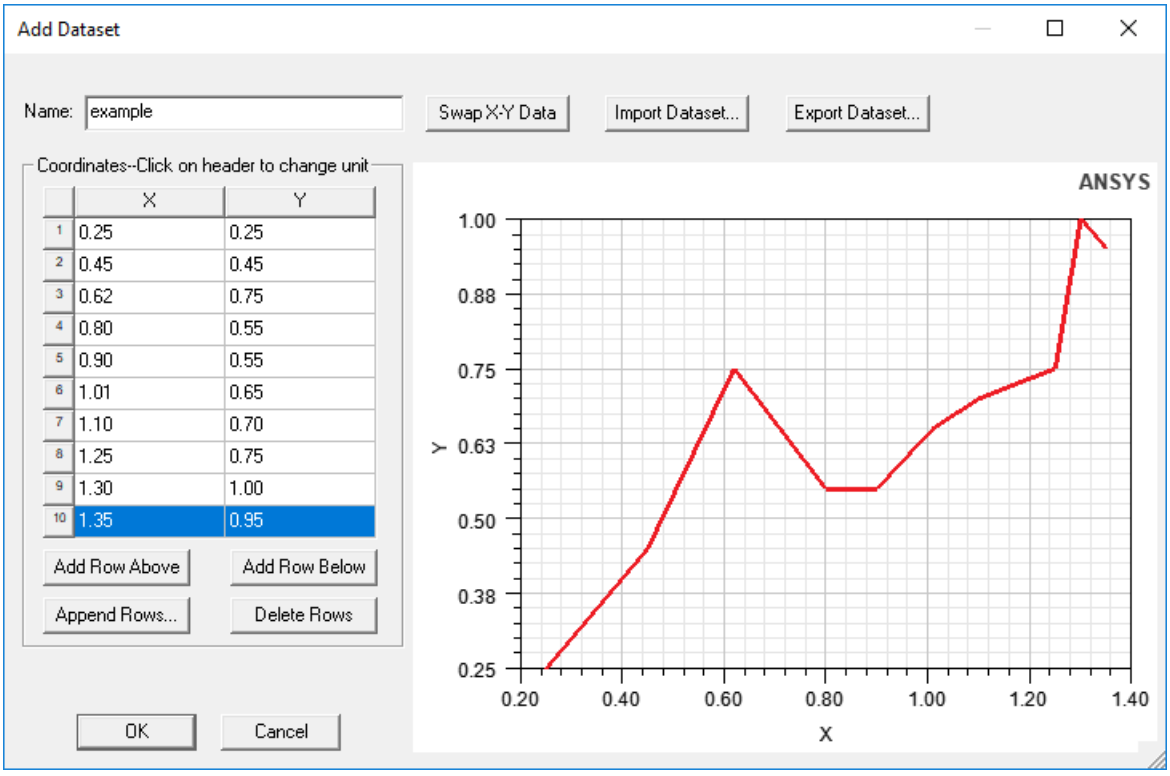
3. Enter a **Name** for the dataset.

> **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.

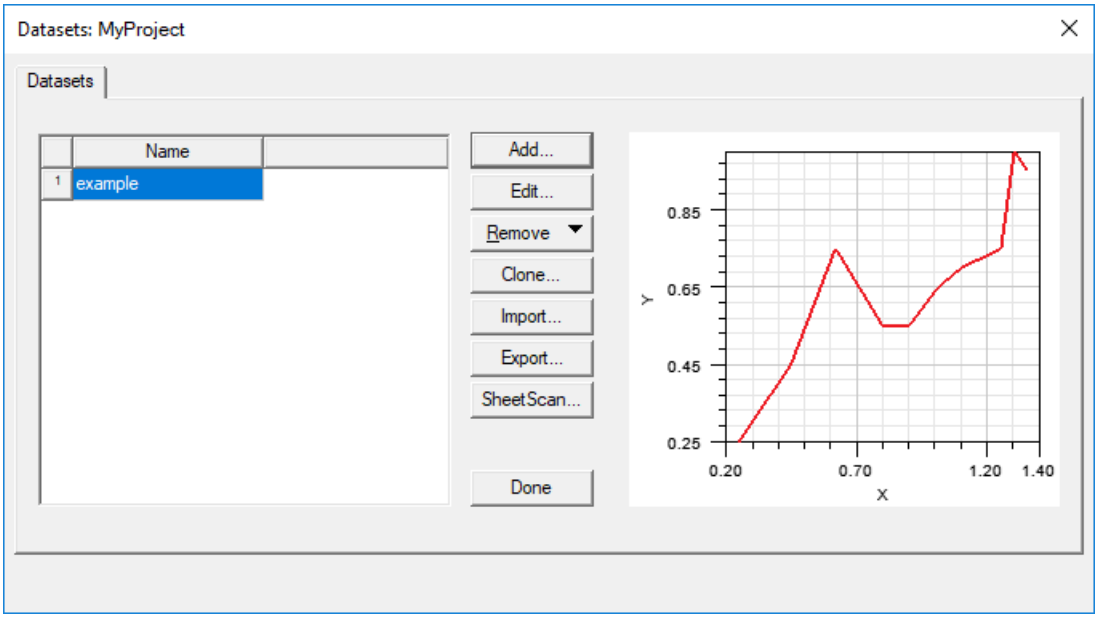
   The display updates to show your coordinates.



5. When you are finished, click **OK** to exit the **Add Datasets** window.

   The new dataset appears in the **Datasets** list.

Ansys Electromagnetics Suite 2026 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.

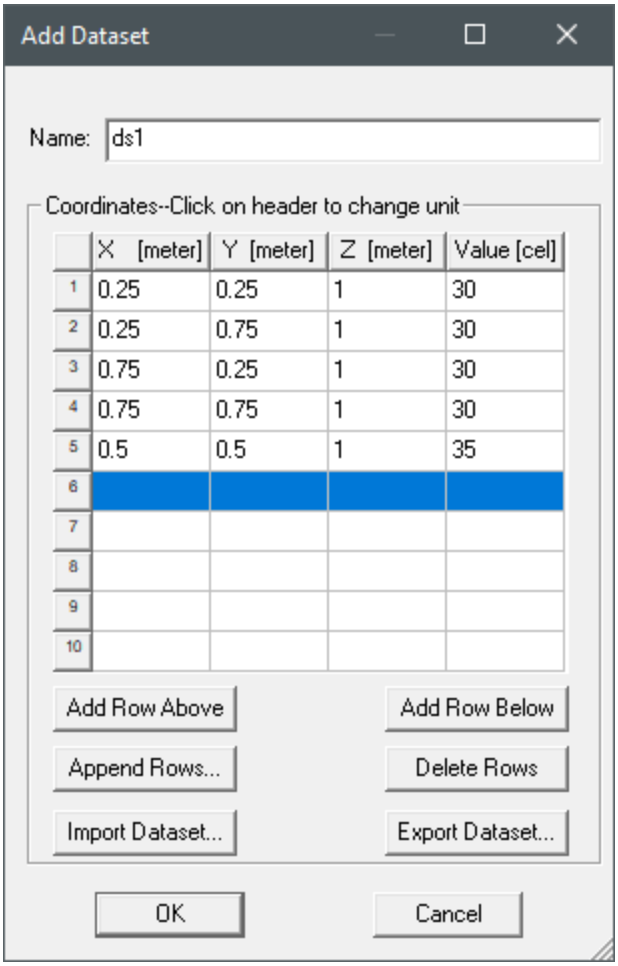
Adding 3D Datasets

HFSS, Maxwell, and Icepak designs support 3D datasets (representing a spatial or volumetric profile). Depending on which of these three design types you are using, you will be able to create only design-level 3D datasets, only project-level 3D datasets, or both.

In Icepak designs, you can define only design-level 3D datasets and assign them to certain thermal boundary condition properties and temperature patches in transient simulations.

To define a 3D dataset, specify a list of **X**, **Y**, and **Z** coordinates and the corresponding **Values** in the **Add Dataset** window. For example, the first line in the image below specifies a temperature of 30° C at (0.25, 0.25, 1). The data specifies a variation of temperature on the plane $0.25 \le X \le 0.75$, $0.25 \le Y \le 0.75$, $Z = 1$. In this example, the values in the **Values** column are temperature, but can be set to a different unit depending on the property you wish to assign. Icepak interpolates the data you provide to create a profile for the whole region of the boundary condition. To change the unit, click the **X**, **Y**, **Z**, or **Value** header row.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

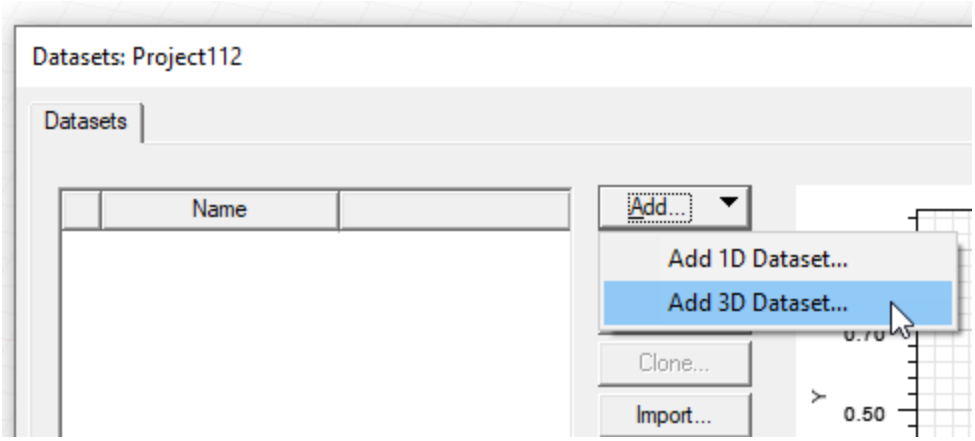


1. Using the menu bar, click **Icepak** > **Design Datasets**.

   The **Datasets** dialog box appears. This lists any existing datasets.

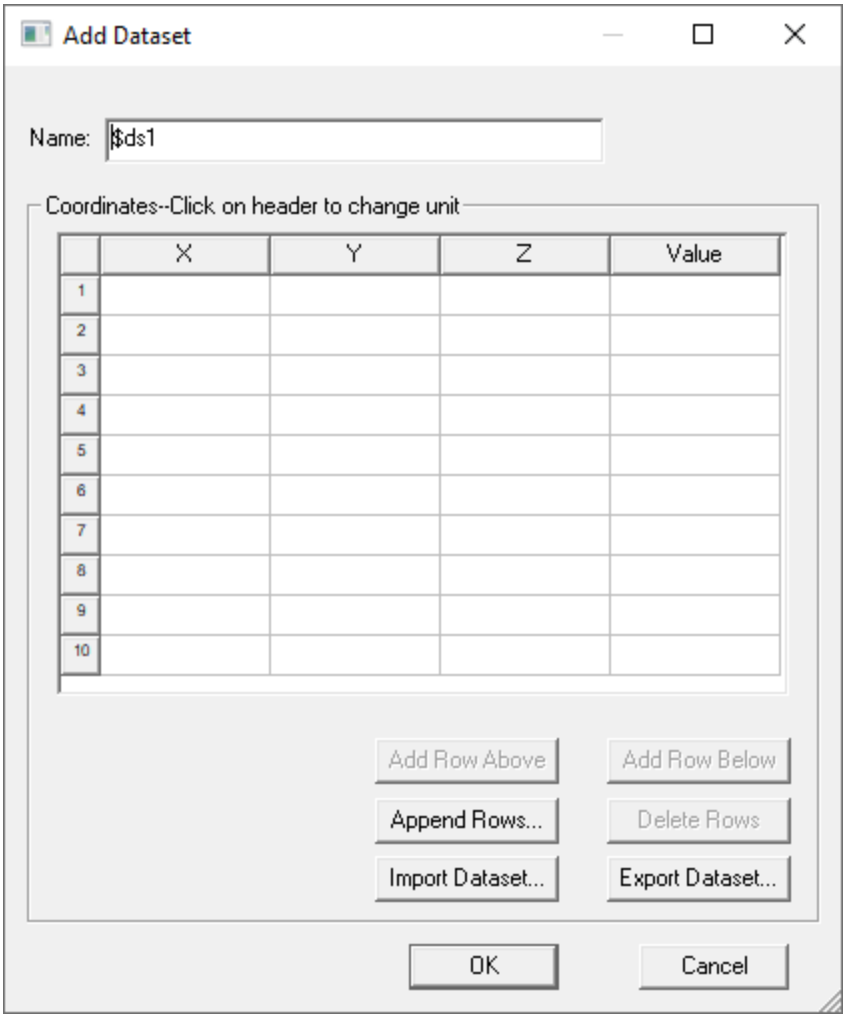
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Add** > **Add 3D Dataset**.



The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The window contains fields for the dataset name and a table for X, Y, and Z coordinates and corresponding values. It can be resized to display additional rows and columns, and includes buttons for the following functions:

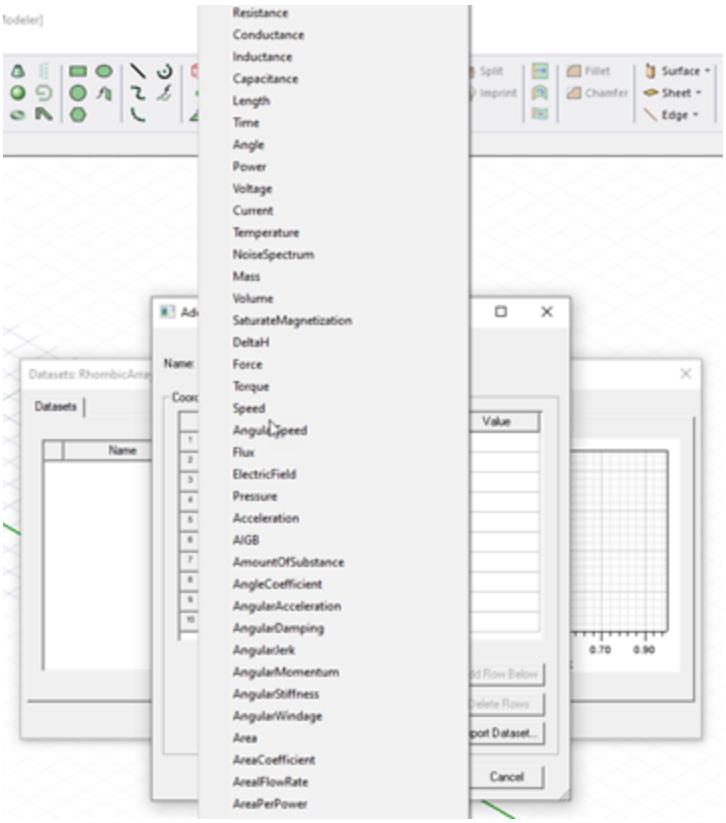
- **Add Row Above** – adds a new row to the table above the selected row.
- **Add Row Below** – adds a new row to the table below the selected row.
- **Append Row** – opens a dialog box that lets you specify a number of rows to add to the table.
- **Delete Row** – deletes the selected row or rows.
- **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> The imported dataset can only contain data; if it includes any text, such as comments, the data import will fail.

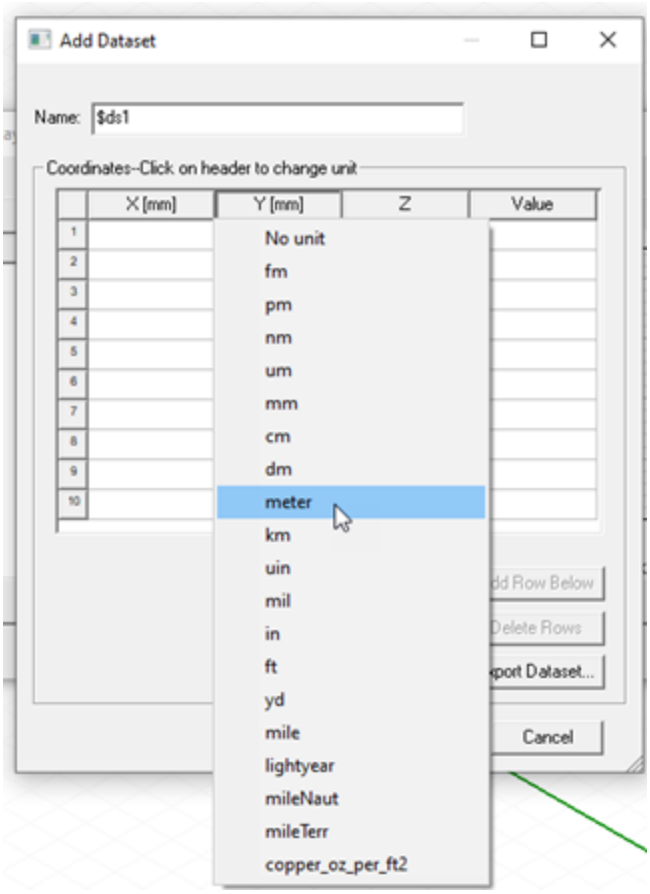
- **Export Dataset** – this provides a way to export the current dataset to a tab separated points file. Clicking the button opens a file browser window.

3. Optionally, type a name other than the default for the dataset in the **Name** field.
4. Click the **X**, **Y**, **Z** and **Value** column headers to select a unit category. The default is No Unit.



Click the header again to select a unit in that category. You can select No unit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selected units are displayed for each column in the header cell.

5. Enter the X, Y, and Z coordinates and the corresponding values by one of the following methods:
   - **Import Dataset**
   - Type coordinates and associated value in the **X**, **Y**, **Z**, and **Value** columns.
6. When you are finished entering the data point coordinates and values, click **OK**.

> **Note:**
>
> When used with a clp function (Closest point interpolation) in an expression, performance will degrade with large 3D dataset.

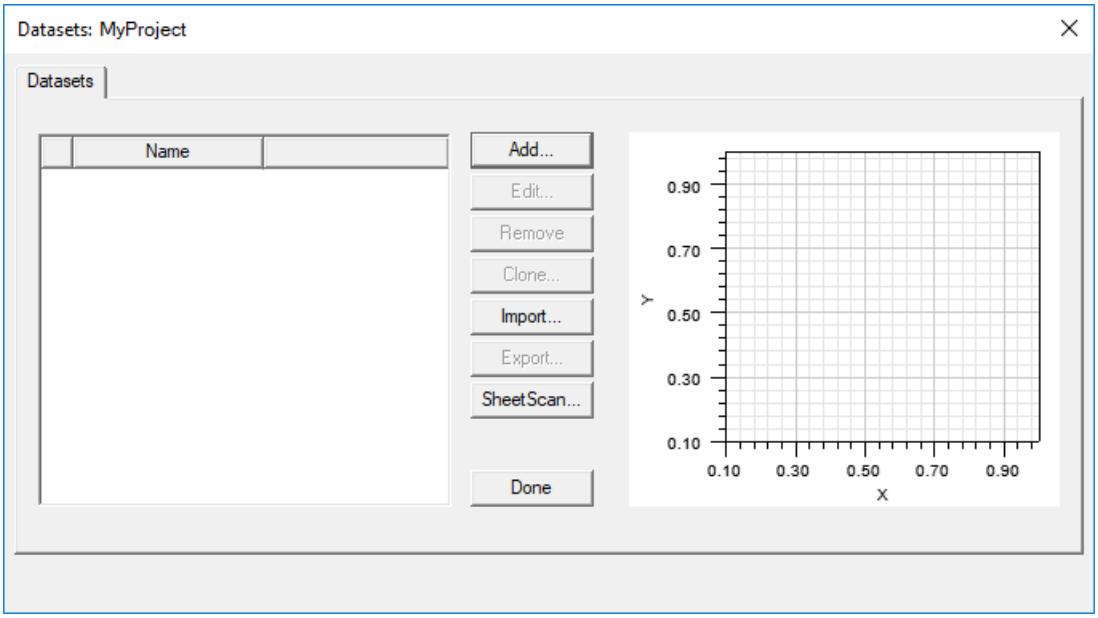
Importing Datasets

To import a dataset from a file:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Icepak** > **Design Datasets**.

   The **Datasets** window appears.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. If you select an **.mdx**, **.mda**, **.out**, **.cfg**, or **.tab** file, the data is imported immediately into **Datasets** window.

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.

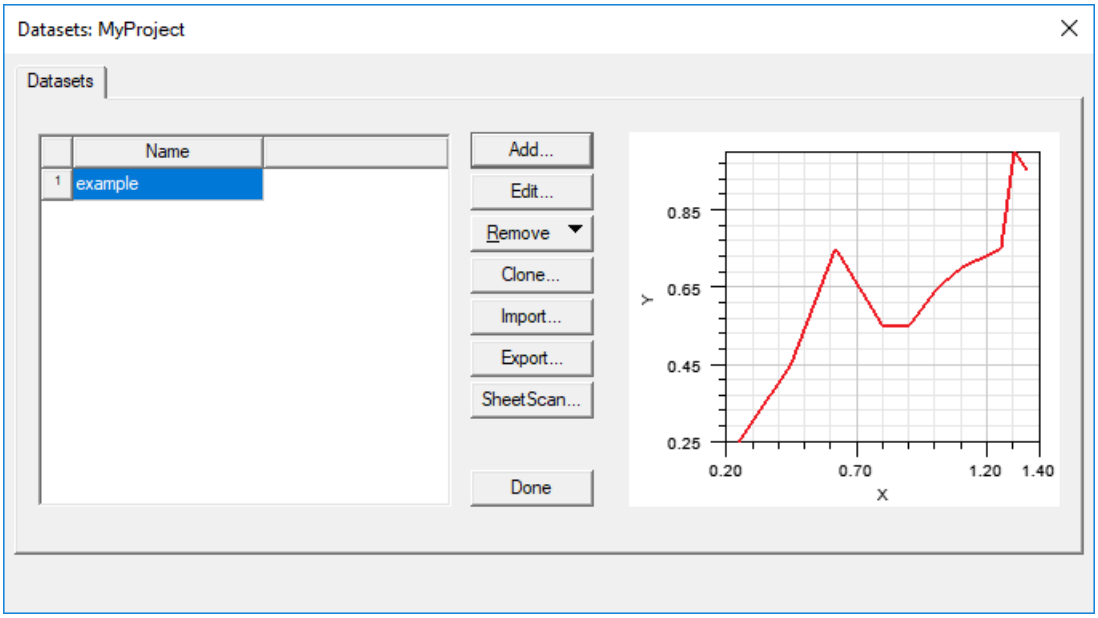
Editing Datasets

To modify an existing dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Icepak** > **Design Datasets**.

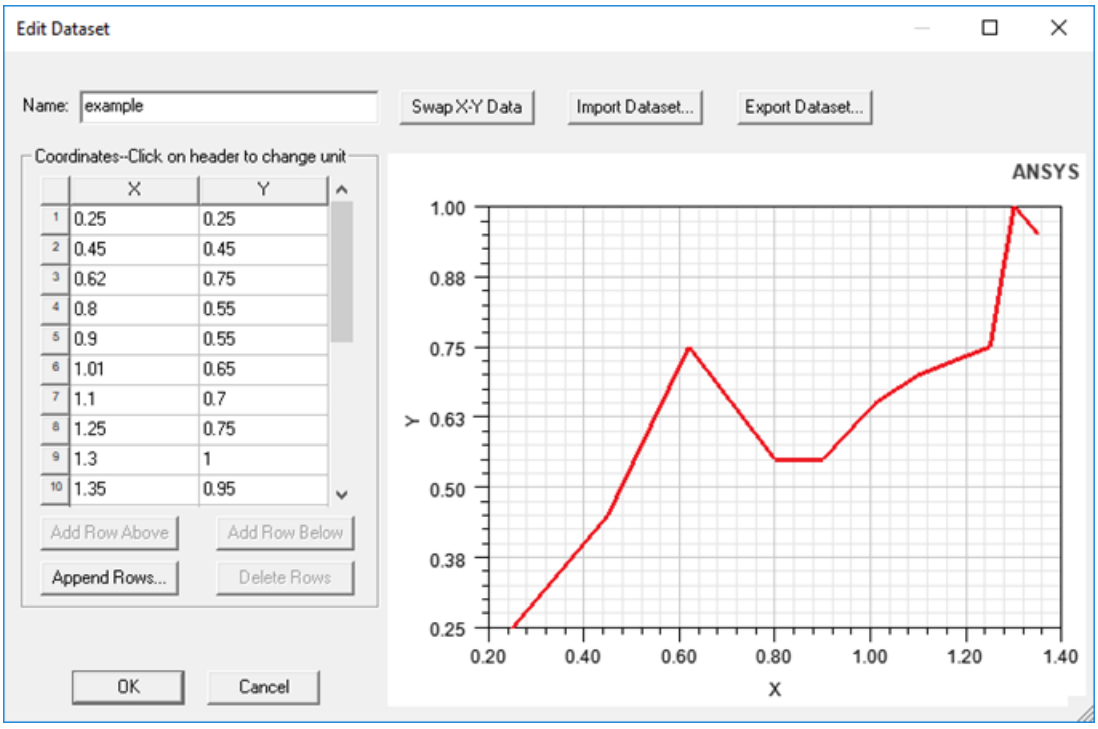
   The **Datasets** window appears, listing any project or design-level datasets.

Ansys Electromagnetics Suite 2026 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.



3. Change the **Name** and **X/Y** coordinates as desired.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. When you are finished, click **OK** to exit the **Edit Dataset** window.
5. Click **Done** to exit the **Datasets** window.

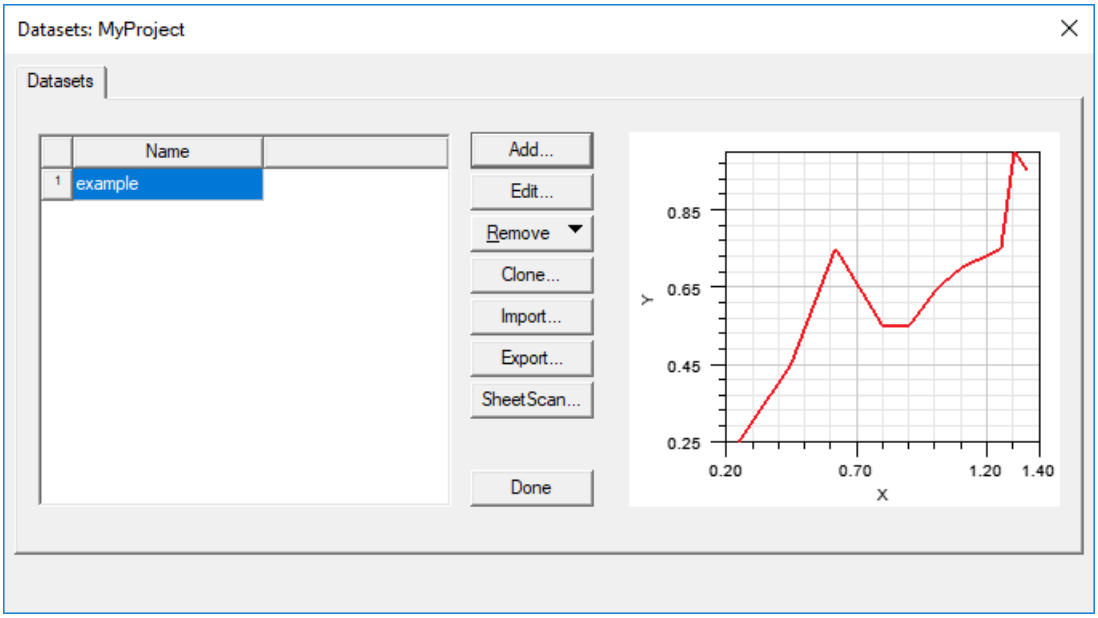
Cloning Datasets

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 **Icepak** > **Design Datasets**.

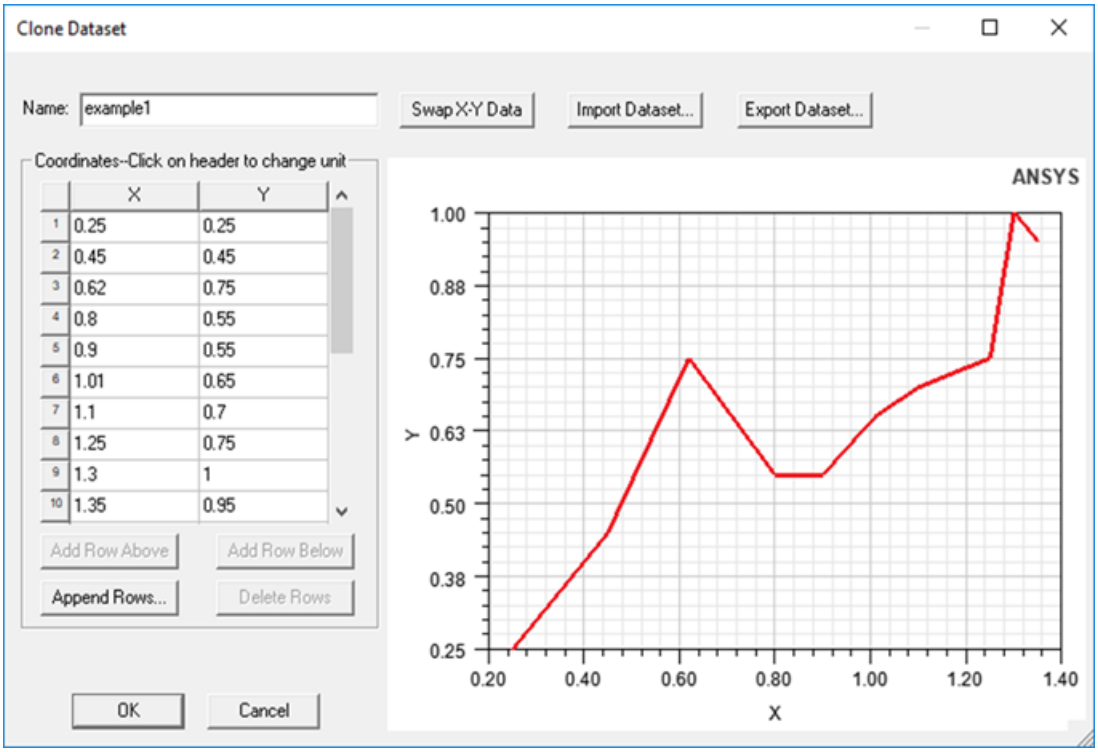
   The **Datasets** window appears, listing any project or design-level datasets.



2. Click **Clone**.

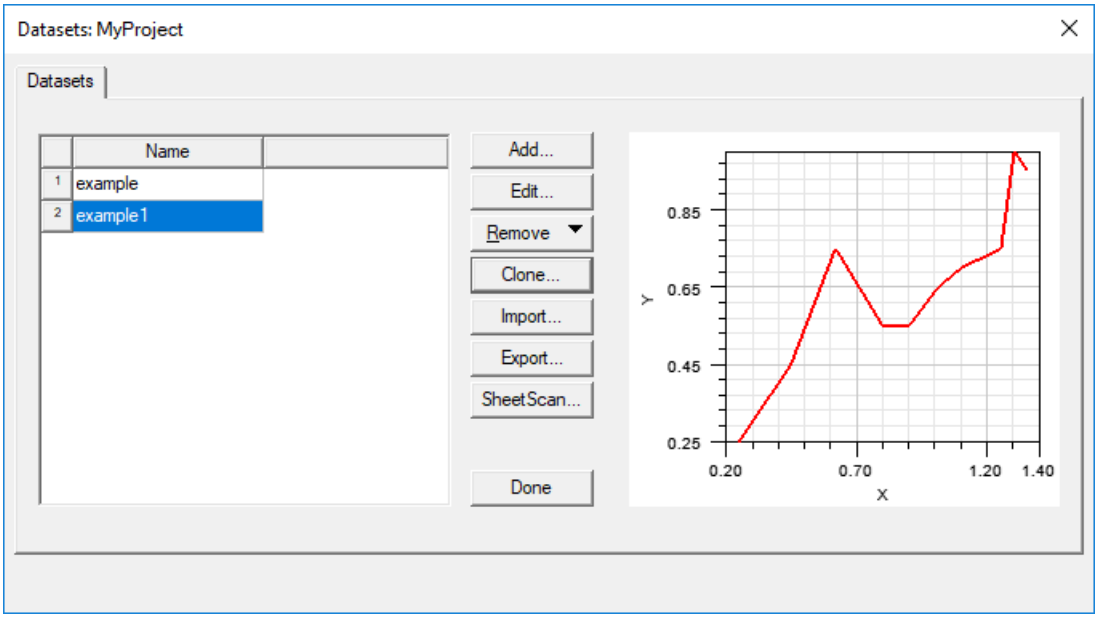
   The **Clone Dataset** window appears. The default name is the original dataset name with a number appended to the end.

Ansys Electromagnetics Suite 2026 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 2026 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.

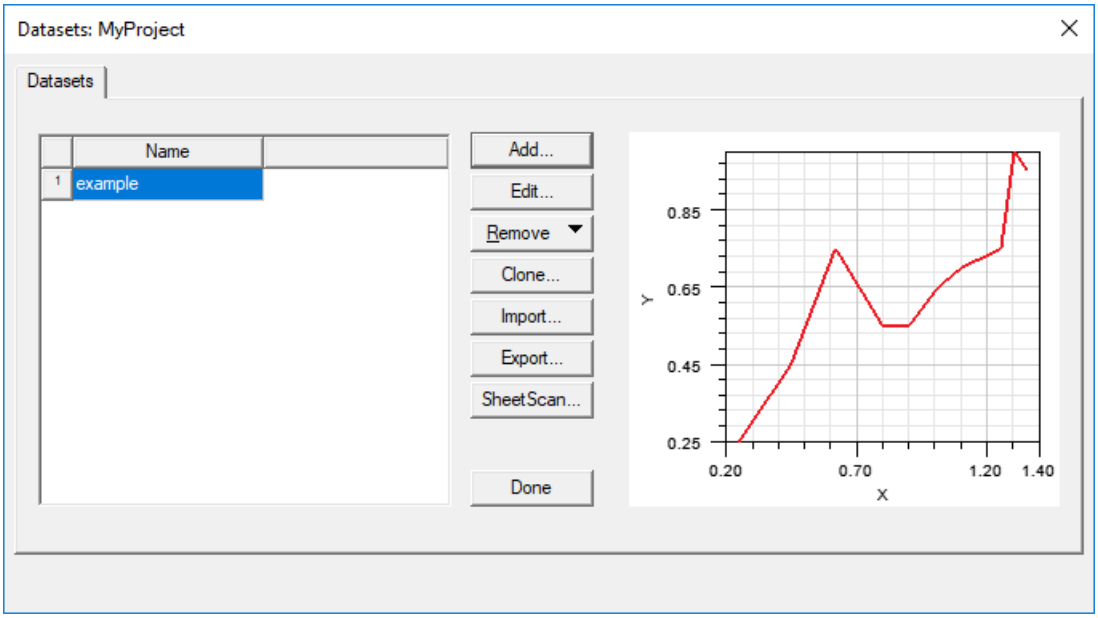
Exporting Datasets

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 **Icepak** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.



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.

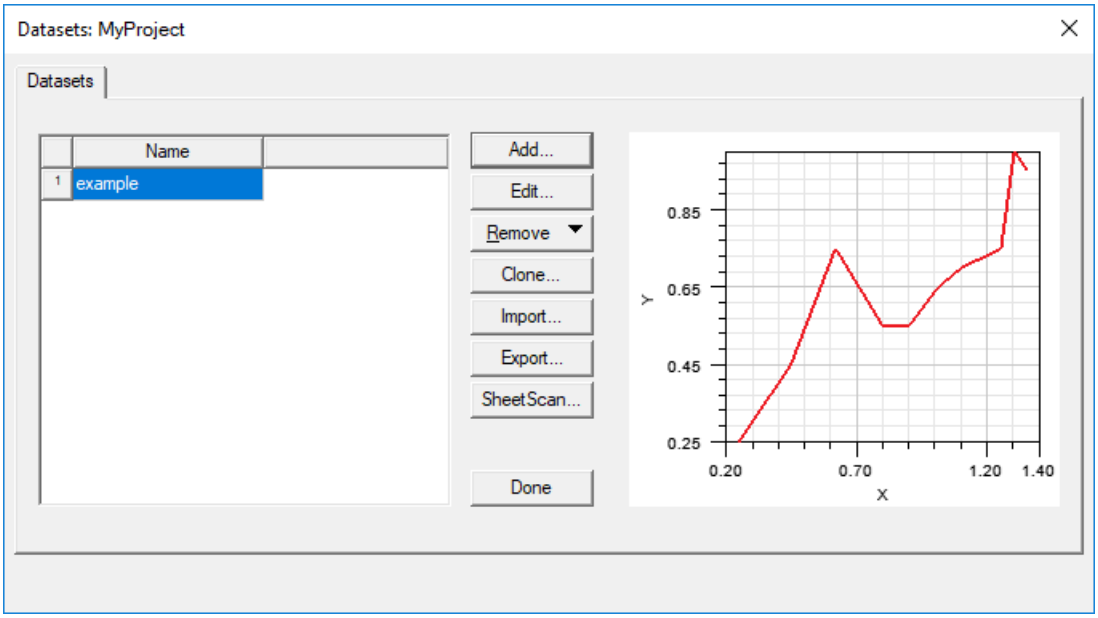
Removing Datasets

To remove a previously added dataset:

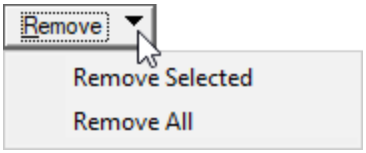
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Icepak** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.



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.

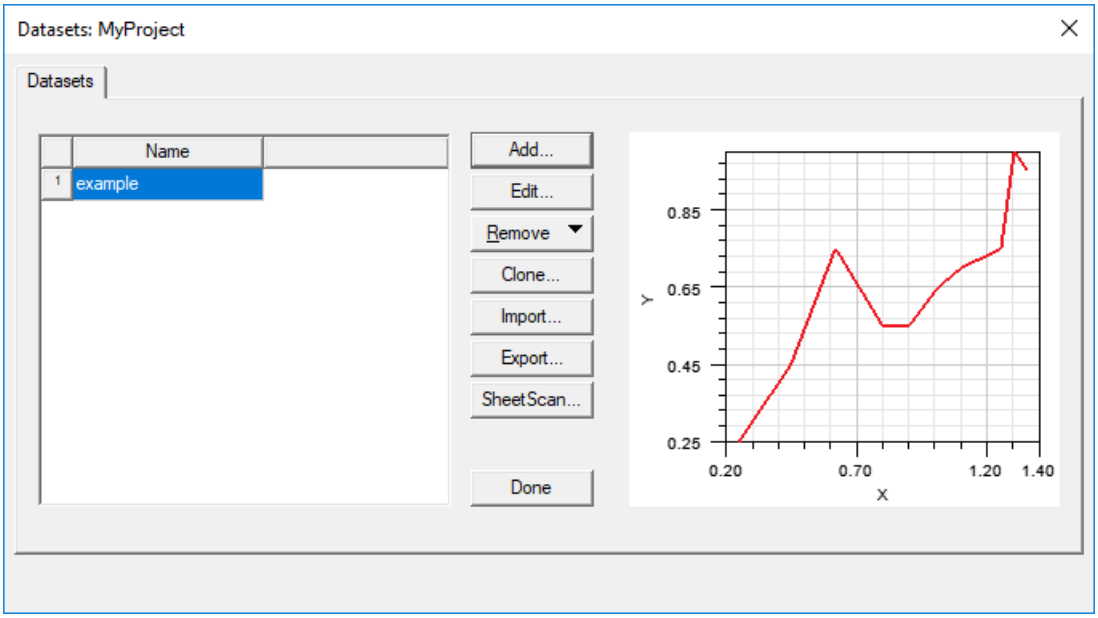
> **Note:**
>
> The **Remove All** option does not work across dataset types. That is, selecting **Remove All** from the **Project Datasets** window does not remove design-level datasets, and vice versa.

4. Click **Done** to exit the **Datasets** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Dataset Plot Properties

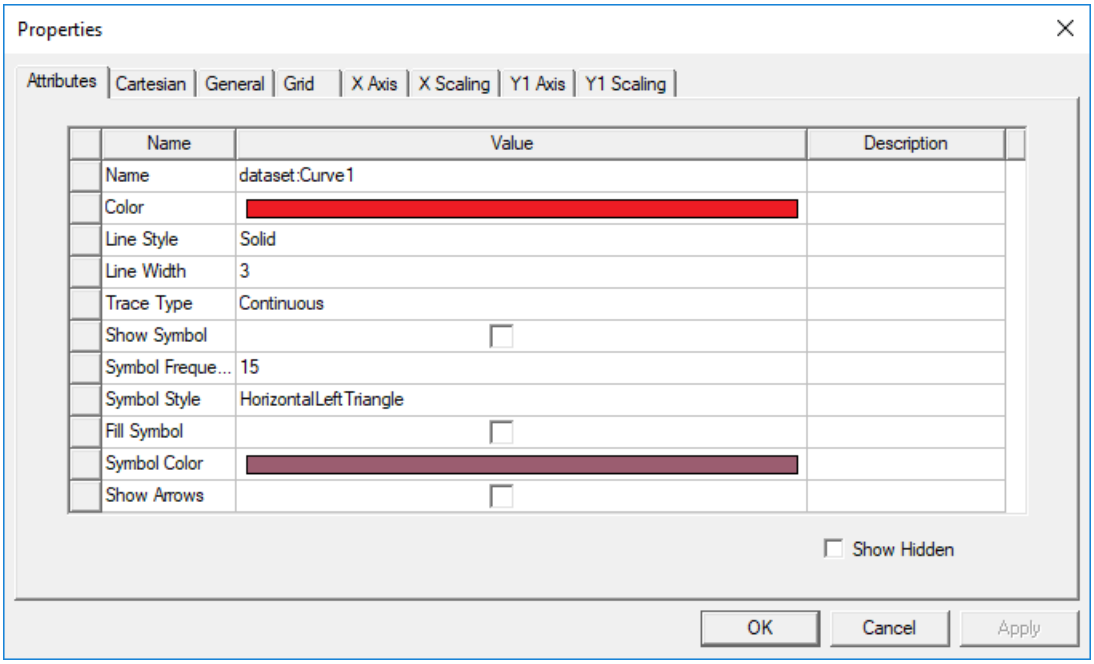
From the **Datasets** window, you can change the appearance of a dataset's plot.



To change a plot's appearance:

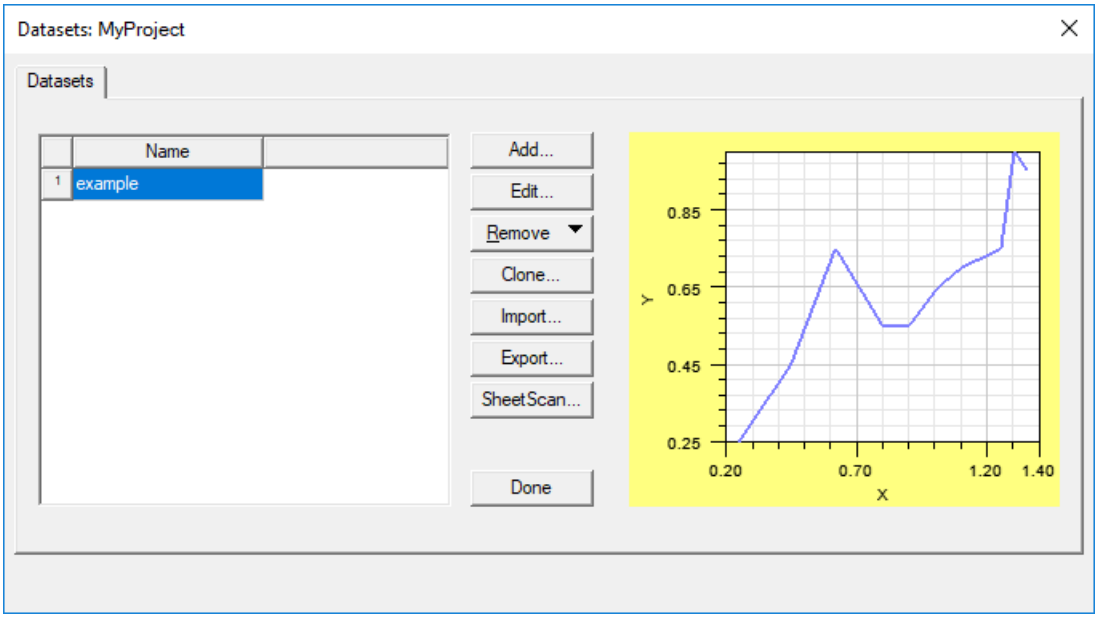
1. Double-click a plot element (for example, a line, axis, or title) to open that element's **Properties** window.

Ansys Electromagnetics Suite 2026 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 2026 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 **Icepak** > **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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

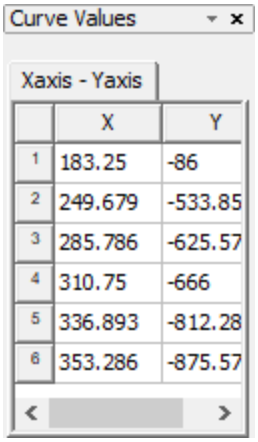


From the **View** menu, you can also toggle the following:

- **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. Data for each curve on a sheet is displayed on its own tab. You can manually change the X and Y values in the table to fine-tune the characteristic curve.

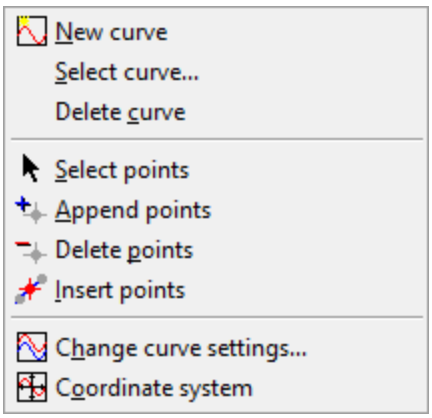


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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:

- **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

After loading a datasheet picture into the SheetScan editor, the next step is defining a coordinate system for the imported picture.

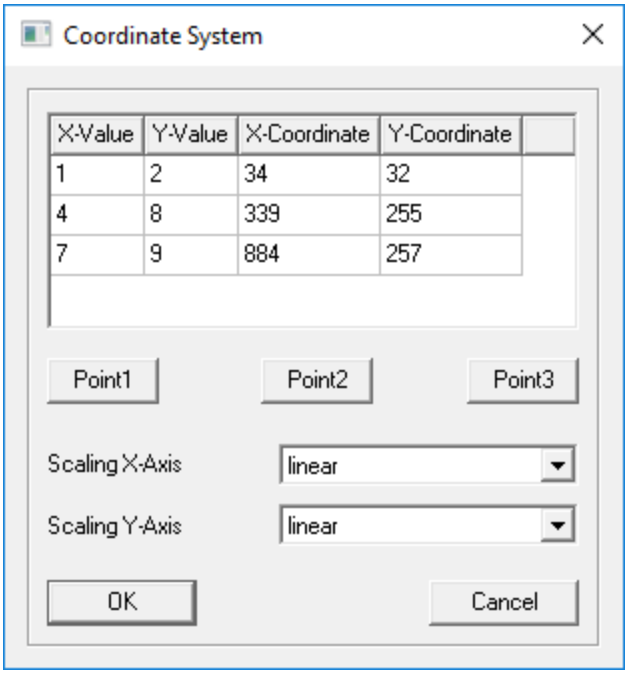
Defining a SheetScan Coordinate System

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.
7. Click **OK**.

   The grid is placed over the graphic.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

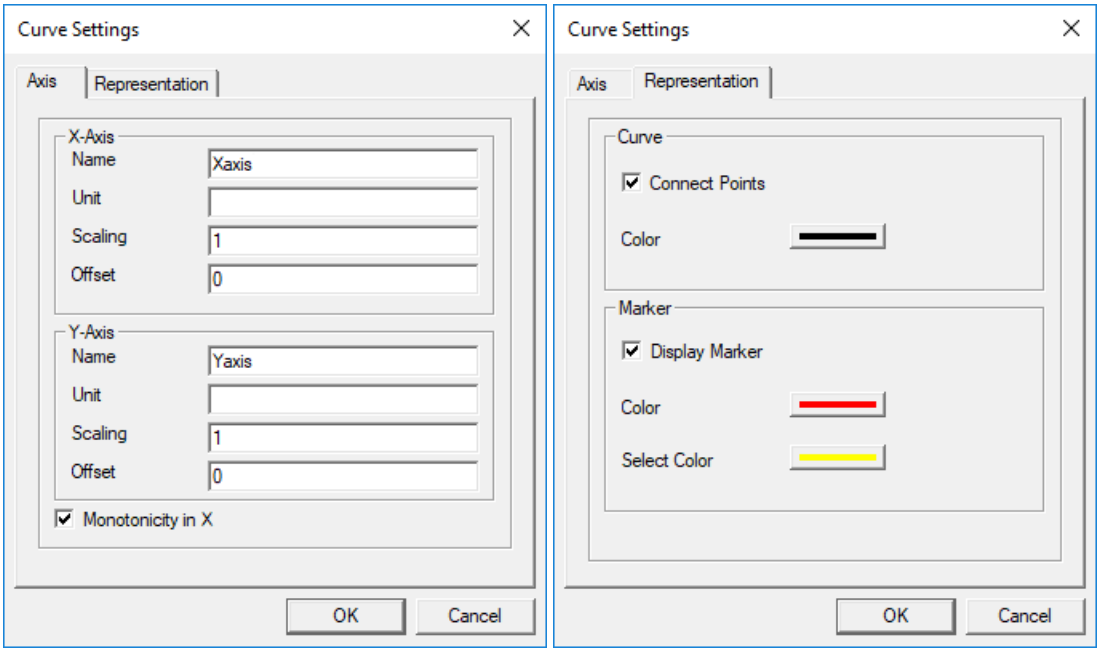
8. You can now define a Characteristic Curve.

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.



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.

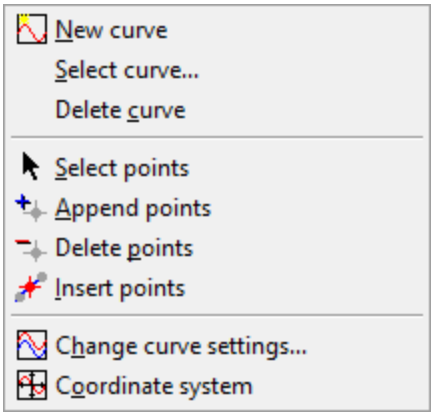
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.

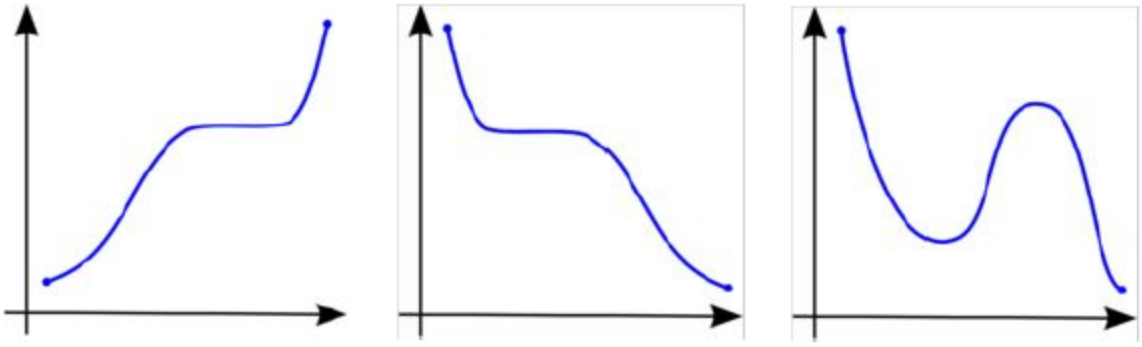
Checking for Monotonicity in X

Icepak 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:

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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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. Otherwise, selecting an **.xls** or **.xlsx** file imports the data immediately into the **Add Dataset** dialog box.

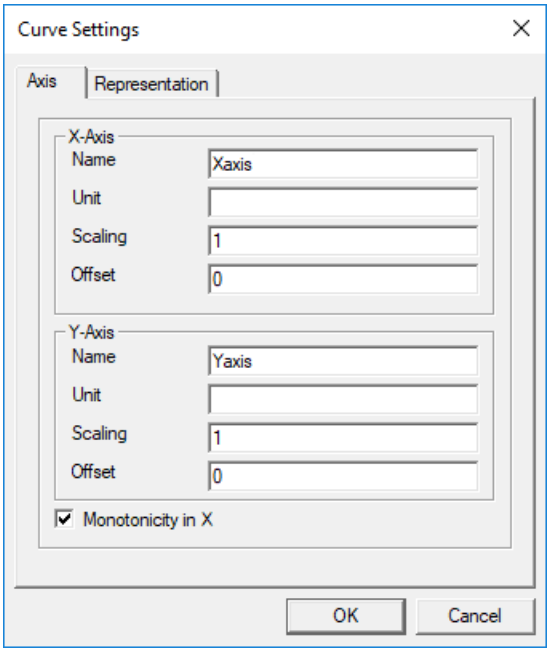
      > **Note:**
      >
      > - Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
      > - The first 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 2026 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:

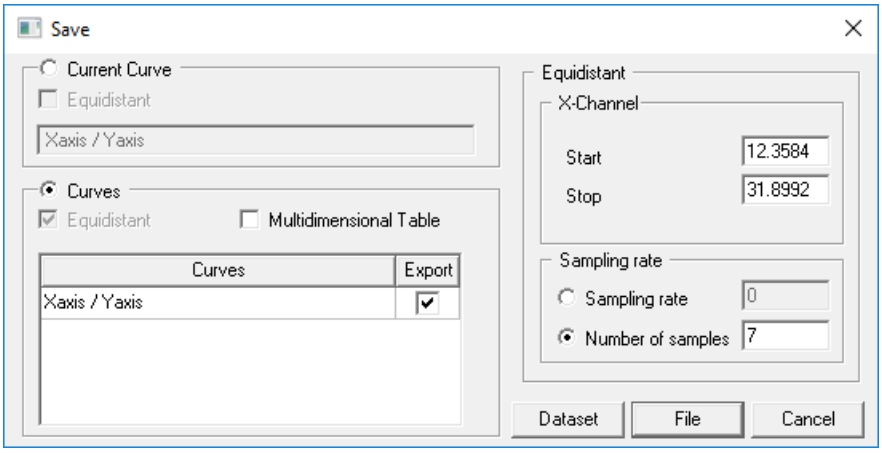
- Electronics Desktop Dataset
- Twin Builder Characteristic (*.mdx) file
- Comma Separated Value (*.csv) file
- Comtrade (*.cfg) file

To export curve data:

1. Click **File** > **Export**.

   The **Save** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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 predefined 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
||	Logical or	8

Using Intrinsic Functions in Expressions

Icepak 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, return values of inverse trigonometric functions are output in radians. When the argument to a trigonometric expression is a variable, the units of the variable 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)
exp	Exponential ($\mathbf{e}^x$)	exp(x)
pow	Raise to power (x^y)	pow(x,y)
if	If	if(cond_exp,true_exp,false_exp)
pwc	Piecewise Constant. Each specified Y value applies as a constant value for the range between the corresponding X value and the *preceding* X value. See Using a Piecewise Constant (pwc) Function for more information. (**pwc** can be used with datasets for Design Variables but not for Project variables).	pwc(dataset_exp,variable)
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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

$$clp\left(Dataset,X,Y,Z\right)=\begin{cases} DatasetValue_i \text{ if } r_i\left(X,Y,Z\right)<r_{rounding} \\ \Sigma_i \frac{DatasetValue_i}{r_i^4} \Big/ \Sigma_i \frac{1}{r_i^4} \quad otherwise \end{cases}$$

$$r_i\left(X,Y,Z\right)=\sqrt{\left(X-DatasetX_i\right)^2+\left(Y-DatasetY_i\right)^2+\left(Z-DatasetZ_i\right)^2}$$

Using Piecewise Linear Functions in Expressions

The following piecewise linear intrinsic functions are accepted in expressions:

- **`pwl (dataset_expression, variable)`** – piecewise linear along the X-axis and returns a corresponding Y value.
- **`pwlx (dataset_expression, variable)`** – piecewise linear with linear extrapolation on x 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).

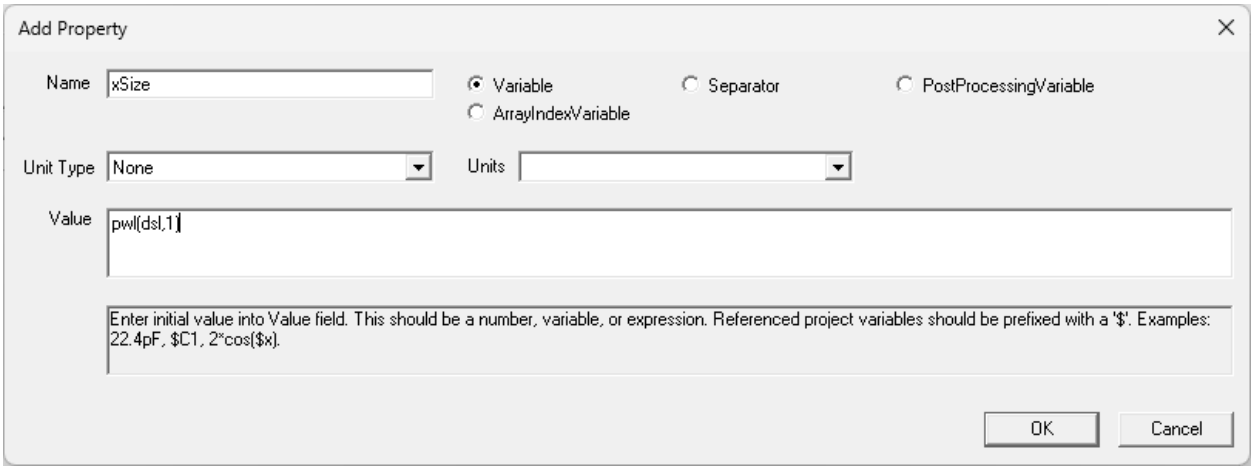
For example, you can specify BulkConductivity as:

```
pwlx($ds1, Freq)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can create a design variable representing a dimension `xSize` as `pwl(ds1, 1)` where `ds1` is a dataset.

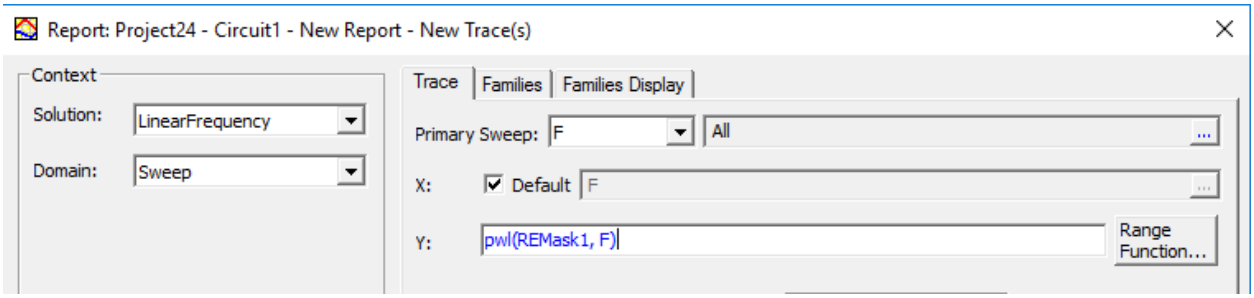
Doing so looks like this:



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:



> **Note:**
>
> pwl can be used with datasets for Design Variables but not for Project variables.

Using a Piecewise Constant (pwc) Function

For Transient solutions, you can use a **piecewise constant** (pwc) function along with a dataset expression to vary the **Time Step** parameter or the interval for saving fields data in the *Icepak Solve Setup* dialog box. You can also use a pwc function to specify time-dependent parameters

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in certain boundaries. For a complete list of transient parameters, see Transient Thermal Properties Dependency.

> **Tip:**
>
> While the pwc function 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 iterations 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 direction. Once a step change occurs, the time-dependent parameter remains stable at the post-step value.

The process of defining a dataset for use in a piecewise constant function is the same as defining a dataset for use in a piecewise linear (pwl) function. This process is covered in the topic Using Datasets to Define Transient Parameters.

Interpretation of Datasets within pwc Functions: A dataset referenced in a piecewise constant function is interpreted differently from when it is referenced in a piecewise linear function. The characteristics of pwc expressions are:

- For each datapoint interval (that is, between each pair of XY values in the referenced dataset), the slope of the curve is zero.
- All changes in the Y value are step changes.
- 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. For example: using the datapoint numbers as subscripts, Y_2 specifies the constant value for the range X_1 through X_2, Y_3 for X_2 through X_3, and so on, as tabulated below:

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**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

The *Add Dataset, Edit Dataset,* and *Design Settings* dialog boxes are not aware of whether a dataset will be referenced within a piecewise linear (pwl) or piecewise constant (pwc) function. These dialog boxes will always interpret and display the data points as a piecewise **linear** curve. In fact, the same dataset could be referenced in either function type 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:

Figure 1: Dataset (Tstep) Definition

Figure 2: Dataset Interpretation as a PWC *Time Step* control – *pwc(Tstep, Time)*

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.

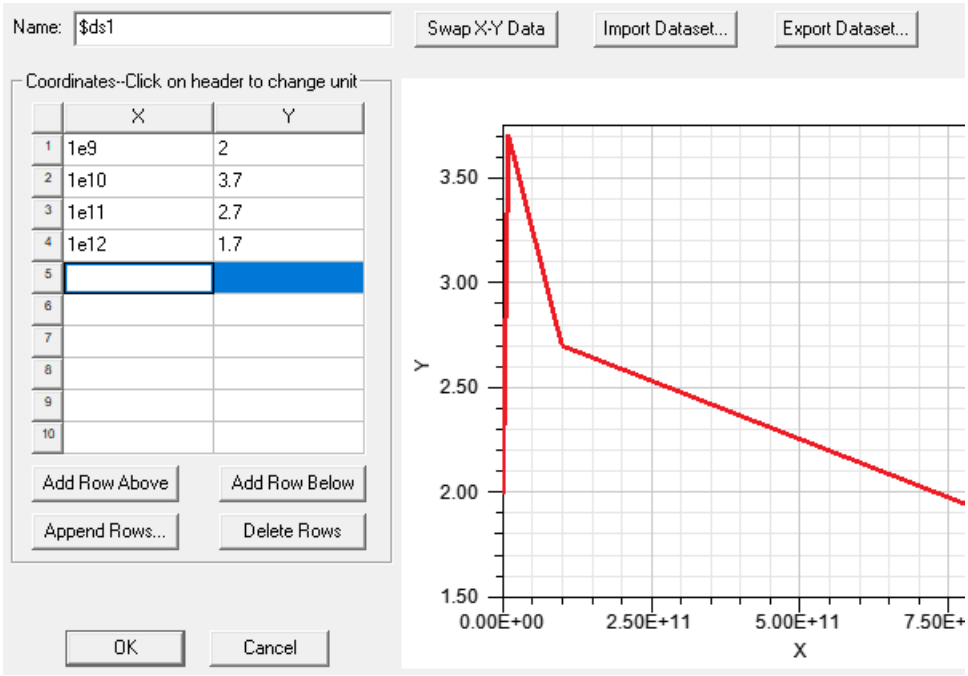
Note:

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 **Icepak** > **Design Datasets**.
2. Click **Add**.
3. Set the **Name** field and **Coordinates** as desired and click **OK**.

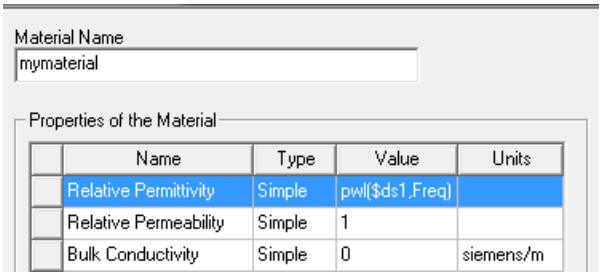
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dataset is created.



> **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.



7. Click **OK**.

Ansys Electromagnetics Suite 2026 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

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 (millikelvin) (Legacy name for delta_mkel, mkeldiff continues to be supported)
- ckel, delta_ckel (centikelvin)
- dkel, delta_dkel (decikelvin)
- kel, delta_kel (Legacy name for delta_kel, keldiff continues to be supported)

Fahrenheit Family

- fah, delta_fah

Temperature-related Use Cases

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 behavior, 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

Steady State Boundary Condition Thermal Properties Dependency

In the Electronics Desktop, you can create datasets, which are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. In Icepak, you can create a dataset to define temperature dependent thermal properties for specific boundary conditions.

The following table displays which thermal boundary condition properties can be defined as a dependent expression.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Condition	Thermal Property	Steady-State Dependency
Fluid Block	Total Power	Temperature, Spatial
	Heat Transfer Coefficient	Temperature
	Temperature	
Hollow Block	Total Power	Temperature, Spatial
	Heat Flux	Temperature
	Temperature - Fixed Temperature	
	Heat Transfer Coefficient	Temperature
	Temperature - Internal Conditions	
Solid Block	Total Power	Temperature, Spatial
	Heat Transfer Coefficient	Temperature
	Temperature	
Openings	Temperature	Spatial
	Static Pressure/Total Pressure	Spatial
	X Velocity	Spatial
	Y Velocity	Spatial
	Z Velocity	Spatial
Plate (Conducting Thin)	Power	Temperature
Resistance	Power	Spatial
3D Source	Power	Temperature
	Surface/Volume Flux	Temperature, Spatial
2D Source	Power	Temperature
	Surface/Volume Flux	Temperature, Spatial
Wall (Stationary)	Heat Flux	Spatial
	Temperature	Spatial
	Heat Transfer Coefficient	Temperature, Spatial

To define a thermal property as temperature dependent, enter the expression in the property's field in thermal model dialog box. The following is an example of a piecewise linear expression using the intrinsic variable for temperature (Temp):

```
pwl(datasetName, Temp)
```

where `datasetName` is the name given to the dataset.

Ansys Electromagnetics Suite 2026 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.

Modeling Quick Links

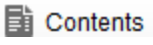
Use the following links for quick information on the following topics.

Setting Units of Measurement
Assigning Transparency to an Object
Subtracting Objects
Measuring Objects
Drawing Bondwires
Keyboard Shortcuts
Modifying the Model View
Working with Variables

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Modeler Drawing Options
Selecting Items in the Modeler Window
Creating 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 Engineering Copilot** – Help menu entry that opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant. For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

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 Icepak design. Toolkits can aid in:

- Geometry creation
- Modeling
- Productivity
- Post processing & reporting

> **Note:**
>
> See Getting Started with Icepak - Toolkits for a workflow related to this feature.

Use IronPython for scripting the user interface and commands. Custom toolkits can be accessed from the **Icepak** > **Toolkit** menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

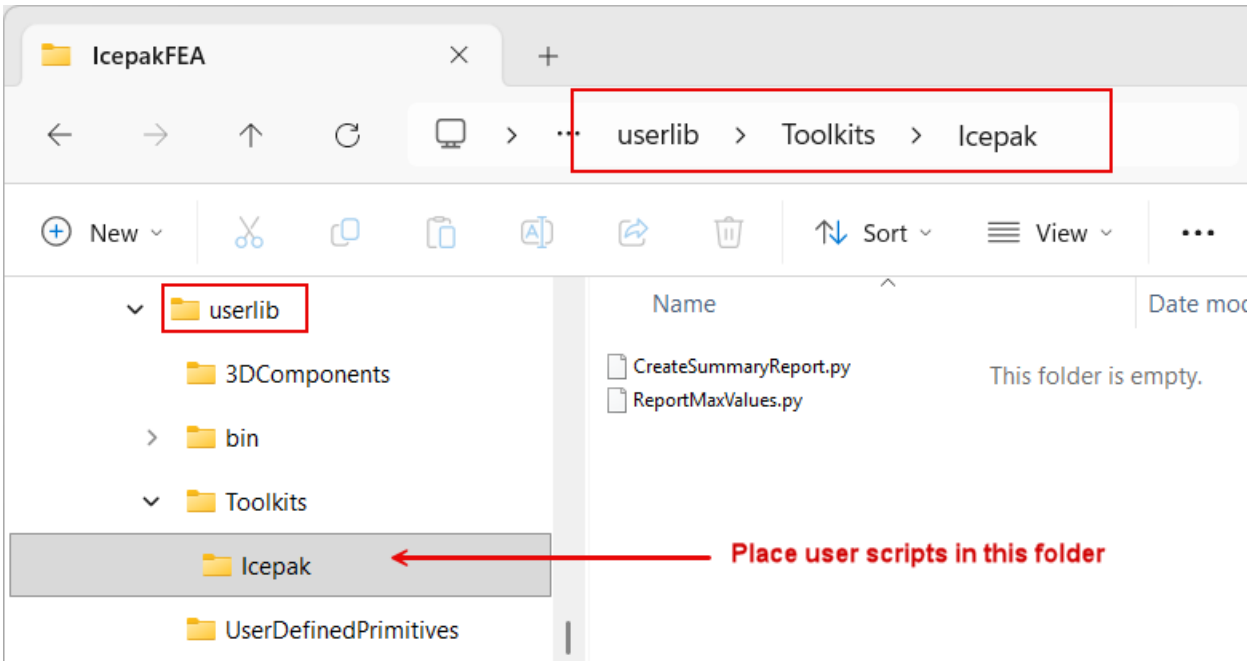
See Icepak Toolkits to learn more about the toolkits that are included for Icepak designs. Also, you can create toolkit folders, create your own custom scripts, and include them in the **Toolkit** menu. The folder names must be defined exactly as described and pictured in the following instructions:

1. Create a **Toolkits** folder in the **"<*installation_dir*>\v261\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 an **Icepak** subfolder in the *Toolkits* folder.
3. Save one or more script files to the **Icepak** subfolder at the bottom of the path just created.

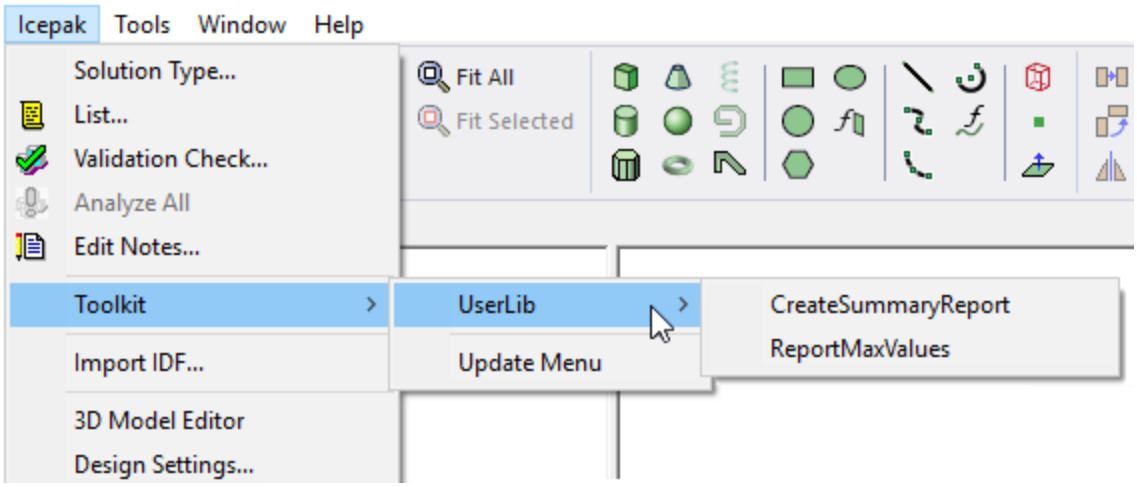


   > **Note:**
   >
   > Do not attempt to record scripts directly to a folder in the *Program Files* branch of your system drive. Record the script to a working folder, edit it as needed, and then copy the *.py file into the **...\userlib\Toolkits\Icepak** folder.

4. Click **Icepak** > **Toolkit** > **Update Menu** to refresh the menu contents.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Go to the **Icepak** > **Toolkit** > **UserLib** menu to verify that your scripts are available for selection:



> **Note:**
>
> When you open or insert an IcepakFEA design, the *Icepak* > *Toolkit* > *UserLib* menu is updated automatically. However, if you add, remove, or rename any scripts, you can use the **Icepak** > **Toolkit** > **Update** command to immediately refresh the menu contents.

Icepak Toolkits

Icepak toolkits provide additional capabilities that help automate routine tasks.

- Geometry Toolkits
- Modeling Toolkits
- Productivity Toolkits
- Reporting Toolkits

Currently, you are unable to record Icepak toolkit scripts; however, the Electronics Desktop supports Icepak toolkit script playback. Icepak toolkits can be run three ways using scripting. Using the Tile geometry toolkit as example, the following are the three toolkit scripting methods and corresponding application behaviors:

```
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", [])
```

Including empty brackets ([]) in the command launches the toolkit user interface for you to enter parameter values.

```
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", ["ng"])
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Including the letters ng in quotations within brackets (["ng"]) in the command does not launch the toolkit user interface and creates the toolkit non-graphically using the default values.

```
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", scriptInput)
```

Including a list variable in the command does not launch the toolkit user interface and creates the toolkit non-graphically using the parameter values included in the list. See below for an example of a list defined as a variable.

```
scriptInput = ["ng"]
scriptInput.append("Name:Tile1")
scriptInput.append("length_unit:meter")
scriptInput.append("Number:1")
scriptInput.append("Offset:0.0")
scriptInput.append("Plane:XY")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:0.6")
scriptInput.append("yL:0.6")
scriptInput.append("zL:0.6")
scriptInput.append("FreeAreaRatioType:Uniform")
scriptInput.append("FreeAreaRatio:0.5")
scriptInput.append("DuplicateTileAxis:+X")
```

Icepak Geometry Toolkits

Icepak has the following toolkits that assist in creating model geometry.

- Approximation - Approximate Circular Object as a Polygon
- Approximation - Approximate Cylinder with Plates
- Approximation - Approximate Cylinder with Polygonal Block
- Approximation - Create Polygonal Enclosure
- Approximation - Approximate 1/4 Cylinder with Polygonal Block
- Approximation - Create Adiabatic Enclosure with Thin Sides
- Data Center Components - CRAC
- Data Center Components - PDU
- Data Center Components - Rack (Recirc)
- Data Center Components - Tile
- Heatsinks - AngledFinHS
- Heatsinks - Heatsink Creation
- Other - Create Network Representation Heat Exchanger

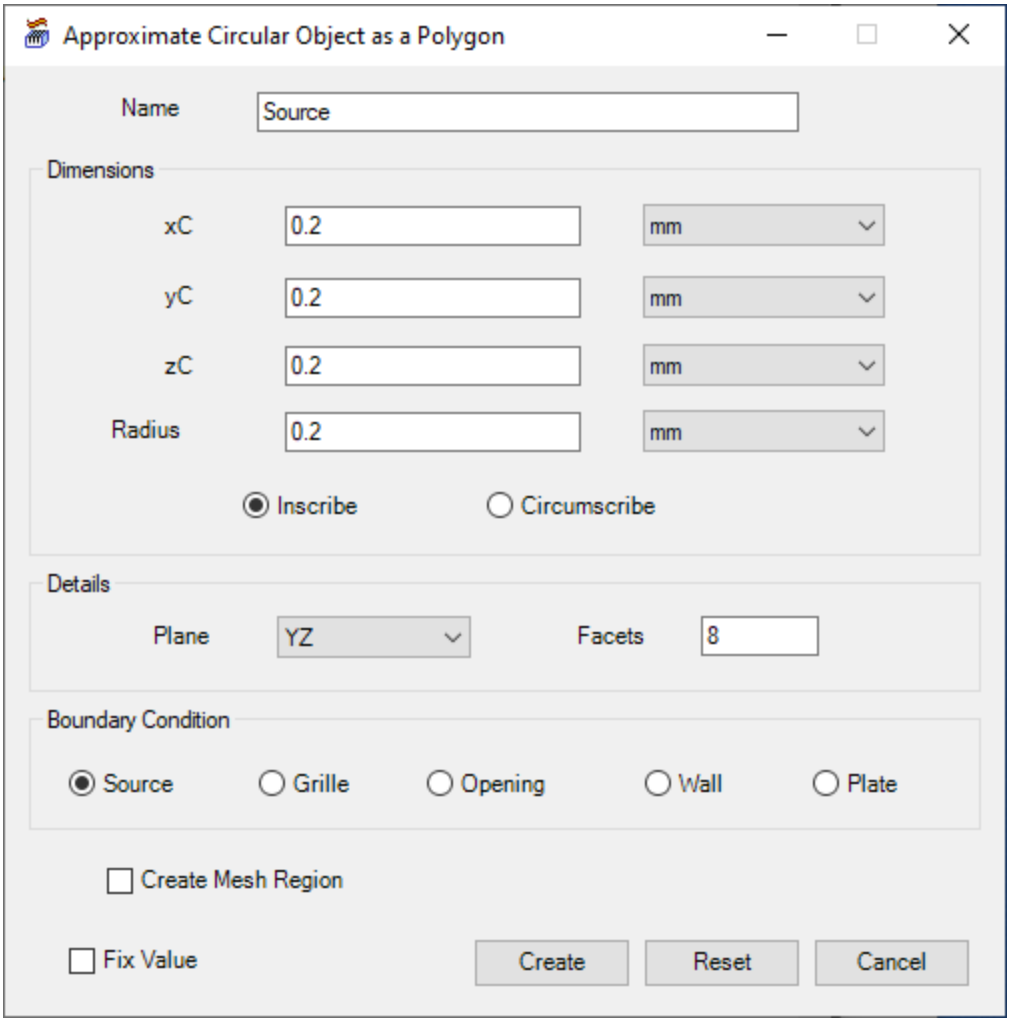
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Packages - Cavity-Down_BGA
- Packages - DUAL
- Packages - Flipchip_BGA
- Packages - FPBGA
- Packages - LCC
- Packages - PBGA
- Packages - QFN
- Packages - QFP
- Packages - SDBGA
- Packages - SOT
- Packages - Network Models - BGA_Network
- Packages - Network Models - Delphi_3Node_Network
- Packages - Network Models - Delphi_BGA_Network
- Packages - Network Models - Delphi_QFP_Network
- Packages - Network Models - GeneralNetwork
- Packages - Network Models - General_3Node_Network
- Printed Circuit Board (PCB)
- Source From CSV
- Write Average Metal Fraction

Approximate Circular Object as a Polygon

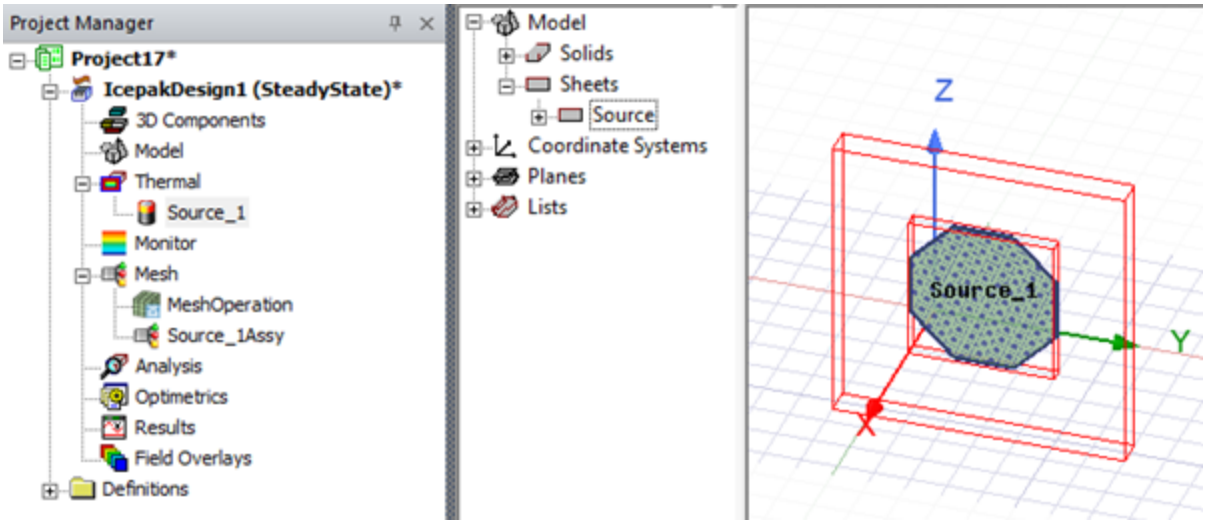
Approximate Circular Object as a Polygon enables you to create geometry, assign a boundary condition, and create a mesh region for a circular polygonal object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Dimensions	Under **Dimensions**, specify the **xC**, **yC**, **zC** lengths to define the object's location (geometric center). Specify the **Radius** to define the object's size. Select **Inscribe** to create the polygon outside the specified circle or **Circumscribe** to create the polygon inside the specified circle.
Details	Under **Details**, specify which **Plane** the object will exist on and how many **Facets** it will have.
Boundary Condition	**Boundary Condition** specifies which type of boundary condition is applied.
Create Mesh Region	Select **Create Mesh Region** to create a mesh region around the geometry.

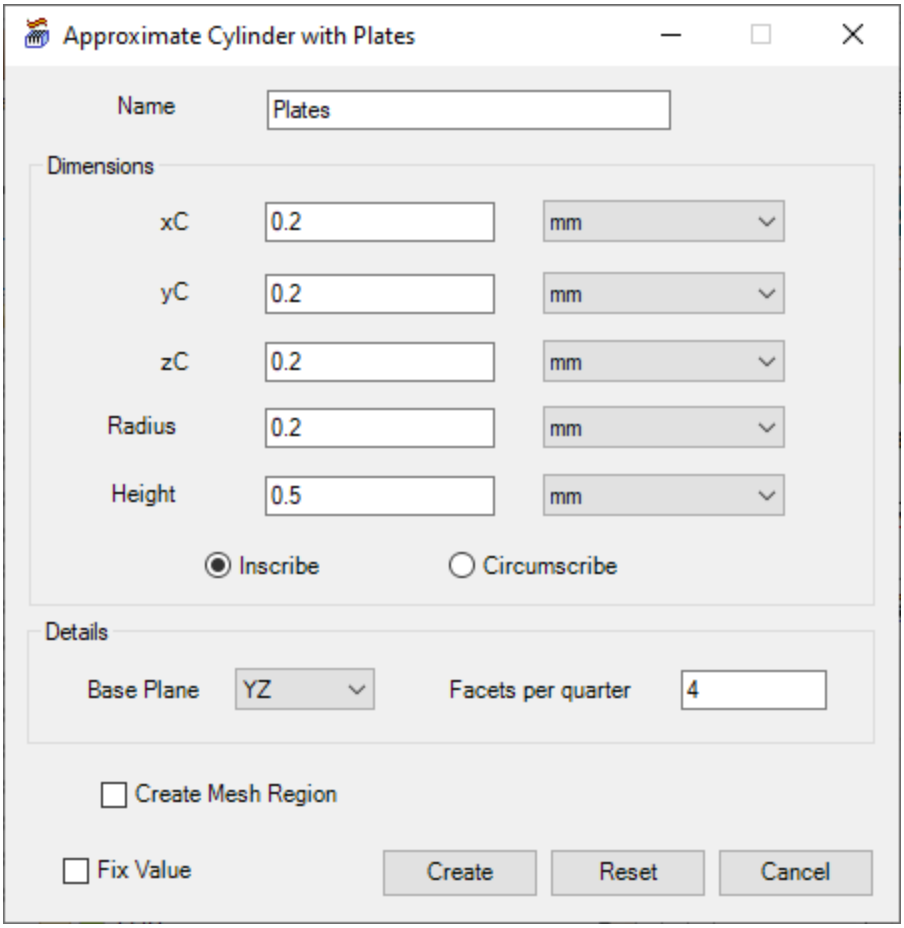
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Approximate Cylinder with Plates

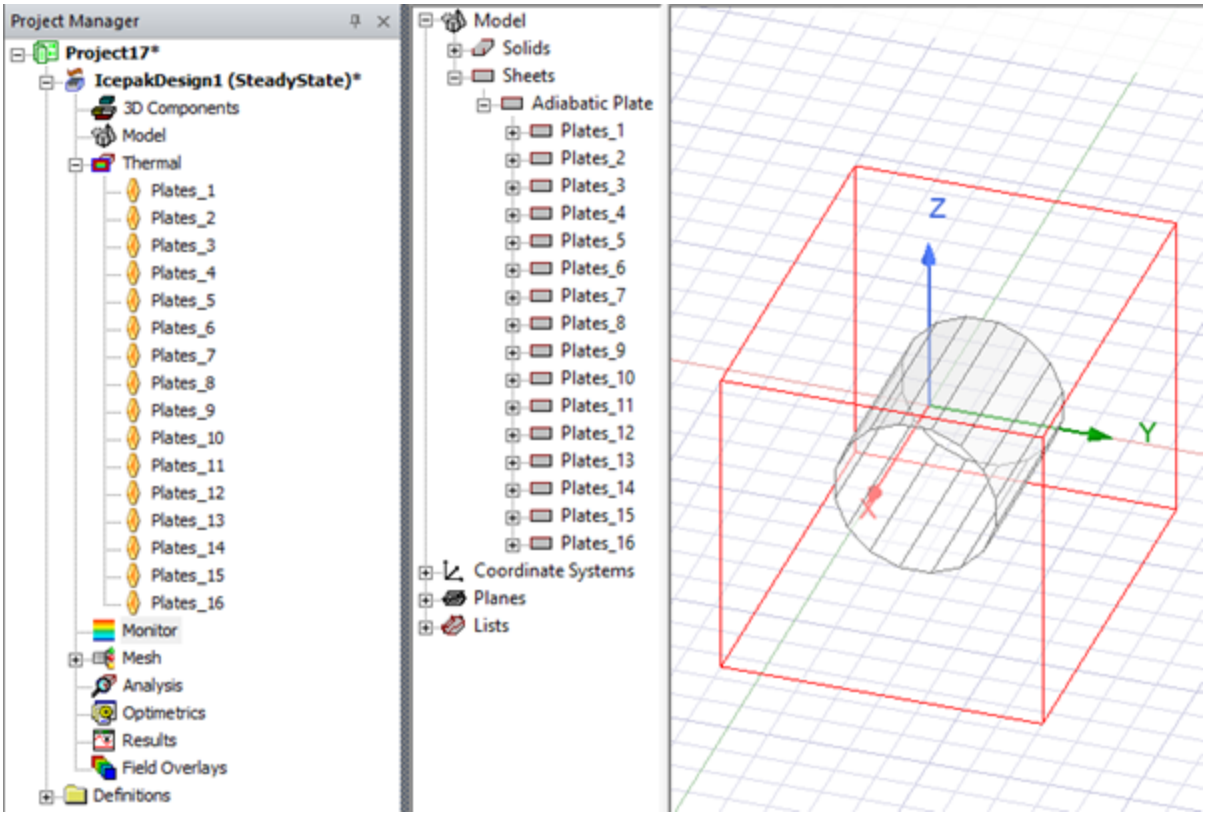
Approximate Cylinder with Plates enables you to create geometry, assign plate boundary conditions, and create a mesh region for a cylinder object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Dimensions	Under **Dimensions**, specify the **xC**, **yC**, **zC** lengths to define the object's location (geometric center). Specify the **Radius** and **Height** to define the object's size. Select **Inscribe** to create the cylinder outside the specified circle or **Circumscribe** to create the cylinder inside the specified circle.
Details	Under **Details**, specify which **Base Plane** the object will exist on and how many **Facets** it will have.
Create Mesh Region	Select **Create Mesh Region** to create a mesh region around the geometry.

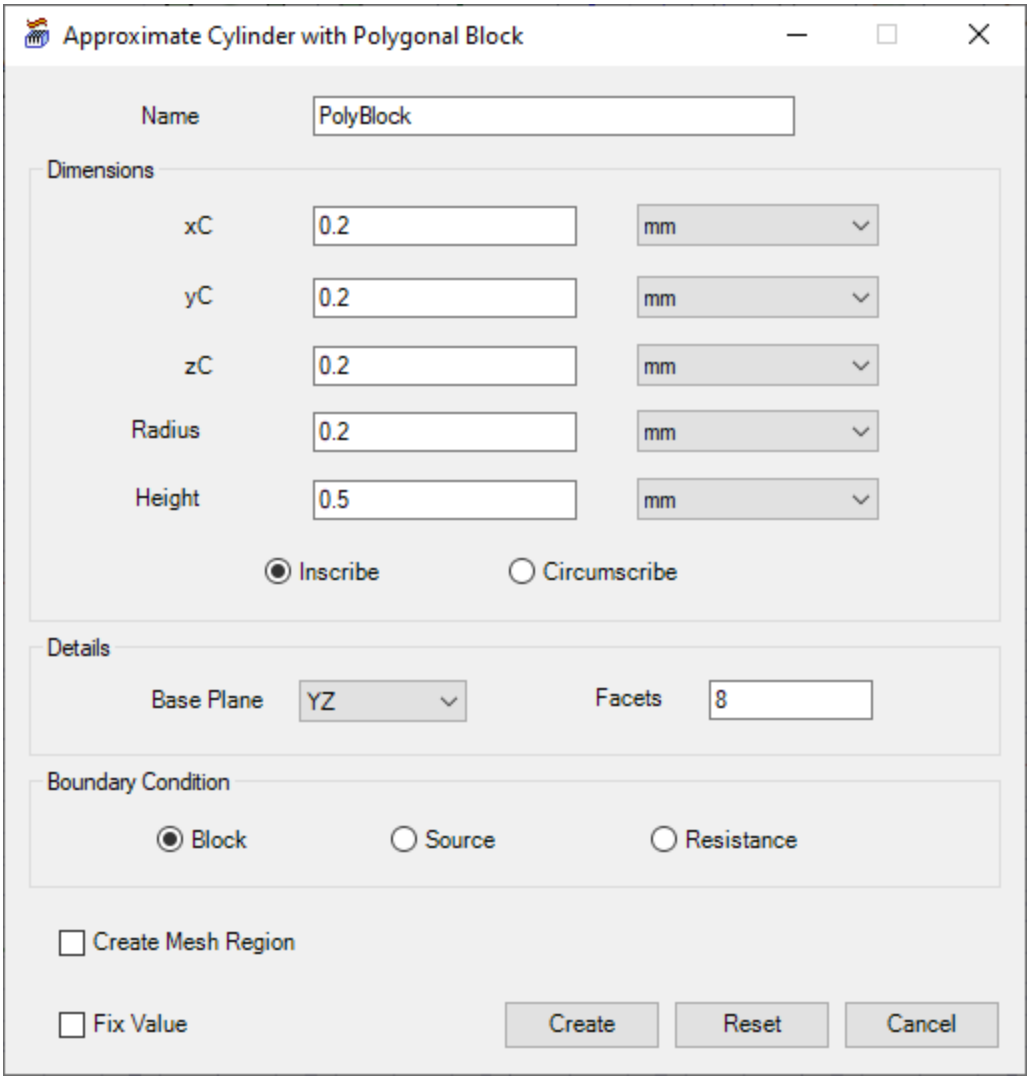
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Approximate Cylinder with Polygonal Block

Approximate Cylinder with Polygonal Block enables you to create geometry, assign a boundary condition, and create a mesh region for a cylinder object.

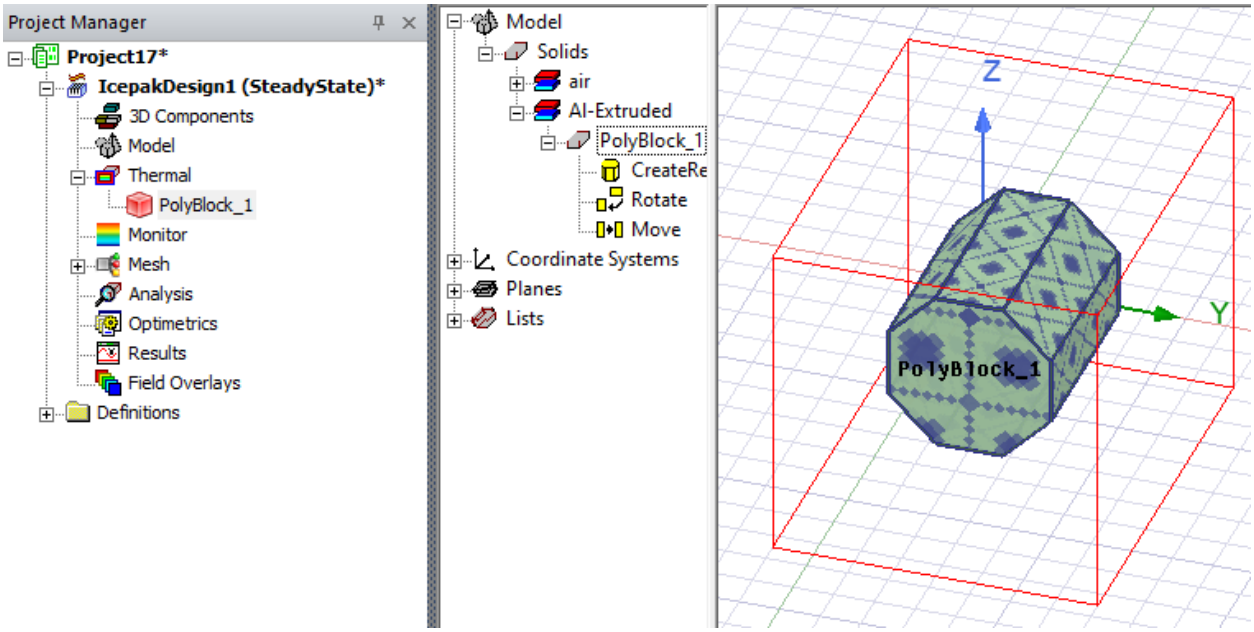
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Dimensions	Under **Dimensions**, specify the **xC**, **yC**, **zC** lengths to define the object's location (geometric center). Specify the **Radius** and **Height** to define the object's size. Select **Inscribe** to create the cylinder outside the specified circle or **Circumscribe** to create the cylinder inside the specified circle.
Details	Under **Details**, specify which **Base Plane** the object will exist on and how many **Facets** it will have.
Boundary Condition	**Boundary Condition** specifies which type of boundary condition is applied.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

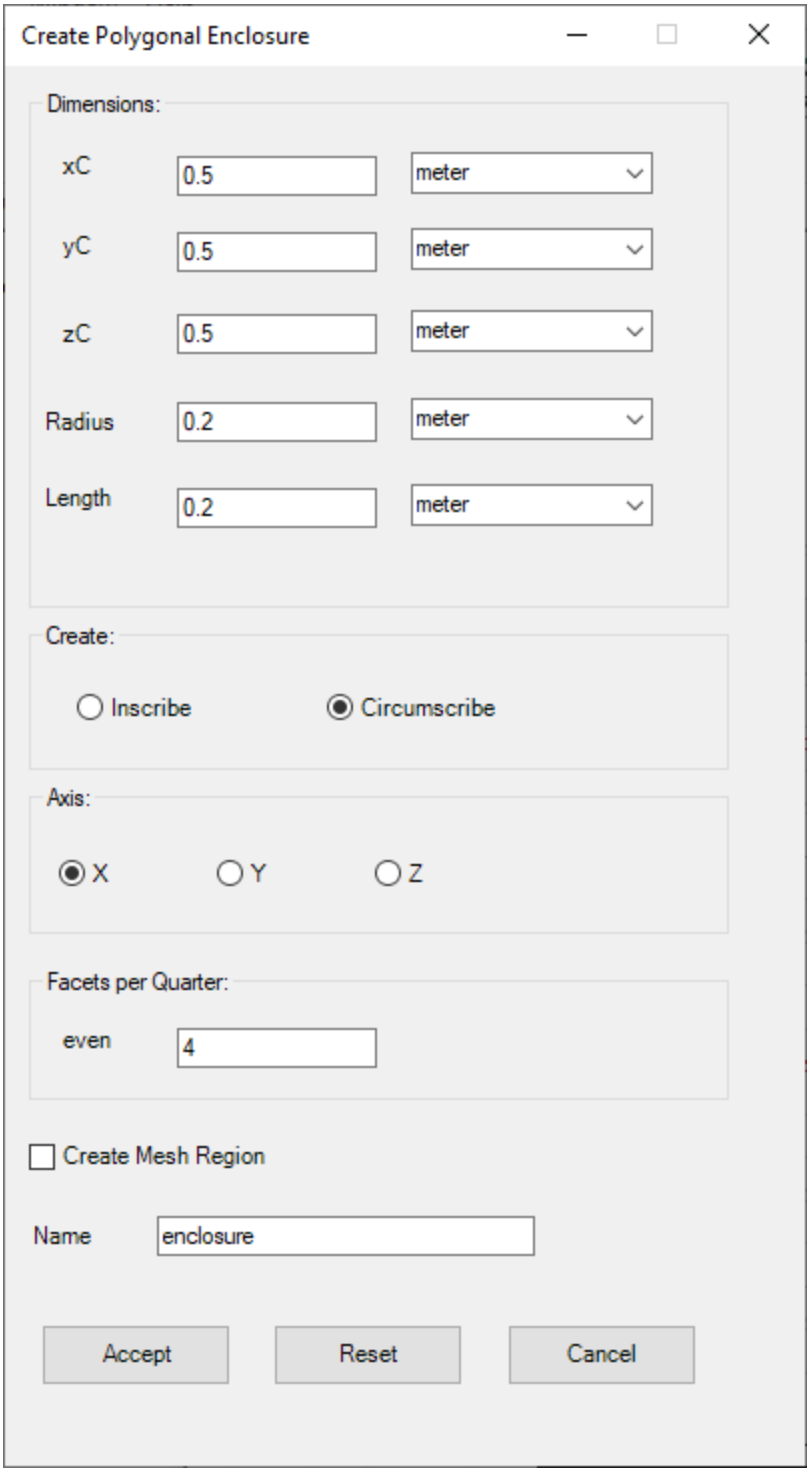
Create Mesh Region	Select **Create Mesh Region** to create a mesh region around the geometry.



Create Polygonal Enclosure

Create Polygonal Enclosure enables you to create geometry, assign a boundary condition, and create a mesh region for a polygonal enclosure.

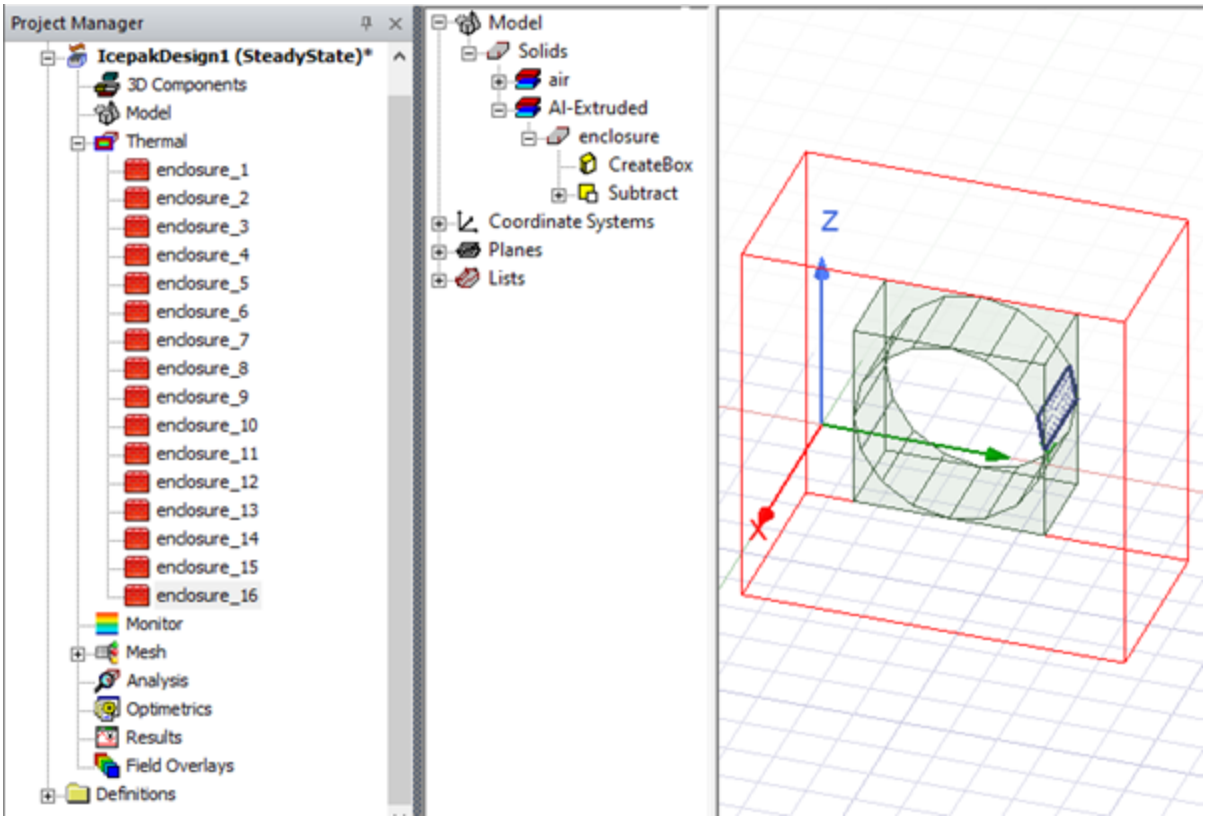
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Dimensions	Under **Dimensions**, specify the **xC**, **yC**, **zC** lengths to define the object's location (geometric center). Specify the **Radius** and **Length**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

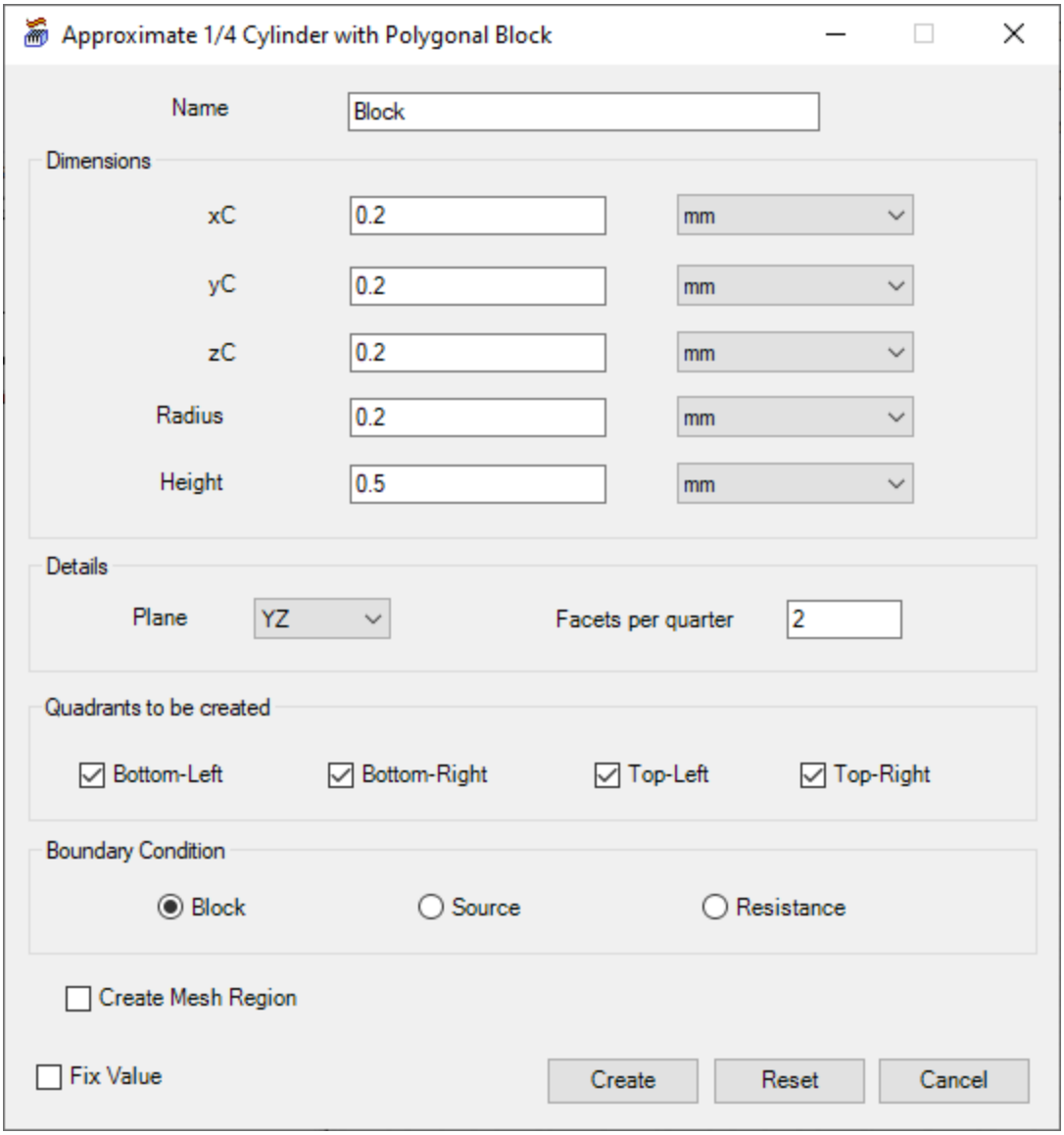
	to define the object's size.
Create	Under **Create**, specify **Inscribe** to create the enclosure outside the specified polygon or **Circumscribe** to create the enclosure inside the specified polygon.
Axis	Under **Axis**, select the axis on which to orient the enclosure.
Facets per Quarter	Under **Facets per Quarter**, specify then number of facets in each quarter of the enclosure.
Create Mesh Region	Select **Create Mesh Region** to create a mesh region around the geometry.
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.



Approximate 1/4 Cylinder with Polygonal Block

Approximate 1/4 Cylinder with Polygonal Block enables you to create geometry, assign a boundary condition, and create a mesh region for a one quarter or more of a cylinder object.

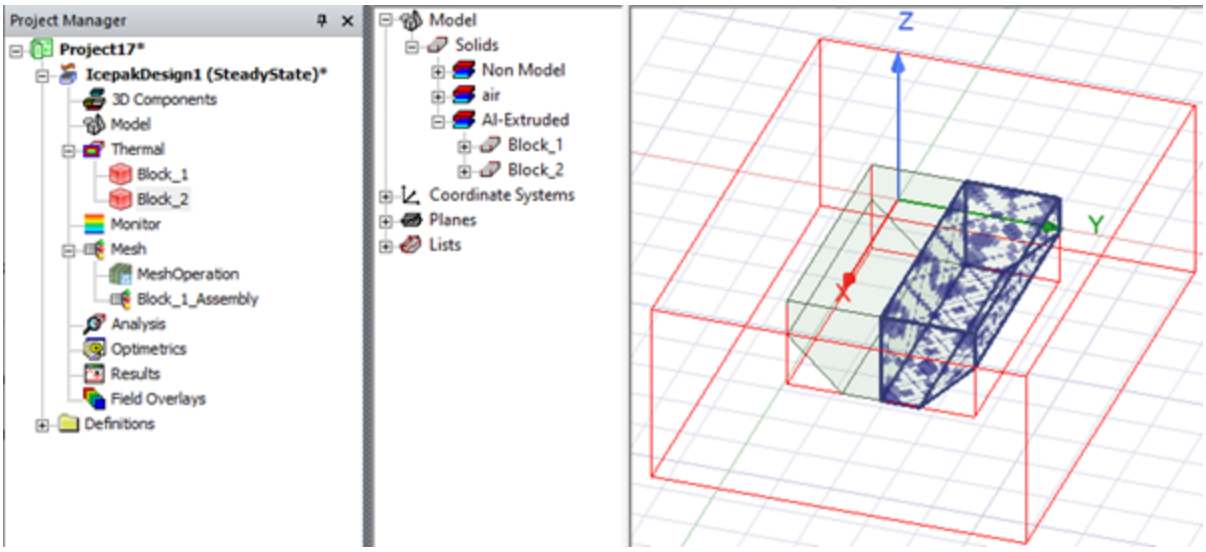
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Dimensions	Under **Dimensions**, specify the **xC**, **yC**, **zC** lengths to define the object's location (geometric center). Specify the **Radius** and **Height** to define the object's size.
Details	Under **Details**, specify which **Base Plane** the object will exist on and how many **Facets** it will have.
Quadrants to be created	Under **Quadrants to be created**, select which quadrants of the cylinder you want to create.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

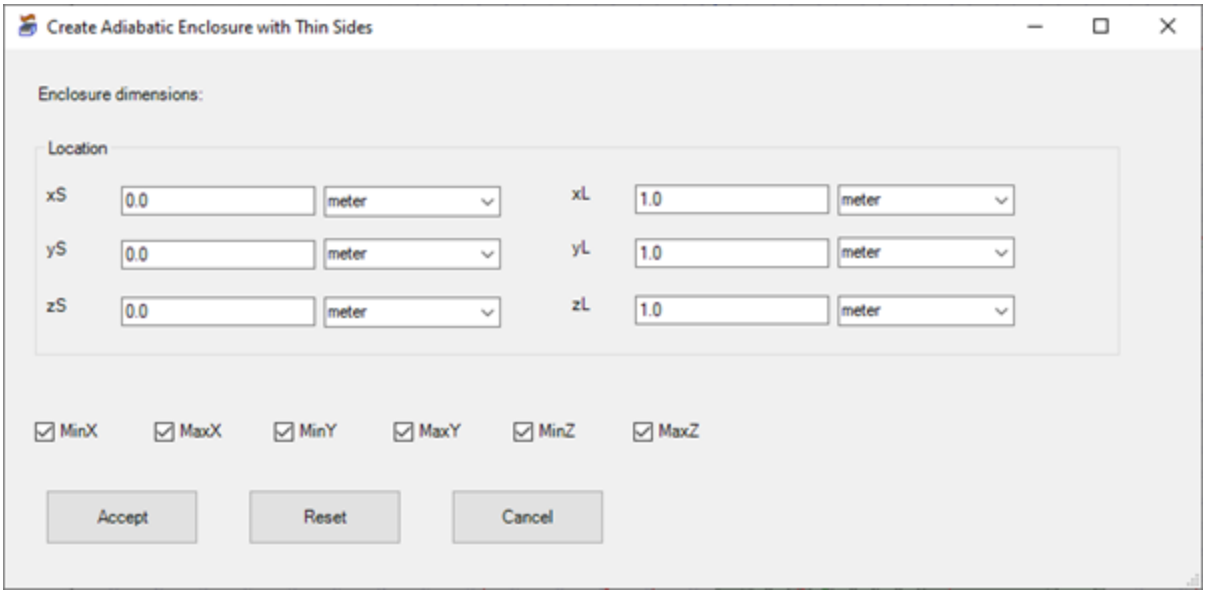
Boundary Condition	**Boundary Condition** specifies which type of boundary condition is applied.
Create Mesh Region	Select **Create Mesh Region** to create a mesh region around the geometry.



Create Adiabatic Enclosure with Thin Sides

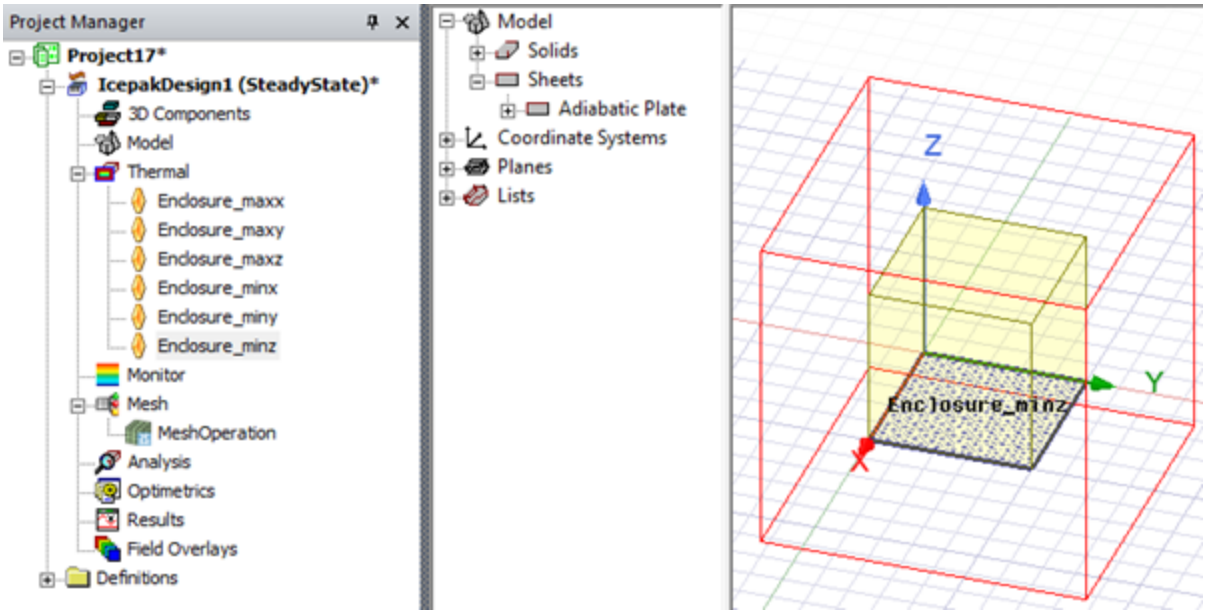
Create Adiabatic Enclosure with Thin Sides enables you to create geometry and assign adiabatic plate boundary conditions for an enclosure with thin sides.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Location	Under **Location**, enter **xS**, **yS**, and **zS** values to define the enclosure's start location in each plane, and enter **xL**, **yL**, and **zL** to define the length of the sides in each coordinate direction.
Min and **Max** Sides	Enable the **Min** and **Max** side check boxes to create the sheet geometry and boundary condition for the corresponding side. Disable the check box to exclude the side.
Accept	Click **Accept** to create the enclosure.
Reset	Click **Reset** to restore the default values.
Cancel	Click **Cancel** to close the dialog box without creating the enclosure.

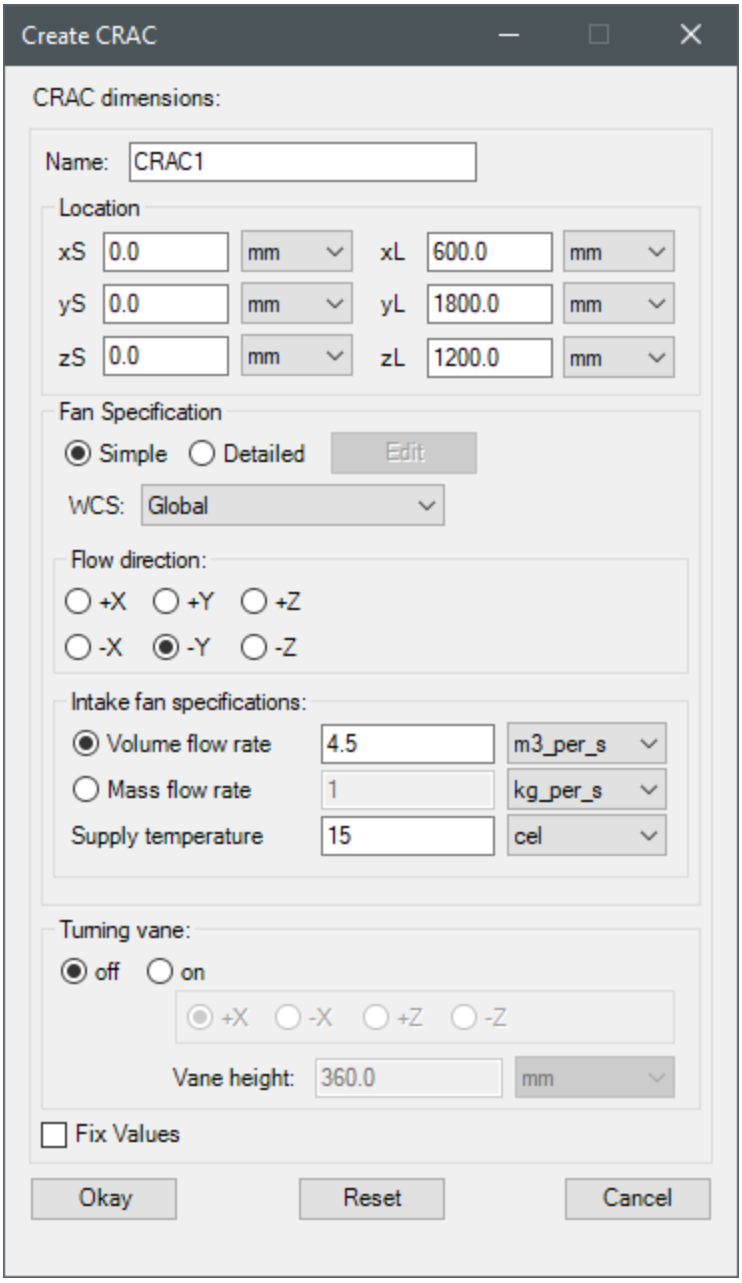
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Data Center Components - CRAC

CRAC enables you to create geometry and boundary conditions to represent a computer room air conditioning unit.

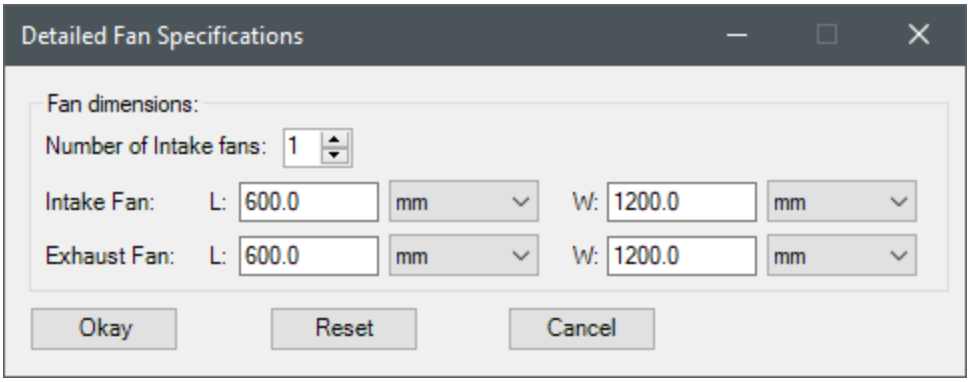
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



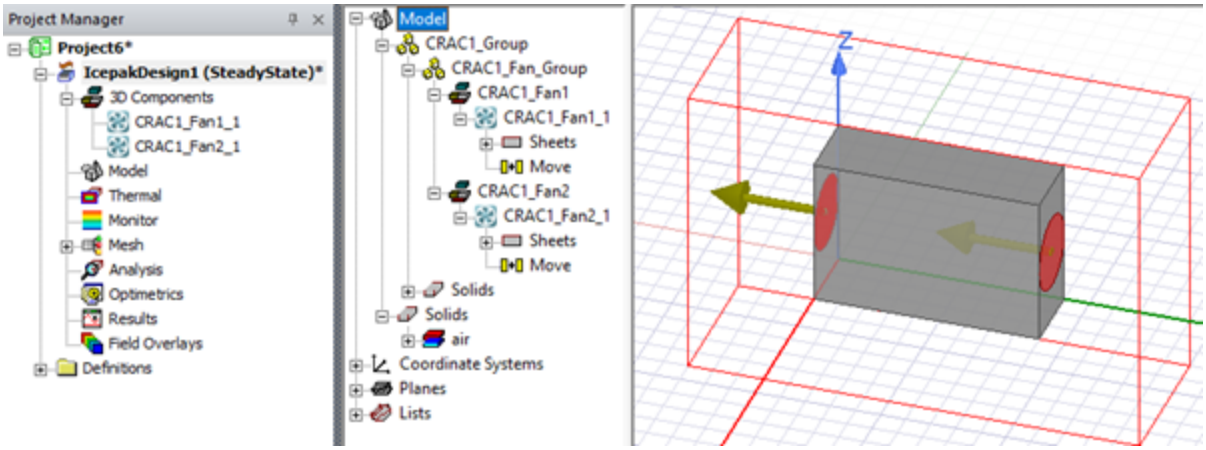
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Location	Under **Location**, specify the x, y, and z start and length location to define the object's size.
Fan Specification	Under **Fan Specification**, specify whether the fan are defined as Simple or Detailed. See Detailed Fan Specifications below for more

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	information. Also, select the coordinate system on which to create the fan.
Flow direction	**Flow direction** specifies the direction in which the air flows from the fan.
Intake fan specifications	Under **Intake fan specifications**, specify **Volume flow rate** or **Mass flow rate** and the **Supply temperature**.
Turning vane	Under **Turning vane**, specify off or on. If enabled, specify which coordinate direction the flow is directed and the **Vane height**.



Number of Intake fans	**Number of Intake fans** specifies the amount of intake fans included in the model.
Intake Fan	**Intake Fan** specifies the length and width of the intake fan.
Exhaust Fan	**Exhaust Fan** specifies the length and width of the exhaust fan.



Iron Python Script – *CRAC.py*:

The following is the toolkit's scripting information.

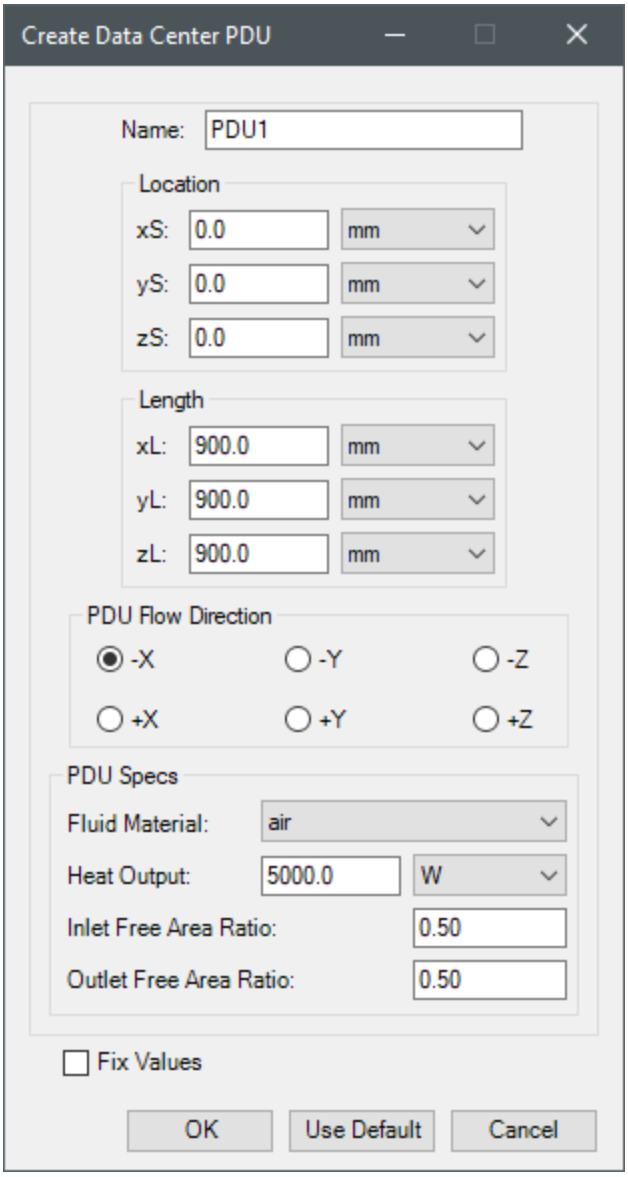
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
scriptInput = ["ng"]
scriptInput.append("Name:CRAC1")
scriptInput.append("length_unit:meter")
scriptInput.append("VFRUnit:m3_per_s")
scriptInput.append("MFRUnit:kg_per_s")
scriptInput.append("tempUnit:cel")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:0.6")
scriptInput.append("yL:1.8")
scriptInput.append("zL:1.2")
scriptInput.append("WCS:Global")
scriptInput.append("FanSpec:Detailed")
scriptInput.append("IntakeLength:1.2")
scriptInput.append("IntakeWidth:0.6")
scriptInput.append("ExhaustLength:1.2")
scriptInput.append("ExhaustWidth:0.6")
scriptInput.append("NumIntake:2")
scriptInput.append("FlowDirection:-Y")
scriptInput.append("IntakeFanSpec:VolumeFlowRate")
scriptInput.append("VolumeFlowRate:4.5")
scriptInput.append("MassFlowRate:1")
scriptInput.append("SupplyTemp:15")
scriptInput.append("TurningVane:On")
scriptInput.append("TurningVaneDirection:+Z")
scriptInput.append("VaneHeight:0.36")
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/CRAC", [])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/CRAC", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/CRAC", scriptInput)
```

Data Center Components - PDU

PDU enables you to create a fluid block, partitions, and vent geometry and boundary conditions to represent a power distribution unit.

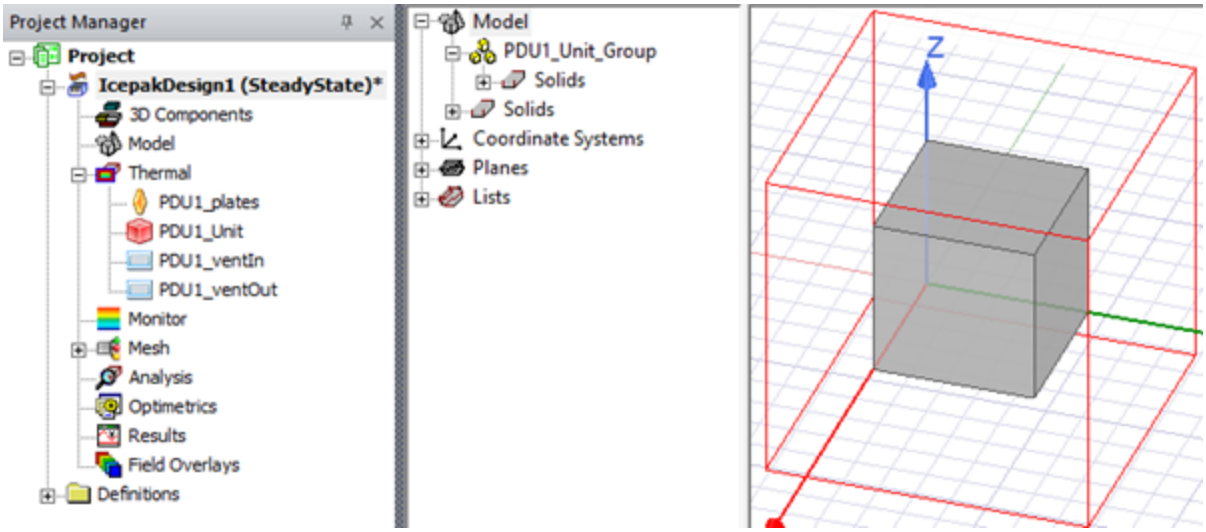
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Location	Under **Location**, specify the x, y, and z start and length location to define the object's size.
PDU Flow Direction	**PDU Flow Direction** specifies the direction in which air flows.
PDU Specs	Under **PDU Specs**, specify the fluid material, heat output, and free area ratio of the inlet and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	outlet.



Iron Python Script – *PDU.py*:

The following is the toolkit's scripting information.

```
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/PDU",
["ng","Name:PDU1","length_unit:meter","power_
unit:W","FluidMaterial:air",
"xS:0","yS:0","zS:0","xL:0.9","yL:0.9","zL:0.9","FlowDirection:-
X",
"HeatOutput:5000","InletAreaRatio:0.5","OutletAreaRatio:0.5"])
scriptInput = ["ng"]
scriptInput.append("Name:PDU1")
scriptInput.append("length_unit:meter")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:0.9")
scriptInput.append("yL:0.9")
scriptInput.append("zL:0.9")
scriptInput.append("FlowDirection:+Y")
scriptInput.append("FluidMaterial:air")
scriptInput.append("HeatOutput:5000")
scriptInput.append("power_unit:W")
scriptInput.append("InletAreaRatio:0.50")
scriptInput.append("OutletAreaRatio:0.50")
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

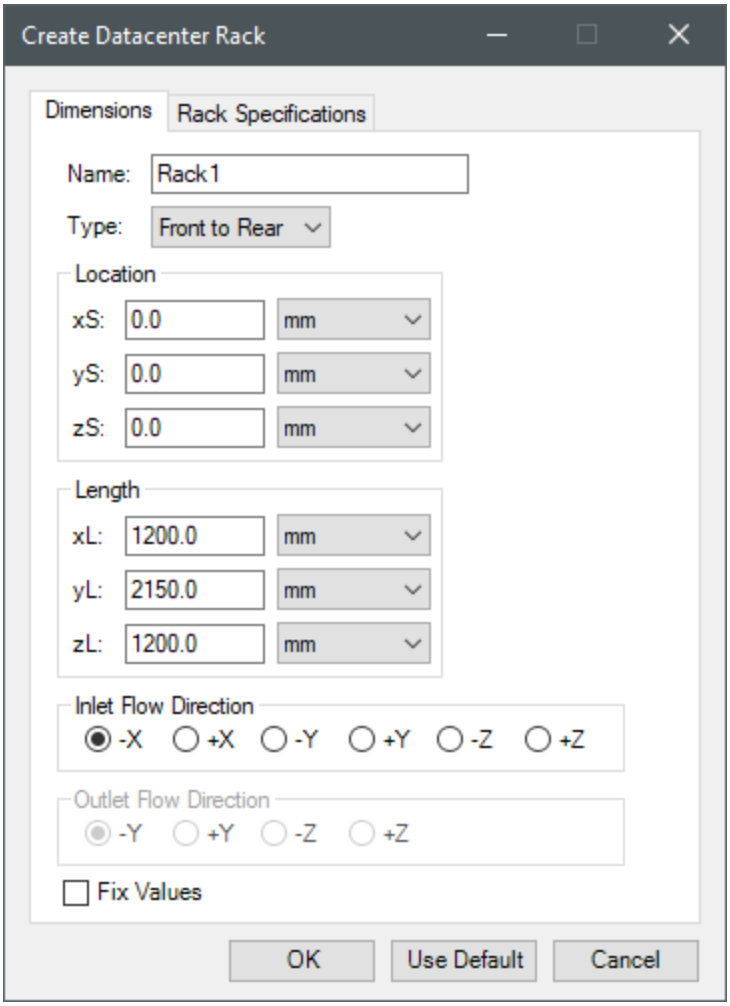
```
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/PDU", [])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/PDU", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/PDU", scriptInput)
```

Data Center Components - Rack (Recirc)

Rack (Recirc) enables you to create geometry and boundary conditions to represent a rack. For **Type**:

- **Rack (Front to Top)** should be chosen when flow enters from the front face and leaves from the top face of the rack.
- **Rack (Front to Rear)** should be chosen when flow enters from one face and leaves from the other face in the same direction.

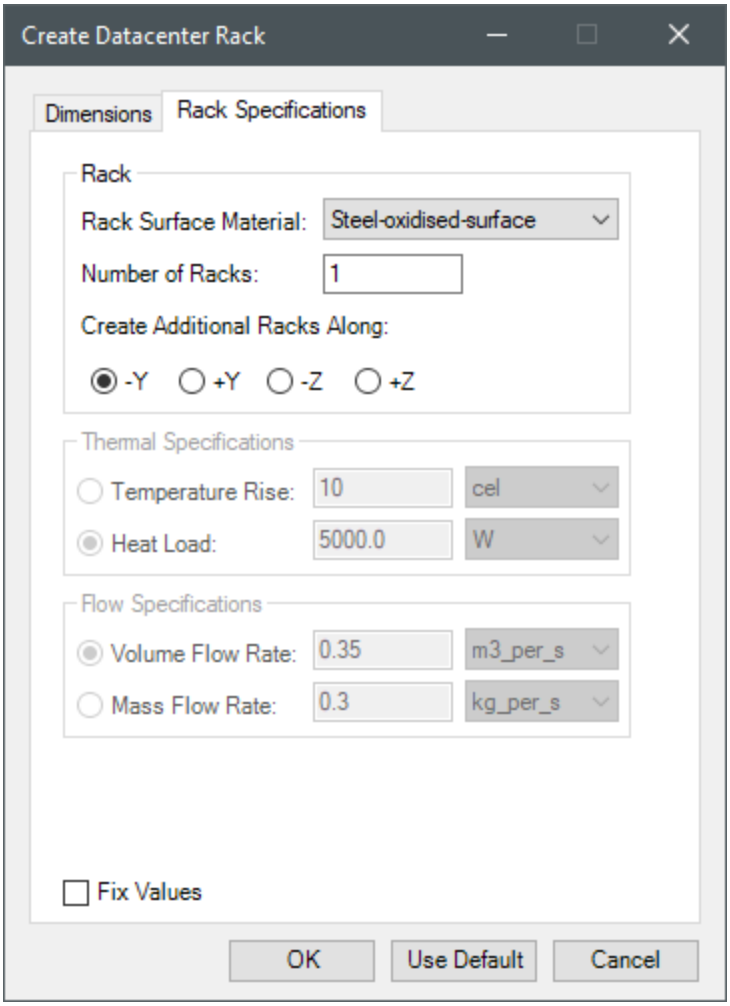
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Type	**Type** specifies the rack type. **Front to Rear** specifies that air flows in a linear fashion from the inlet and outlet. **Front to Top** requires the specification of the **Outlet Flow Direction**.
Location **Length**	Under **Location**, specify the x, y, and z start and length location to define the object's size.
Inlet Flow Direction	**Inlet Flow Direction** specifies the direction air flows into the rack.
Outlet Flow Direction	When **Front to Top** is selected, **Outlet Flow**

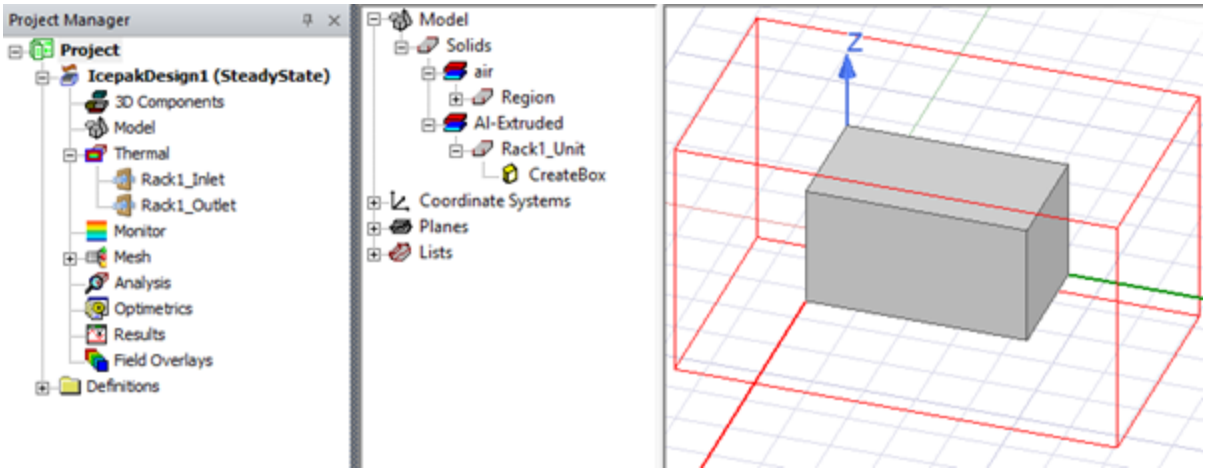
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Direction specifies the direction flows out of the rack.



Rack	Under **Rack**, specify the surface material, the number or racks to create, and the direction along which to create additional racks (if more than one rack is specified).
Thermal Specifications	Under **Thermal Specifications**, specify the rack's thermal properties as **Temperature Rise** or **Heat Load**.
Flow Specifications	Under **Flow Specifications**, specify the rack's fluid properties as **Volume Flow Rate** or **Mass Flow Rate**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *Rack (Recirc).py*:

The following is the toolkit's scripting information.

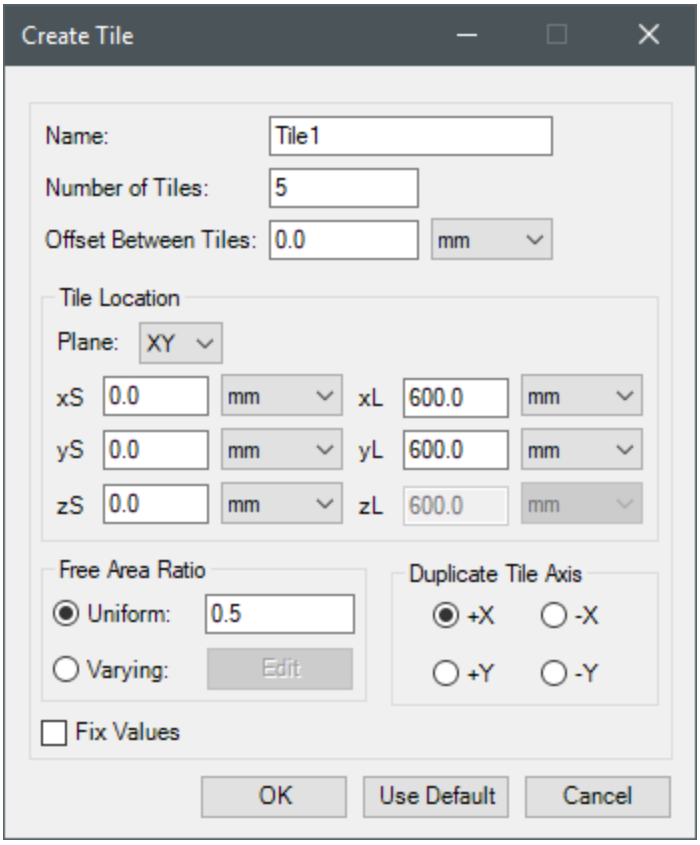
```
scriptInput = ["ng"]
scriptInput.append("Name:Rack1")
scriptInput.append("Type:Front to Top")
scriptInput.append("length_unit:meter")
scriptInput.append("temp_unit:cel")
scriptInput.append("heat_unit:W")
scriptInput.append("VFR_unit:m3_per_s")
scriptInput.append("MFR_unit:kg_per_s")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:1.2")
scriptInput.append("yL:2.15")
scriptInput.append("zL:1.2")
scriptInput.append("InletDirection:-X")
scriptInput.append("OutletDirection:-Y")
scriptInput.append("ThermalSpec:HeatLoad")
scriptInput.append("FlowSpec:VolumeFlowRate")
scriptInput.append("TemperatureRise:10")
scriptInput.append("HeatLoad:5000.0")
scriptInput.append("VFR:0.35")
scriptInput.append("MFR:0.3")
scriptInput.append("Material:Steel-oxidised-surface")
scriptInput.append("Num:2")
scriptInput.append("DuplicationLine:-Y")
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Rack (Recirc)", [])
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Rack (Recirc)", ["ng"])
    oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Rack (Recirc)",
    scriptInput)
```

Data Center Components - Tile

Tile enables you to create geometry and boundary conditions to represent a tile.



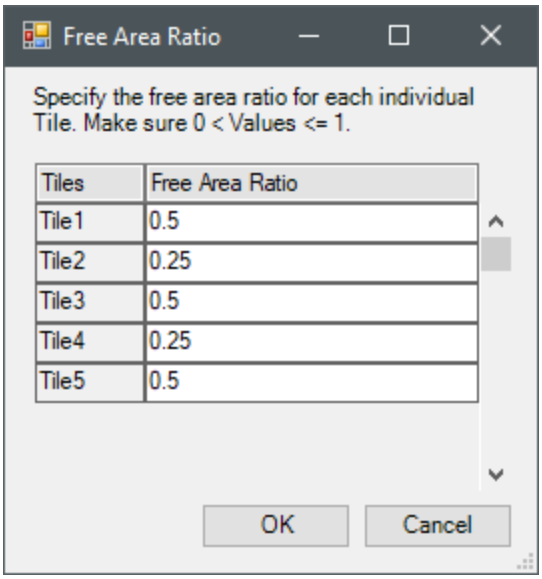
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Number of Tiles	**Number of Tiles** specifies the number of tiles to create.
Tile Location	Under **Tile Location**, specify the plane on which to create the tiles. Also, enter the x, y, and z start and length location to define the object's size.
Free Area Ratio	**Free Area Ratio** specifies whether the tiles have the same free area ratio or varying ratios. See **"Specifying Varying Free Area Ratios" on the facing page** below for more information on specifying varying

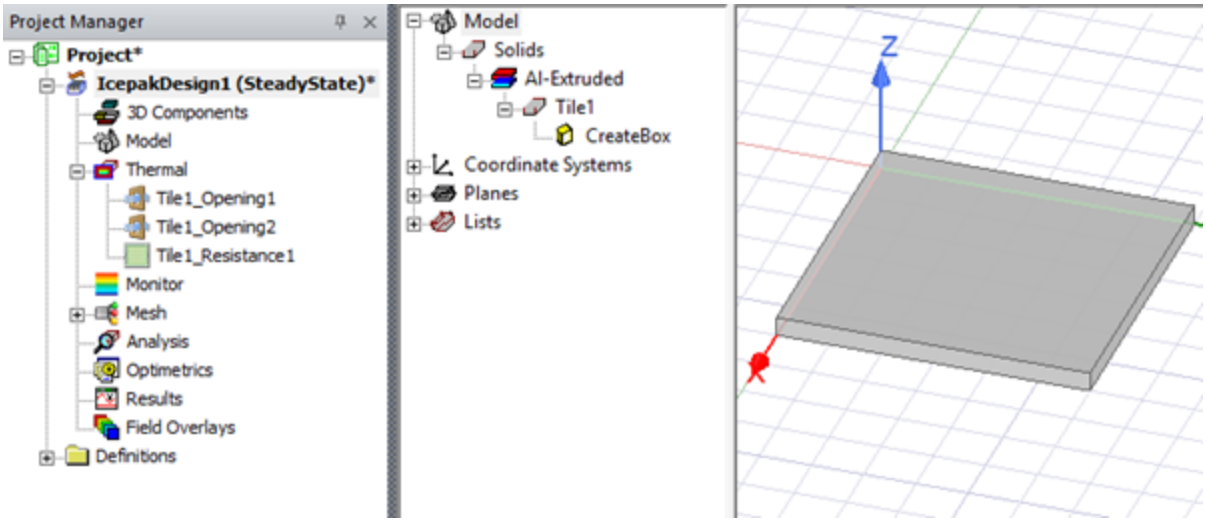
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	ratios.
Duplicate Tile Axis	**Duplicate Tile Axis** specifies the direction along which additional tiles are aligned.

Specifying Varying Free Area Ratios

When **Varying** is enabled, click **Edit** to open the **Free Area Ratio** dialog box. For each tile listed, enter a value to define its free area ratio.





Iron Python Script – *TILE.py*:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

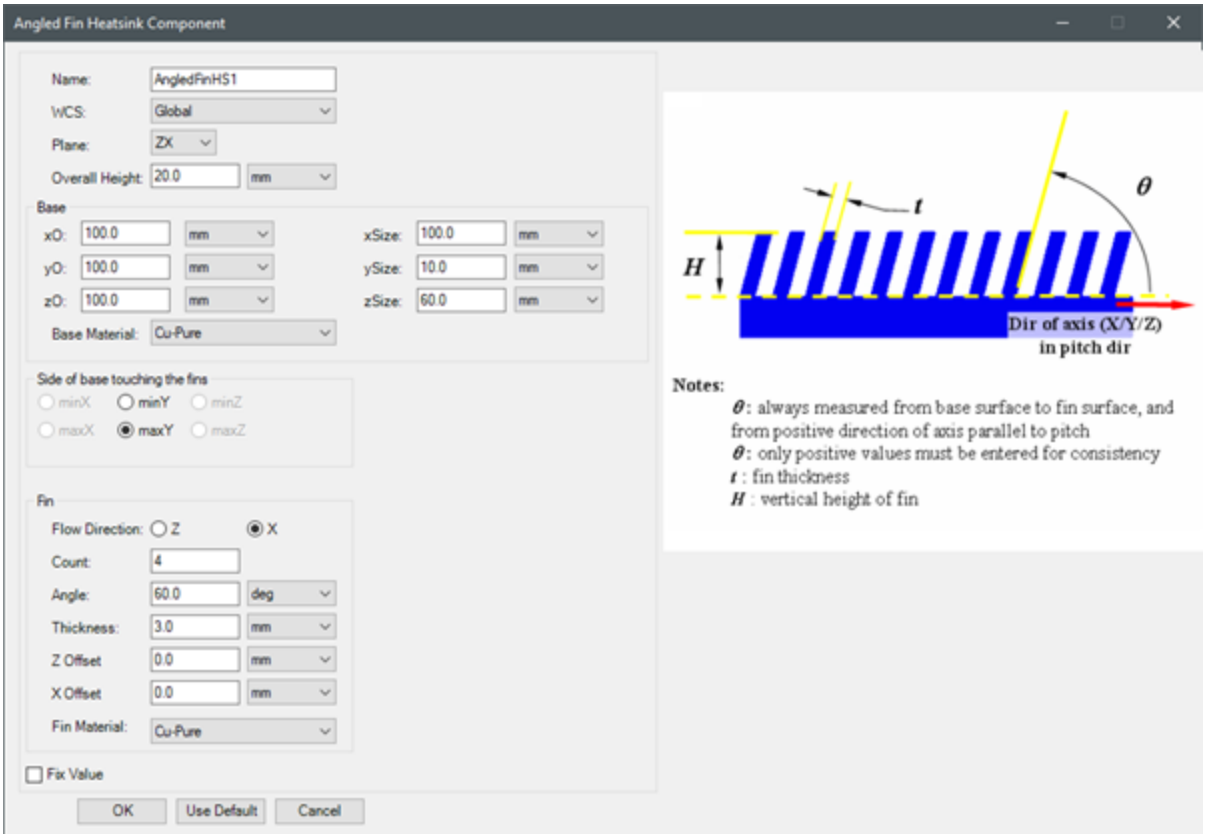
The following is the toolkit's scripting information.

```
### Script for UNIFORM Free Area Ratio
scriptInput = ["ng"]
scriptInput.append("Name:Tile1")
scriptInput.append("length_unit:meter")
scriptInput.append("Number:1")
scriptInput.append("Offset:0.0")
scriptInput.append("Plane:XY")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:0.6")
scriptInput.append("yL:0.6")
scriptInput.append("zL:0.6")
scriptInput.append("FreeAreaRatioType:Uniform")
scriptInput.append("FreeAreaRatio:0.5")
scriptInput.append("DuplicateTileAxis:+X")
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", [])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", scriptInput)
### Script for VARYING Free Area Ratio
scriptInput = ["ng"]
scriptInput.append("Name:Tile1")
scriptInput.append("length_unit:meter")
scriptInput.append("Number:3")
scriptInput.append("Offset:0.0")
scriptInput.append("Plane:ZX")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("xL:0.6")
scriptInput.append("yL:0.6")
scriptInput.append("zL:0.6")
scriptInput.append("FreeAreaRatioType:Varying")
scriptInput.append("FreeAreaRatio:0.5,0.4,0.3")
scriptInput.append("DuplicateTileAxis:-X")
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", [])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Data Center
Components/Tile", scriptInput)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

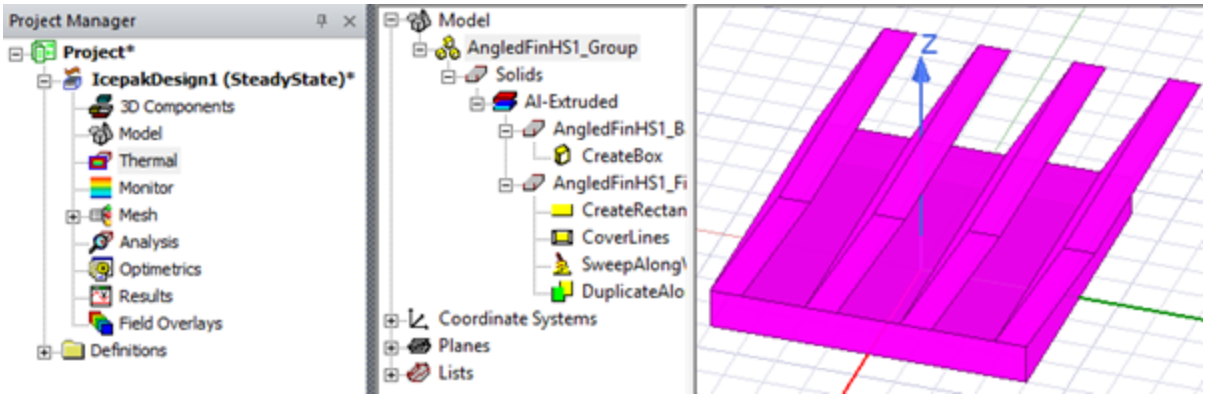
Heatsinks - AngledFinHS

AngledFinHS enables you to create a heat sink with angled fins. The angle of inclination of the fins relative to the base can be specified.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
WCS	**WCS** specifies the coordinate system on which to create the heatsink.
Plane	**Plane** specifies the plane on which to create the heatsink.
Overall Height	**Overall Height** specifies the cumulative height (base and fins) of the heatsink.
Base	Under **Base**, specify the x, y, and z start and length location to define the object's size. Also, specify the base material.
Side of base touching fins	**Side of base touching fins** specifies which face of the heatsink the fins touch.
Fin	Under **Fin**, specify the flow direction, the fin count, the fin angle, fin thickness, the offsets, and fin material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *AngledFinHSpy*:

The following is the toolkit's scripting information.

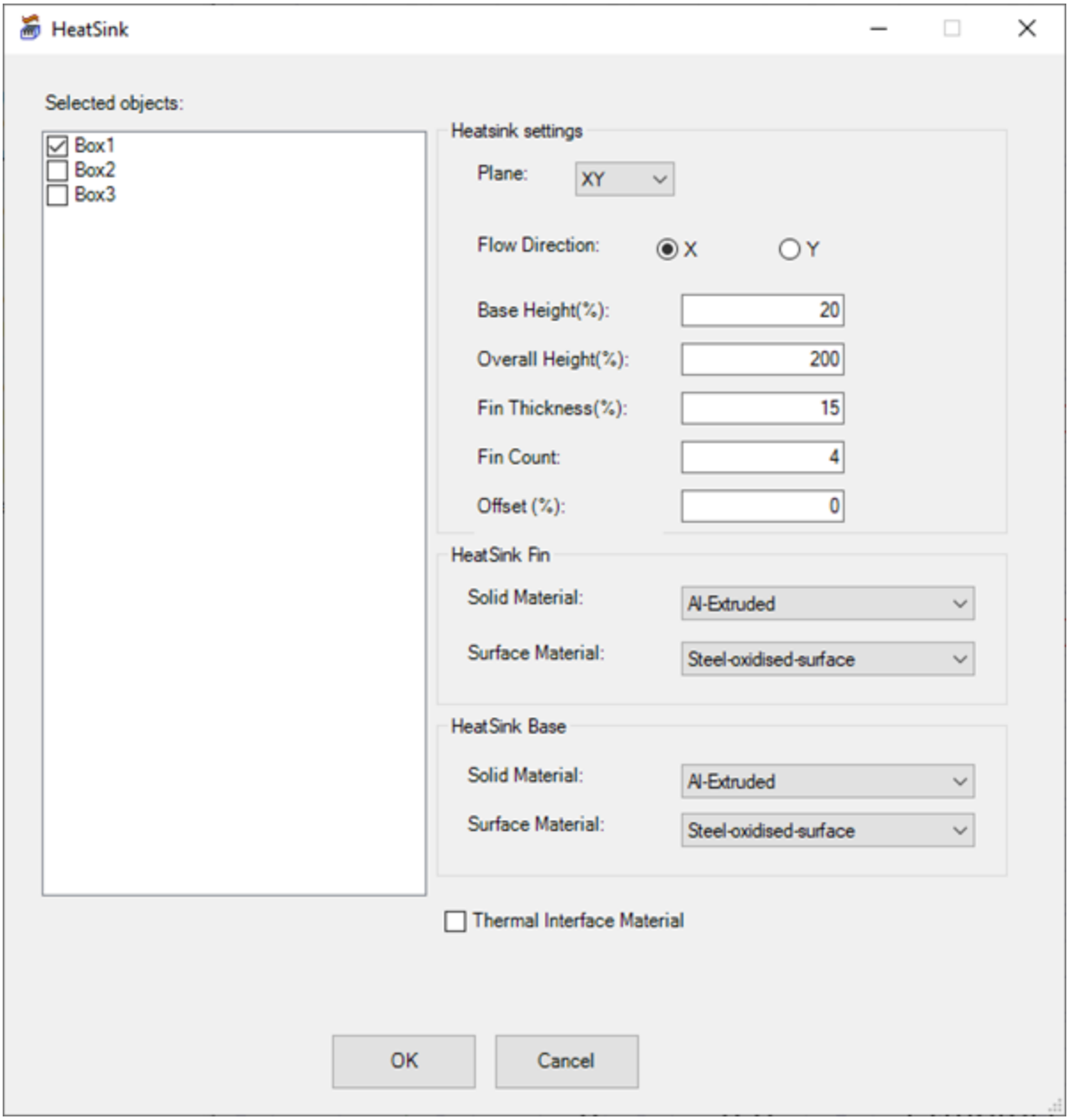
```
scriptInput = ["ng"]
scriptInput.append("Name:AngledHS1")
scriptInput.append("length_unit:meter")
scriptInput.append("angle_unit:deg")
scriptInput.append("Base_Material:Al-Extruded")
scriptInput.append("WCS:Global")
scriptInput.append("Plane:ZX")
scriptInput.append("Baseside:maxY")
scriptInput.append("Overall Height:0.02")
scriptInput.append("xOrigin:0.1")
scriptInput.append("yOrigin:0.1")
scriptInput.append("zOrigin:0.1")
scriptInput.append("Len2:0.015")
scriptInput.append("xSize:0.1")
scriptInput.append("ySize:0.01")
scriptInput.append("zSize:0.06")
scriptInput.append("Flow:X")
scriptInput.append("Count:4")
scriptInput.append("Angle:120")
scriptInput.append("Thickness:0.003")
scriptInput.append("Offset1:0.0")
scriptInput.append("Offset2:0.0")
oDesign.RunToolkit("SysLib", "Geometry/Heatsinks/AngledFinHS",
[])
oDesign.RunToolkit("SysLib", "Geometry/Heatsinks/AngledFinHS",
["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Heatsinks/AngledFinHS",
scriptInput)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heatsinks - Heatsink Creation

Heatsink Creation enables you to create one or more heatsinks on selected geometry.

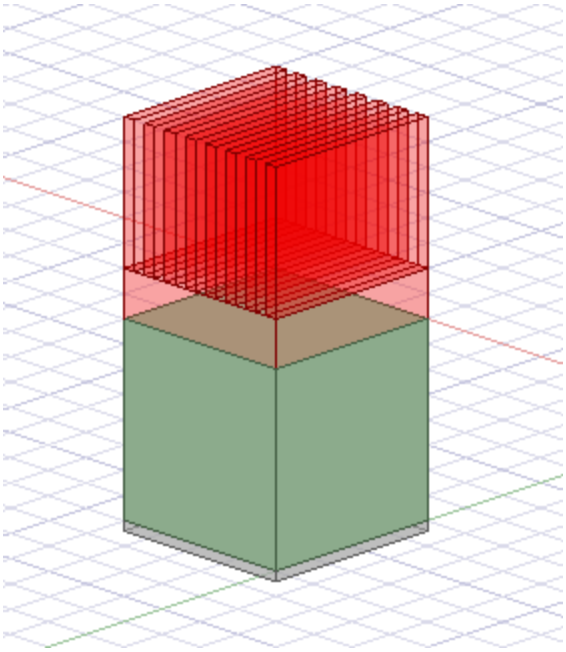
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selected Objects	Under **Selected objects**, select the geometry on which to place the heatsink(s). **Note**: Before opening the **HeatSink** toolkit, you can select geometry in the History tree to automatically select it in the **HeatSink** dialog box.
Heatsink Settings	
Plane	**Plane** specifies the plane on which to create the heatsink(s).
Flow Direction	**Flow Direction** specifies the direction in which the length of the fins run along the base of the heatsink, or the direction of the flow through the domain.
Base Height(%)	**Base Height(%)**specifies the height of the heatsink's base as a percentage of overall height.
Overall Height(%)	**Overall Height(%)** specifies the overall height as a percentage relative to the height of the geometry on which it is placed.
Fin Thickness(%)	**Fin Thickness** specifies the thickness of the fins as a percentage of the base width.
Fin Count	**Fin Count** specifies the number of fins on the heatsink.
Offset(%)	**Offset(%)** specifies the offset of the fins in the direction parallel to the base.
Heatsink Fin	
Solid Material	**Solid Material** specifies the material type of the heatsink fins using the drop-down list.
Surface Material	**Surface Material** specifies the material type of the external sides of the heatsink fins using the drop-down list.
Heatsink Base	
Solid Material	**Solid Material** specifies the material type of the heatsink base using the drop-down list.
Surface Material	**Surface Material** specifies the material type of the external sides of the heatsink base using the drop-down list.
Thermal Interface Material	Enable **Thermal Interface Material** to create a box between the heatsink and the selected box geometry. The Thermal Interface Material geometry is assigned Copper as its material.

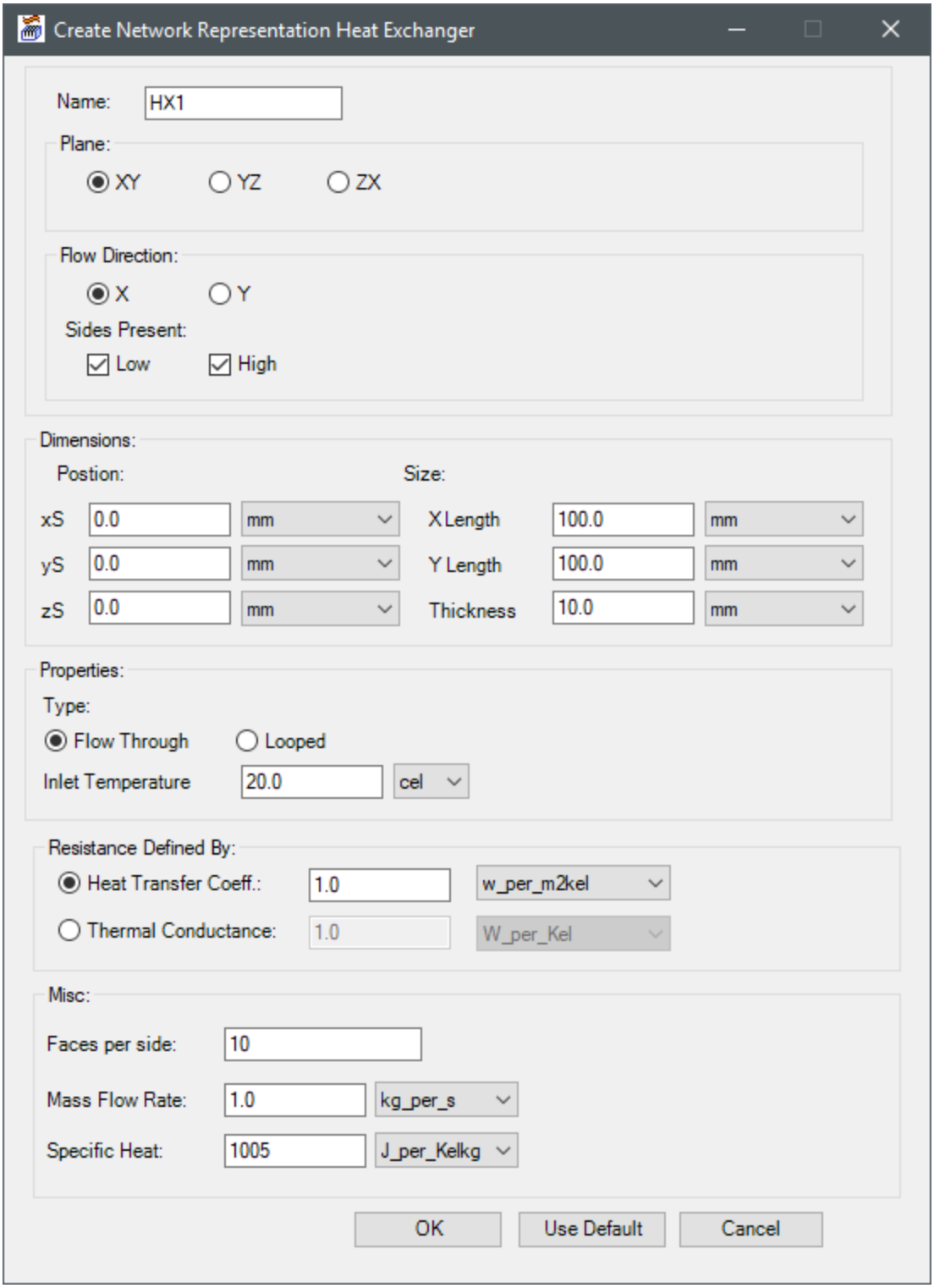
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Other - Create Network Representation Heat Exchanger

Create Network Representation Heat Exchanger enables you to create a flow-through or looped heat exchanger.

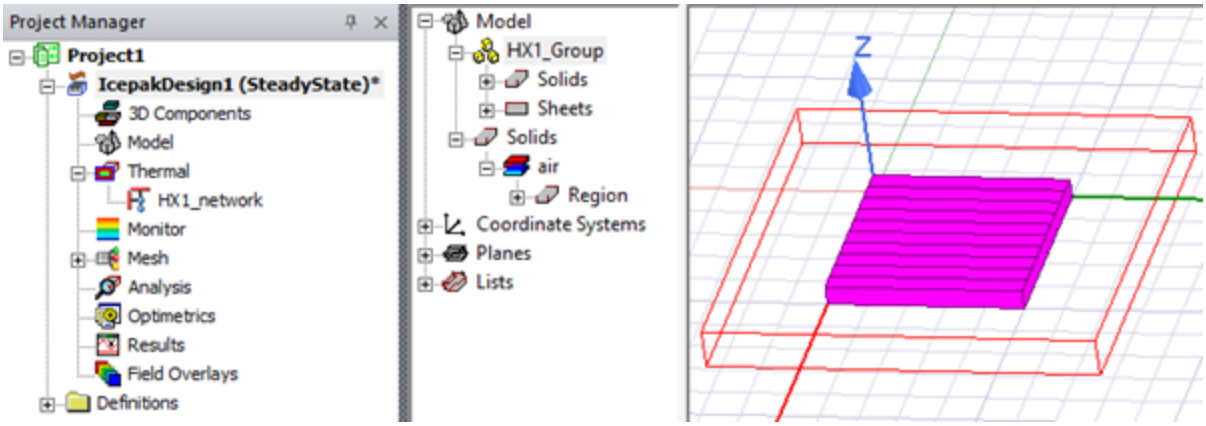
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the heat exchanger.
Flow Direction	Under **Flow Direction**, specify the direction of the flow and which sides are present (low and/or high).
Dimensions	Under **Dimensions**, specify the x, y, and z coordinate locations and the length and thickness of the heat exchanger.
Properties	Under **Properties**, specify **Flow Through** or **Looped**. For flow through heat exchangers, specify the **Inlet Temperature**. For looped, specify the **Ambient Temperature**.
Resistance Defined By	Under **Resistance Defined By**, define resistance by specifying a heat transfer coefficient or thermal conductance.
Misc	Under **Misc**, specify the number of faces on each side and the **Mass Flow Rate** and **Specific Heat**.



Iron Python Script – *Network_Representation_Heat_Exchanger.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
 'Length Unit:meter',
 'Temperature Unit:cel',
 'HTC Unit:w_per_m2kel',
 'Thermal Conductance Unit:W_per_Kel',
 'Mass Flow Unit:kg_per_s',
 'Specific Heat Unit:J_per_Kelkg',
 'Plane:XY',
 'xS:0.0',
 'yS:0.0',
 'zS:0.0',
 'Length1:0.1',
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

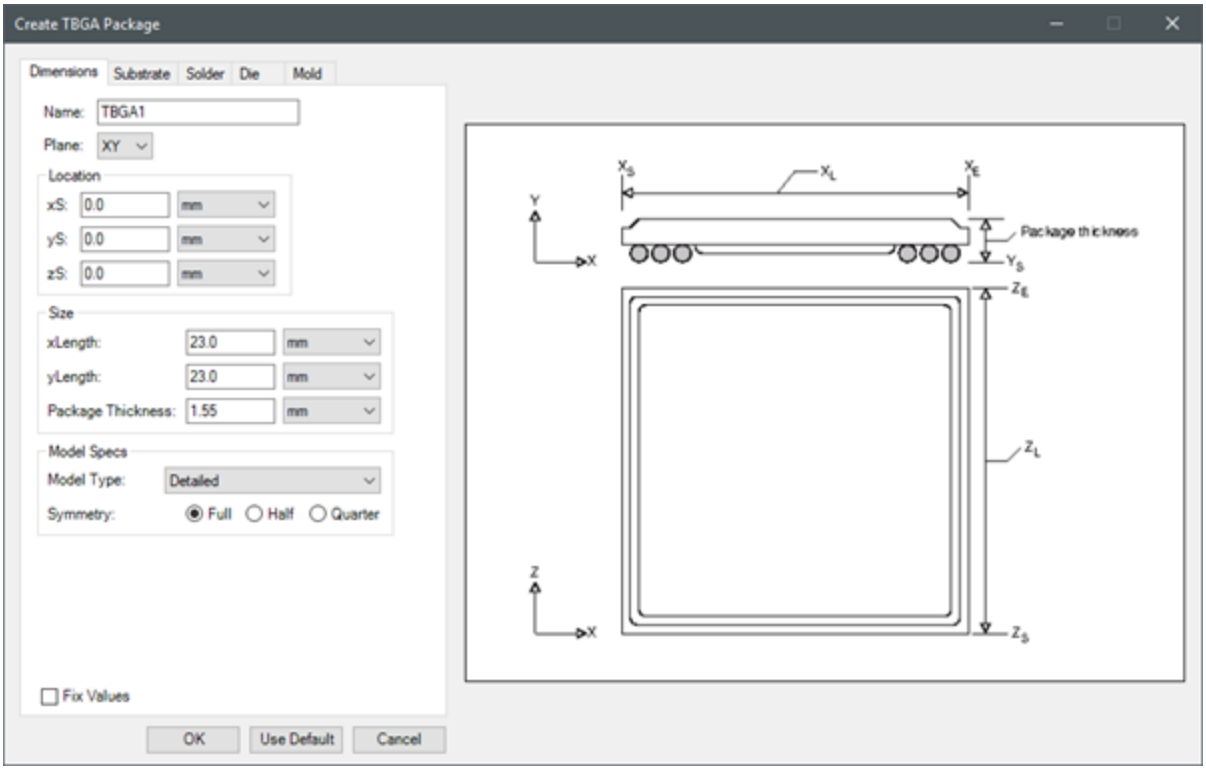
```
    'Length2:0.1',
    'Thickness:0.01',
    'Flow Direction:1',
 'Low Side Present:True',
 'High Side Present:True',
 'Type:Flow Through',
 'Temperature:20.0',
 'Resistance Defined By:HTC',
 'HTC Value:1.0',
 'Thermal Conductance Value:1.0',
 'Faces per side:10',
 'Mass Flow Rate:1.0',
 'Specific Heat:1005'
 ]

oDesign.RunToolkit("SysLib", "Geometry/Other/Network_
Representation_Heat_Exchanger", data)
```

Packages - Cavity_Down_BGA

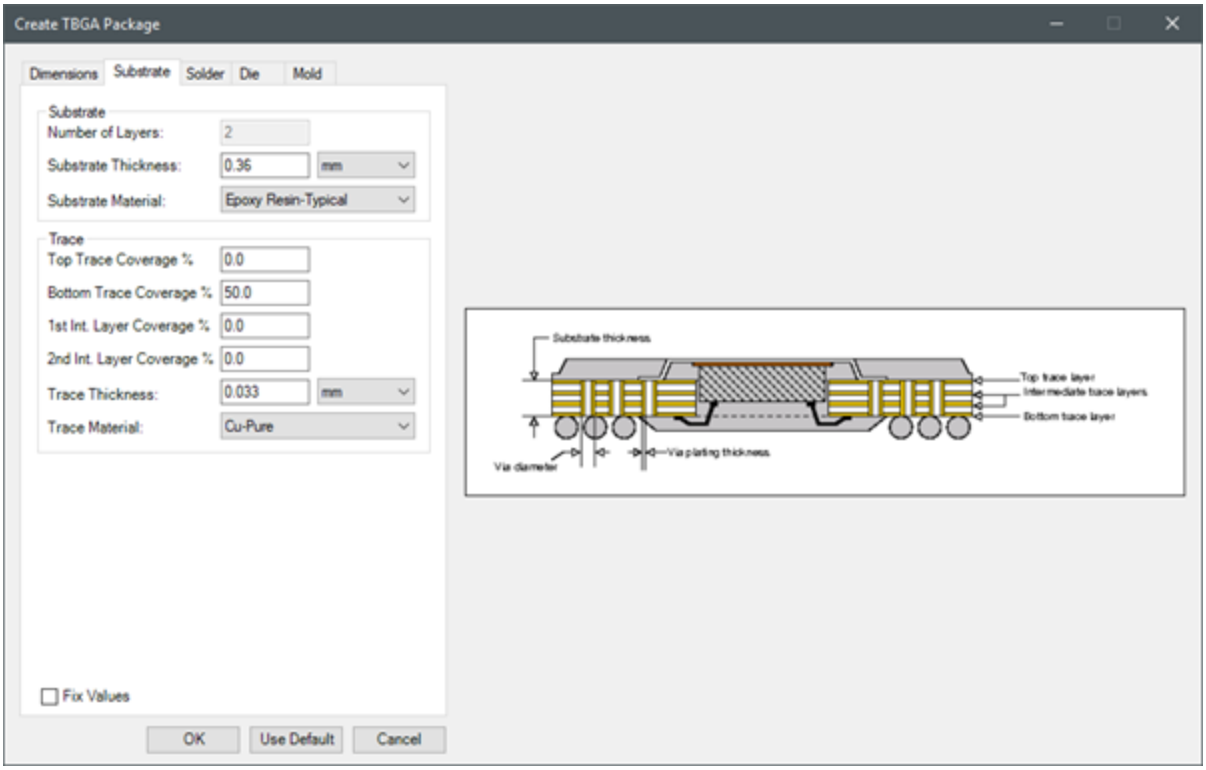
Packages - Cavity_Down_BGA enables you to create geometry and boundary conditions to represent a cavity-down ball grill array package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



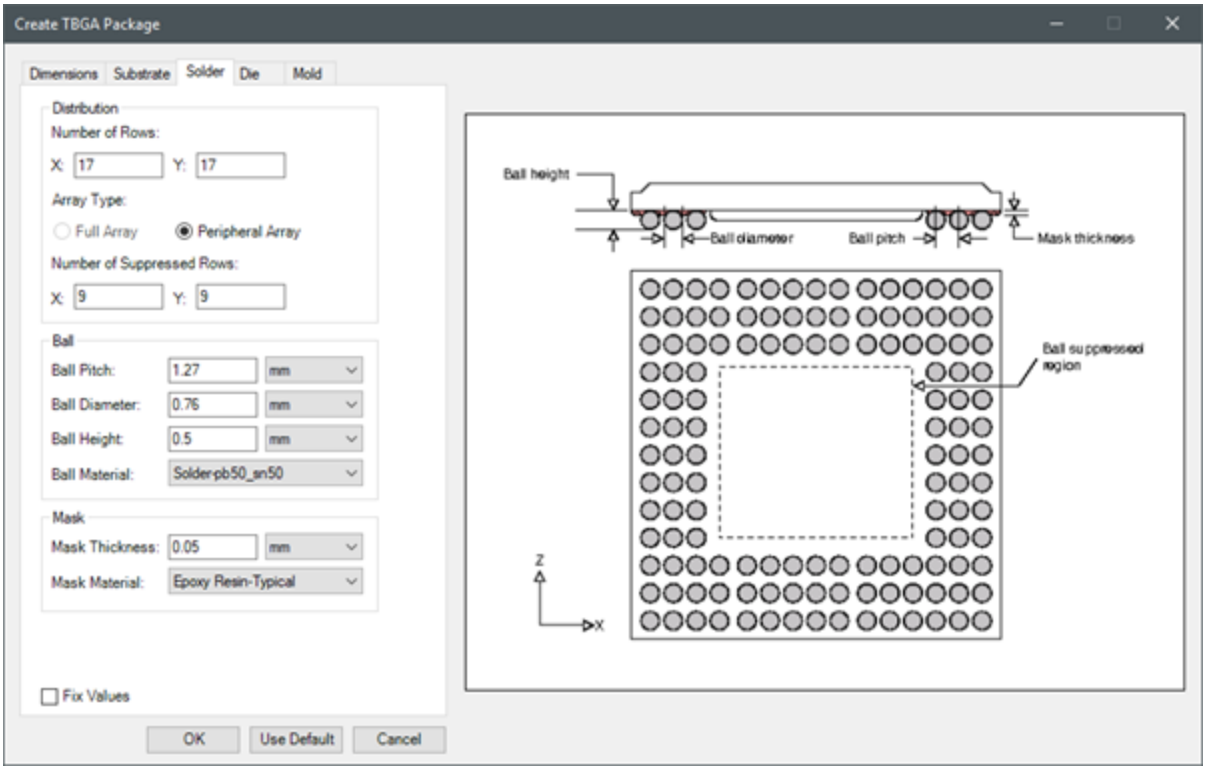
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



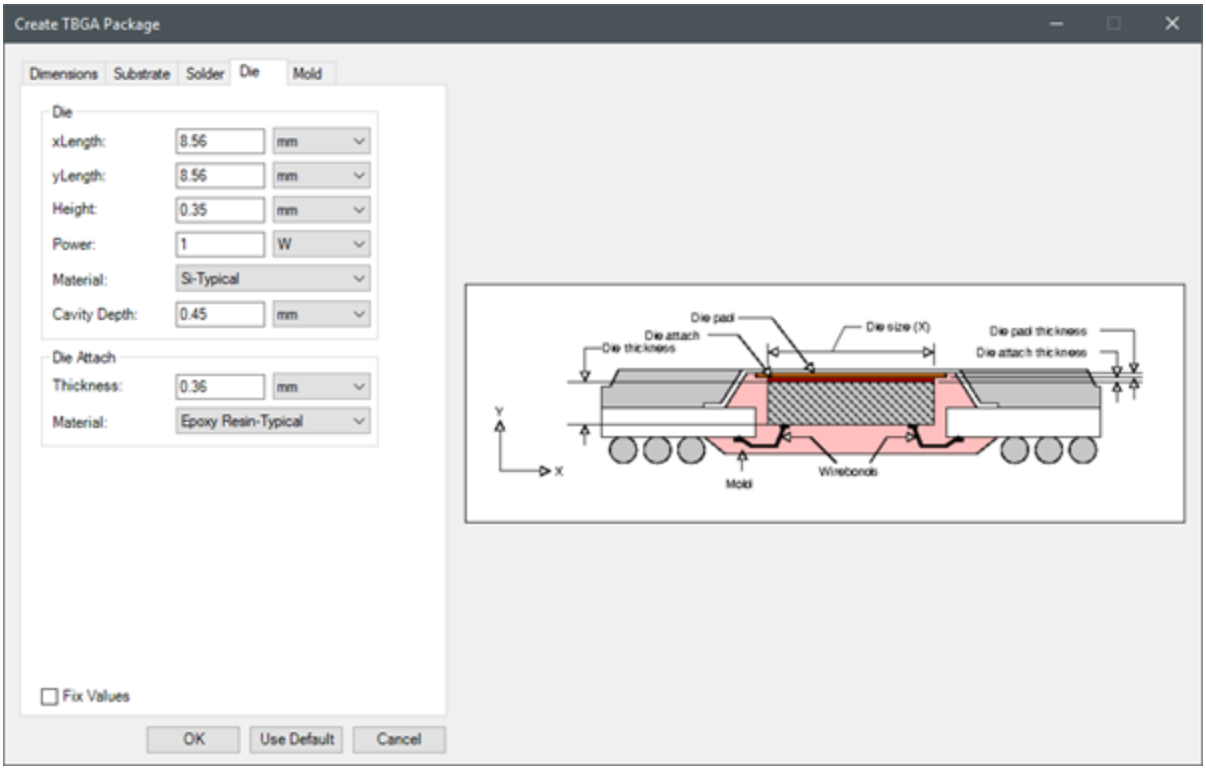
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



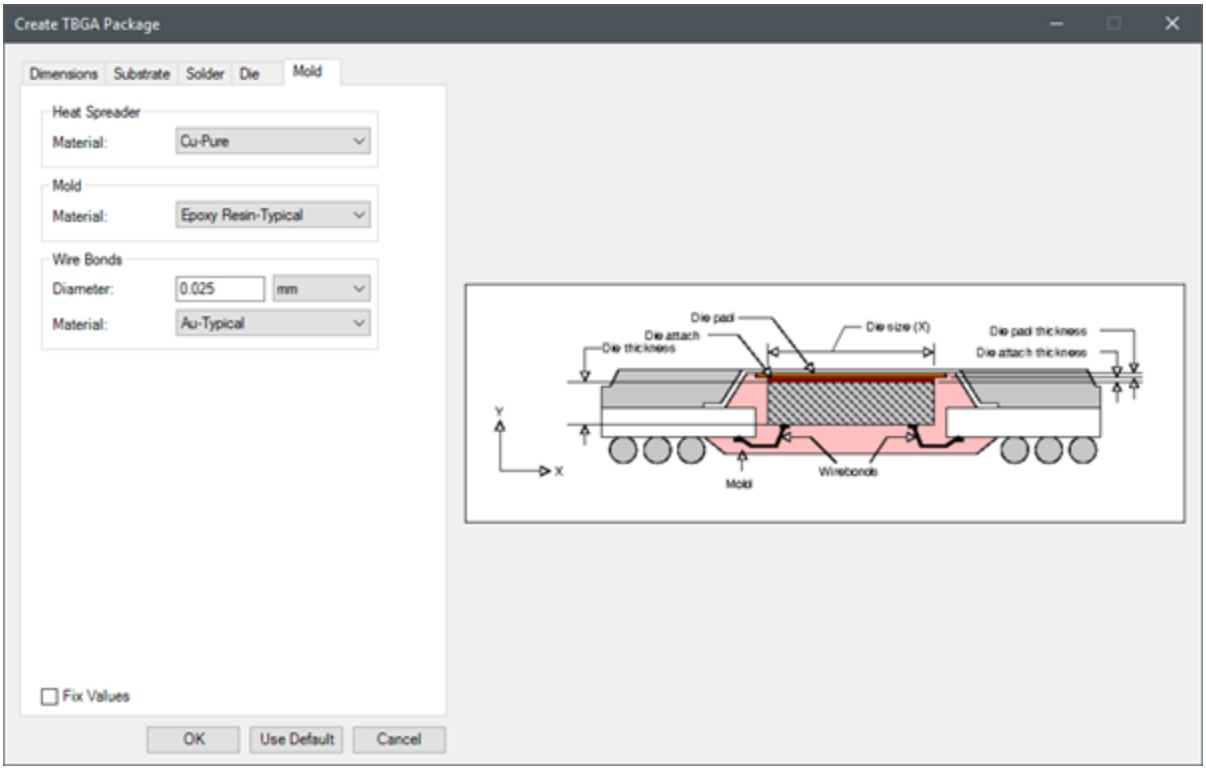
Distribution	Under **Distribution**, enter the number of rows for solder balls and the number of rows to be suppressed.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.
Mask	Under **Mask**, enter the solder ball mask thickness and mask material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

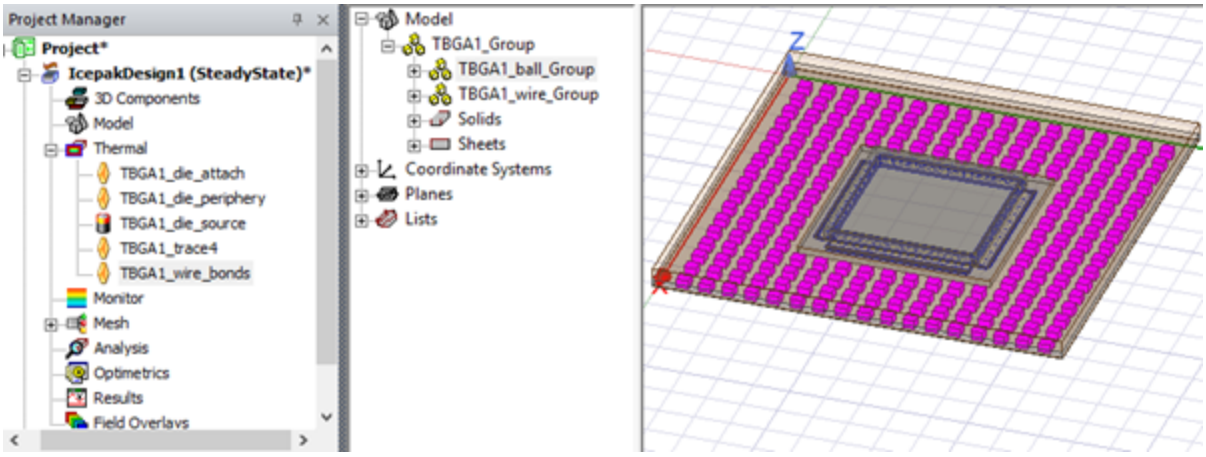


Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material and the depth of the die cavity.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Heat Spreader	Under **Heat Spreader**, select the material for the heat spreader.
Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter and material of the wire bonds.



Iron Python Script – *Cavity_Down_BGA.py*:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is the toolkit's scripting information.

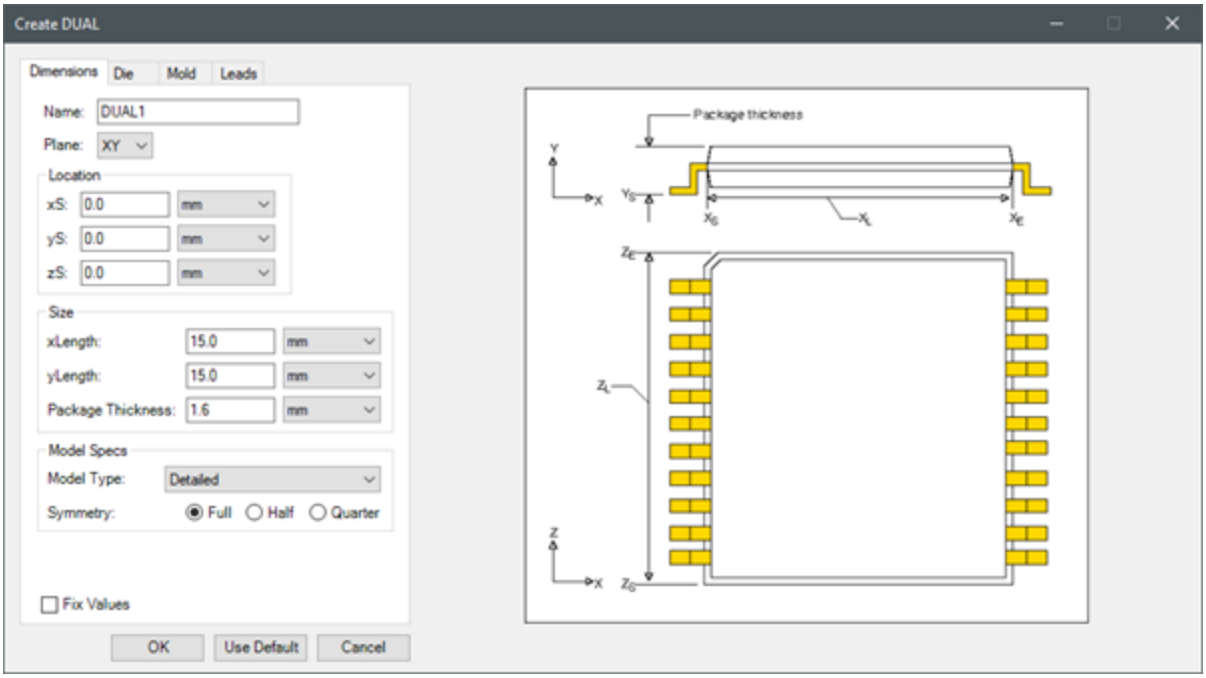
```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.023',
        'Length2:0.023',
        'Package Thickness:0.00155',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Substrate Thickness:0.00036',
        'Substrate Material:Epoxy Resin-Typical',
        'Top Trace:0.0',
        'Bottom Trace:50.0',
        '1st Int. Layer Coverage:0.0',
        '2nd Int. Layer Coverage:0.0',
        'Trace Thickness:0.000033',
        'Trace Material:Cu-Pure',
        'Number of Rows 1:17',
        'Number of Rows 2:17',
        'Suppressed Rows 1:9',
        'Suppressed Rows 2:9',
        'Ball Pitch:0.00127',
        'Ball Diameter:0.00076',
        'Ball Height:0.0005',
        'Ball Material:Solder-pb50_sn50',
        'Mask Thickness:0.00005',
        'Mask Material:Epoxy Resin-Typical',
        'Die Length1:0.00856',
        'Die Length2:0.00856',
        'Die Height:.00035',
        'Power:1',
        'Die Material:Si-Typical',
        'Cavity Depth:0.00045',
        'Attach Thickness:0.00036',
        'Attach Material:Epoxy Resin-Typical',
        'Heat Spreader Material:Cu-Pure',
        'Mold Material:Epoxy Resin-Typical',
        'Wire Diameter:0.000025',
        'Wire Material:Au-Typical'
        ]
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Cavity_Down_
BGA", [])
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Cavity_Down_
BGA", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Cavity_Down_
BGA", data)
```

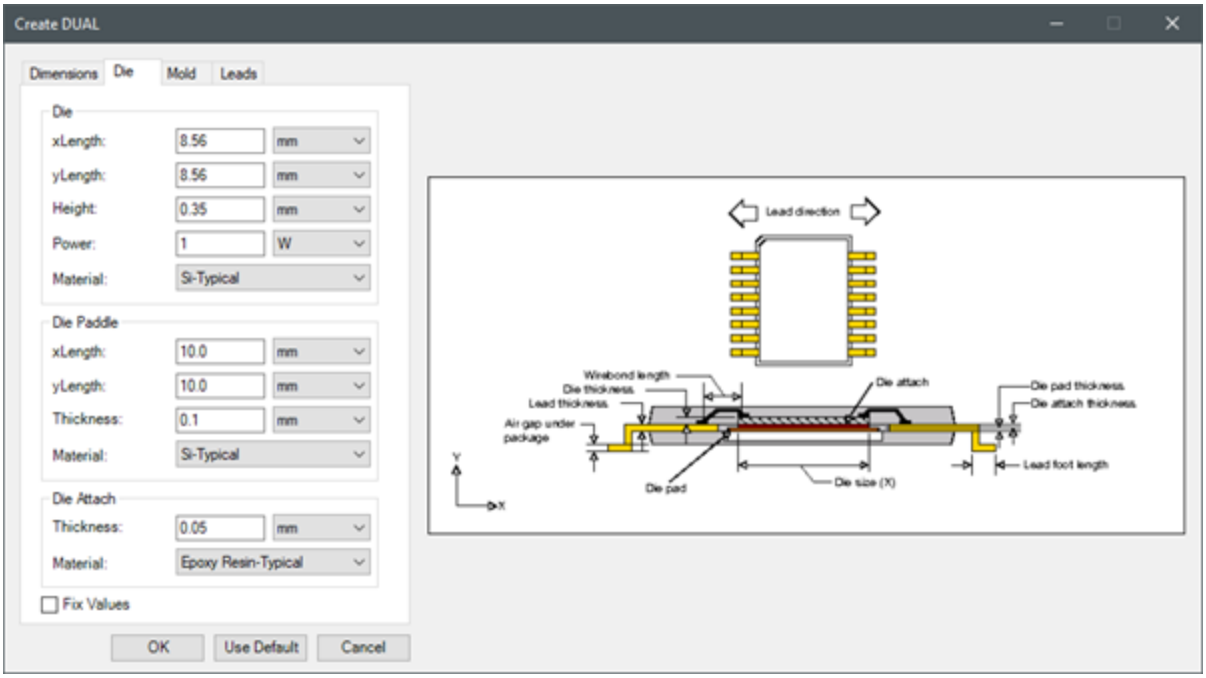
Packages - DUAL

DUAL enables you to create geometry and boundary conditions to represent a DUAL package.



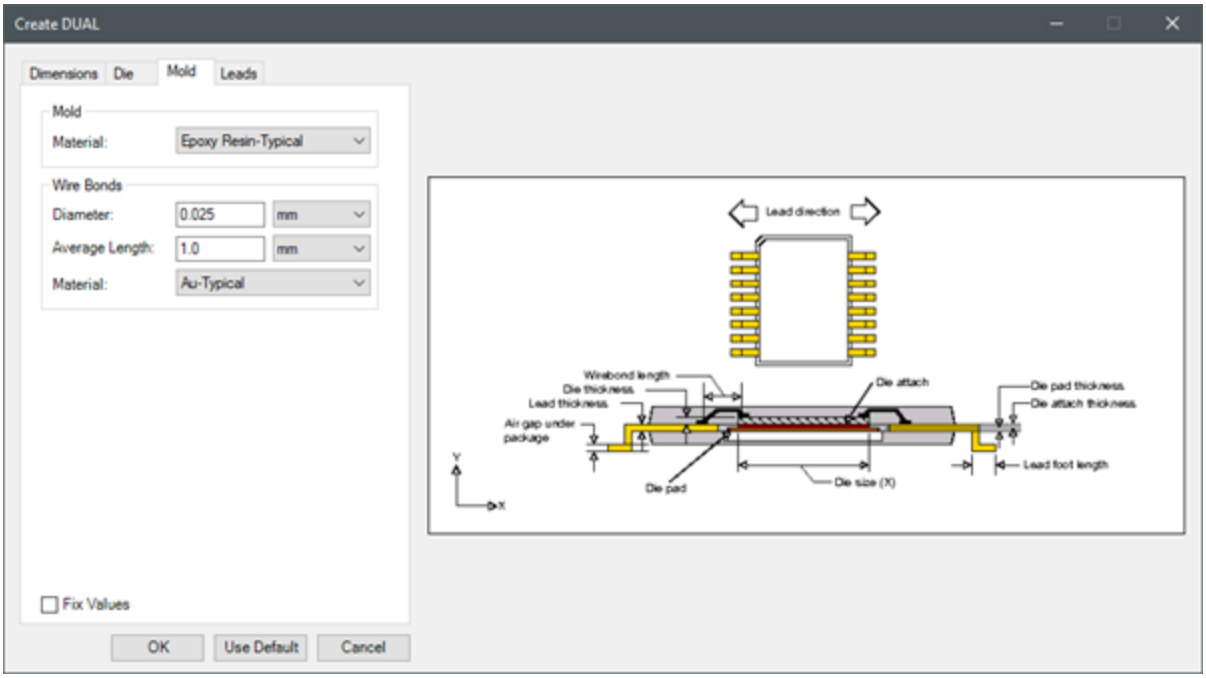
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed** or **Compact Conduction Model (CCM)** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



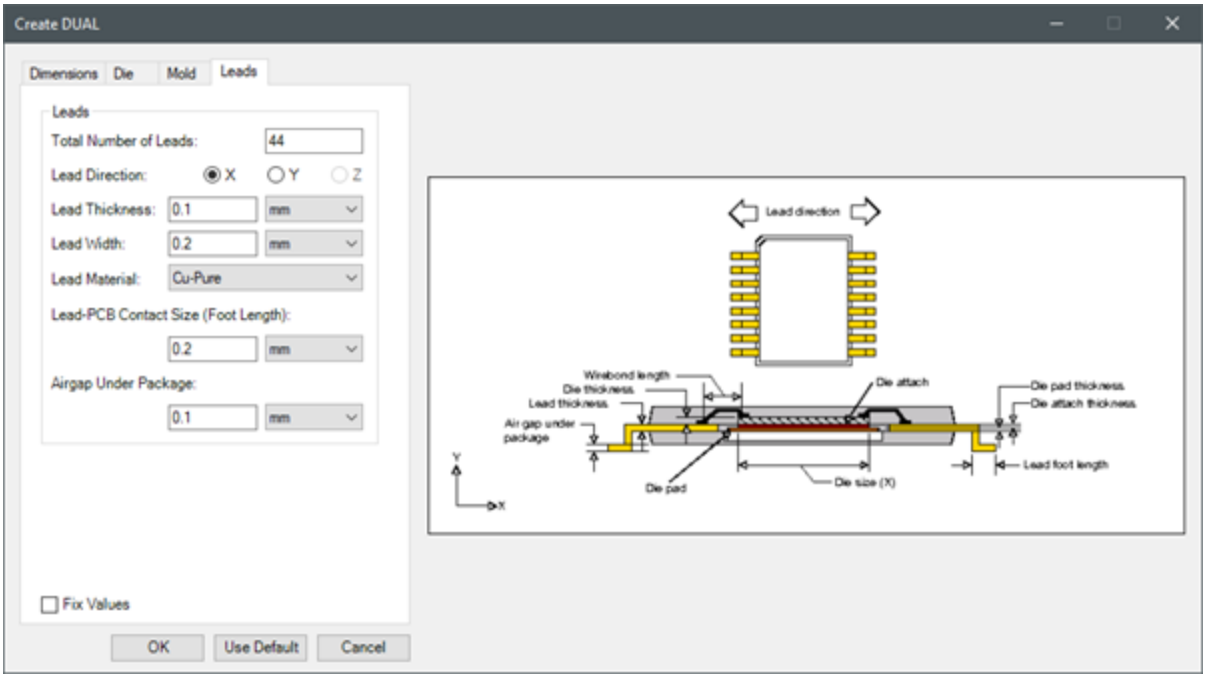
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

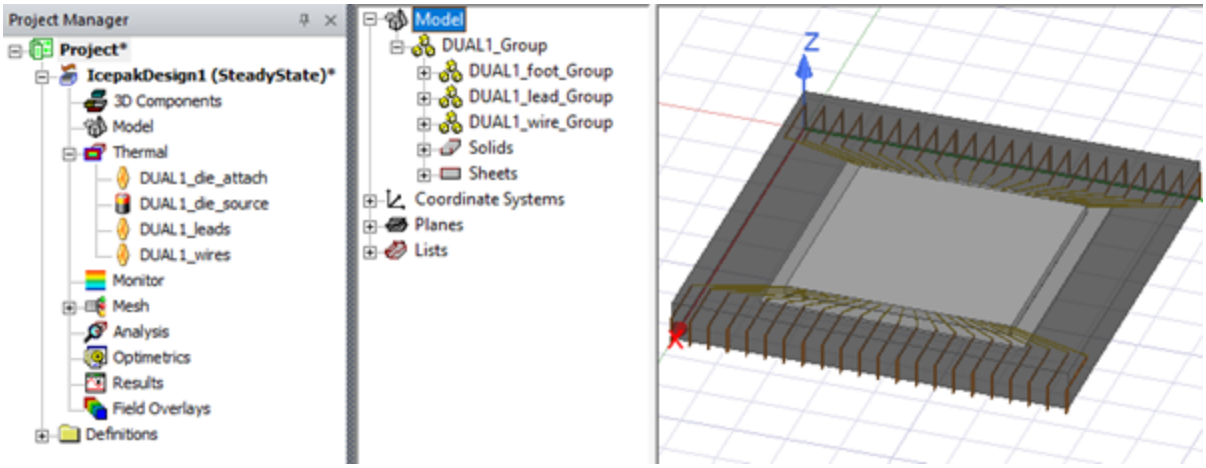


Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads, the direction in which the leads will point, the lead thickness and width, material, foot length, and airgap under the package.



Iron Python Script – *DUAL.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
```

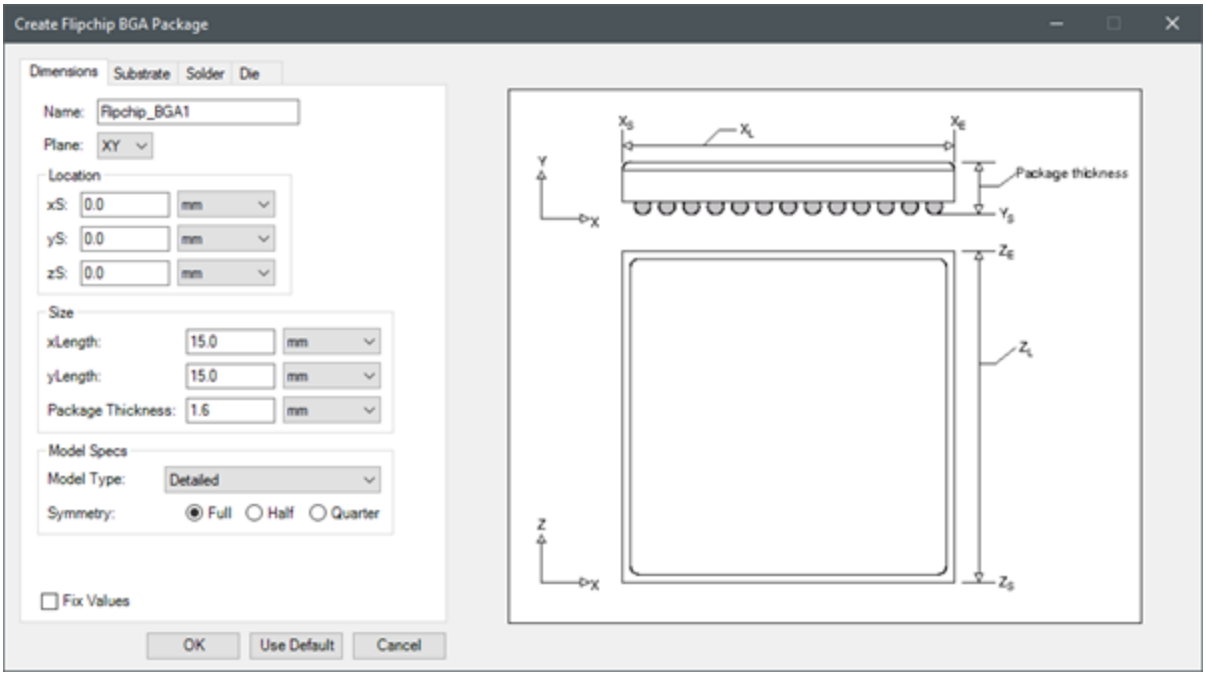

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.015',
        'Length2:0.015',
        'Package Thickness:0.0016',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Die Length1:0.00856',
        'Die Length2:0.00856',
        'Die Height:.00035',
        'Power:1',
        'Die Material:Si-Typical',
        'Pad Length1:0.01',
        'Pad Length2:0.01',
        'Pad Thickness:0.0001',
        'Pad Material:Si-Typical',
        'Attach Thickness:0.00005',
        'Attach Material:Epoxy Resin-Typical',
        'Mold Material:Epoxy Resin-Typical',
        'Wire Diameter:0.000025',
        'Wire Length:0.001',
        'Wire Material:Au-Typical',
        'Total Number of Leads:44',
        'Lead Direction:X',
        'Lead Thickness:0.0001',
        'Lead Width:0.0002',
        'Lead Material:Cu-Pure',
        'Lead-PCB Contact Size:0.0002',
        'Airgap:0.0001'
        ]
oDesign.RunToolkit("SysLib", "Geometry/Packages/DUAL", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/DUAL", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Packages/DUAL", data)
```

Packages - Flipchip_BGA

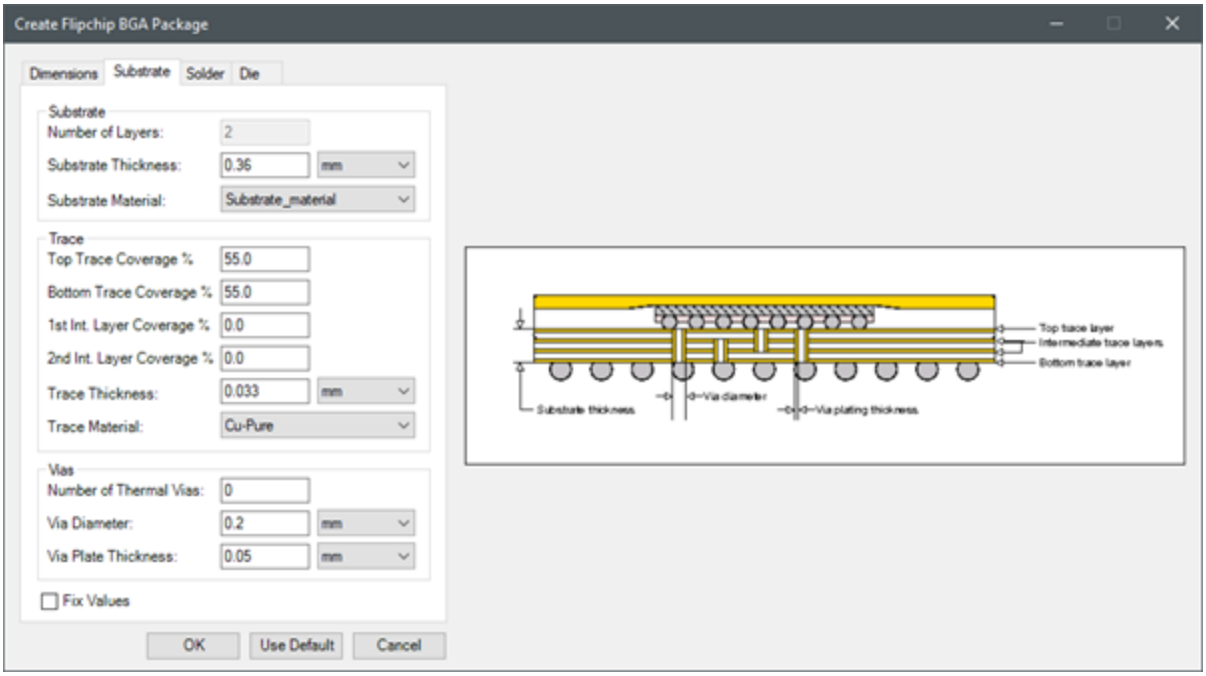
Flipchip_BGA enables you to create geometry and boundary conditions to represent a flipchip ball grid array package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



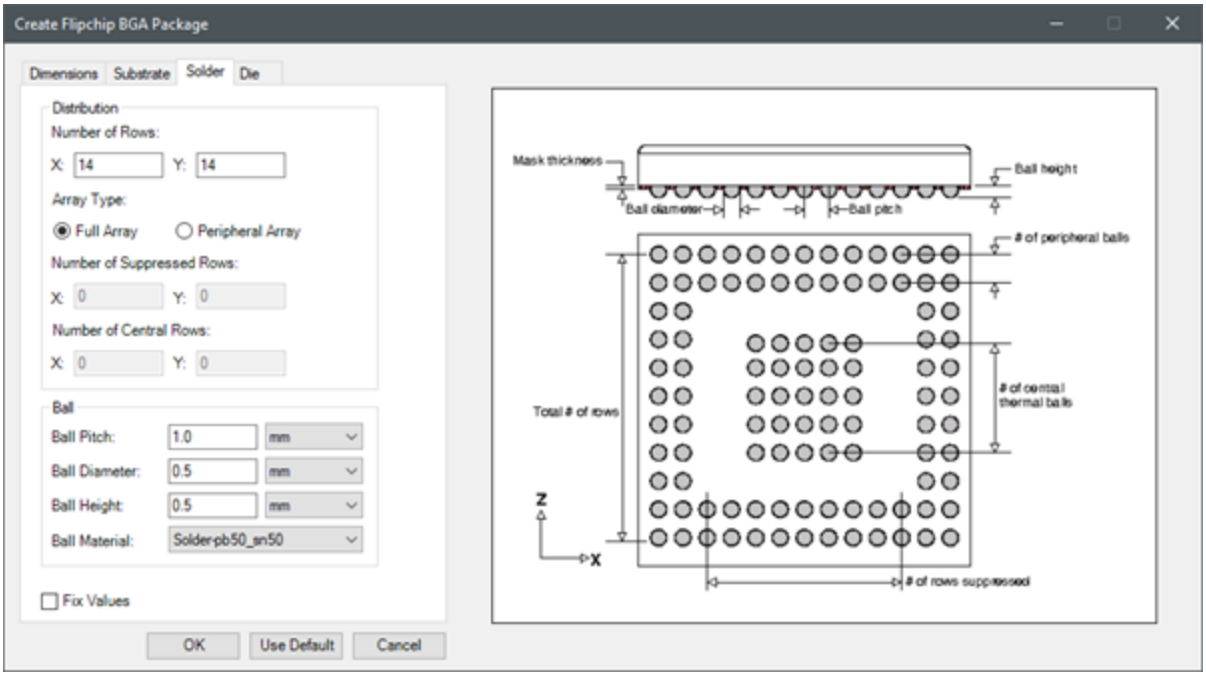
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



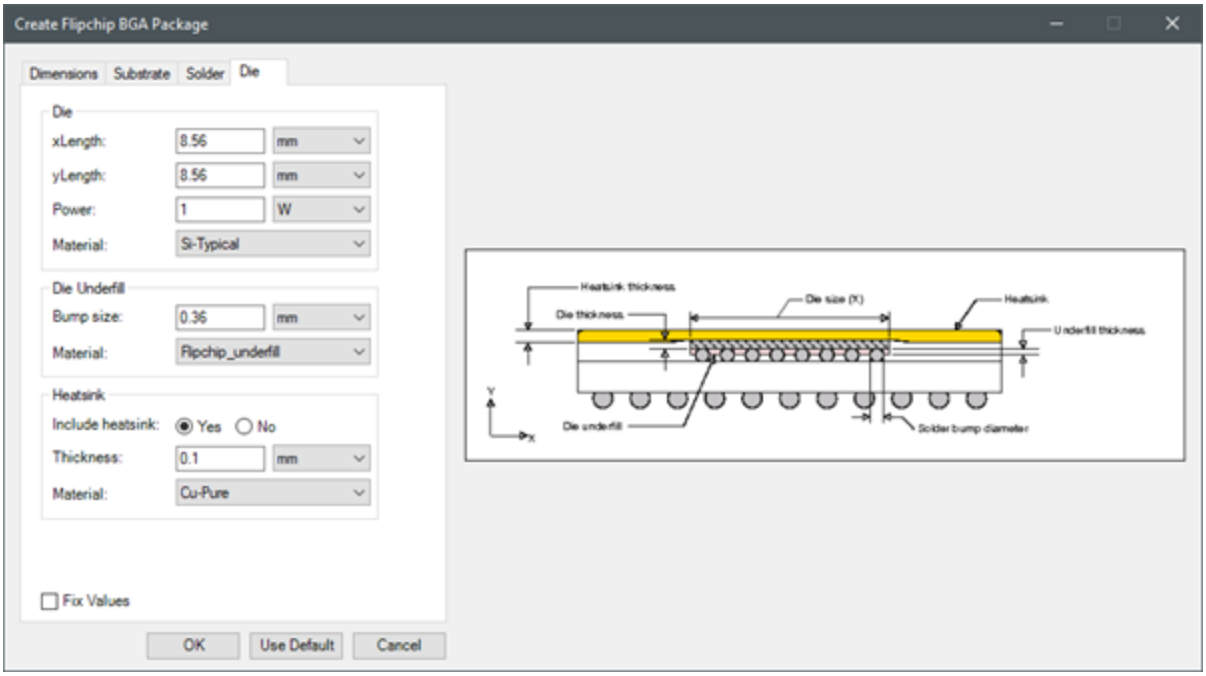
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

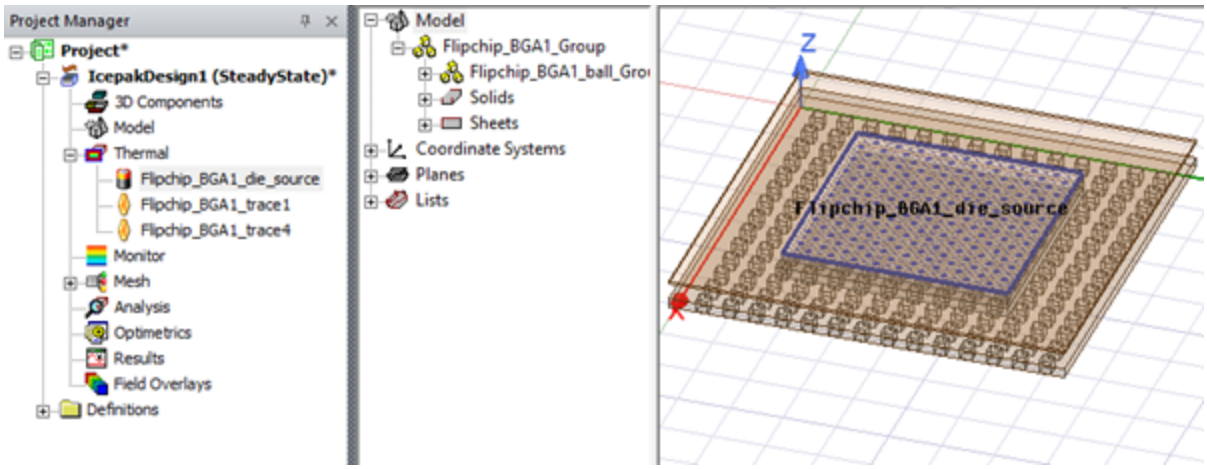


Distribution	Under **Distribution**, enter the number of rows for solder balls, the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed and number of central rows.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Die	Under **Die**, specify the length of the die in each coordinate direction. Also, specify the die power and material.
Die Underfill	Under **Die Attach**, specify the bump size of the die underfill material and the material.
Heatsink	Under **Heatsink**, specify whether to include a heatsink in the package. If so, specify the heatsink thickness and material.



Iron Python Script – *Flipchip_BGA.py*:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is the toolkit's scripting information.

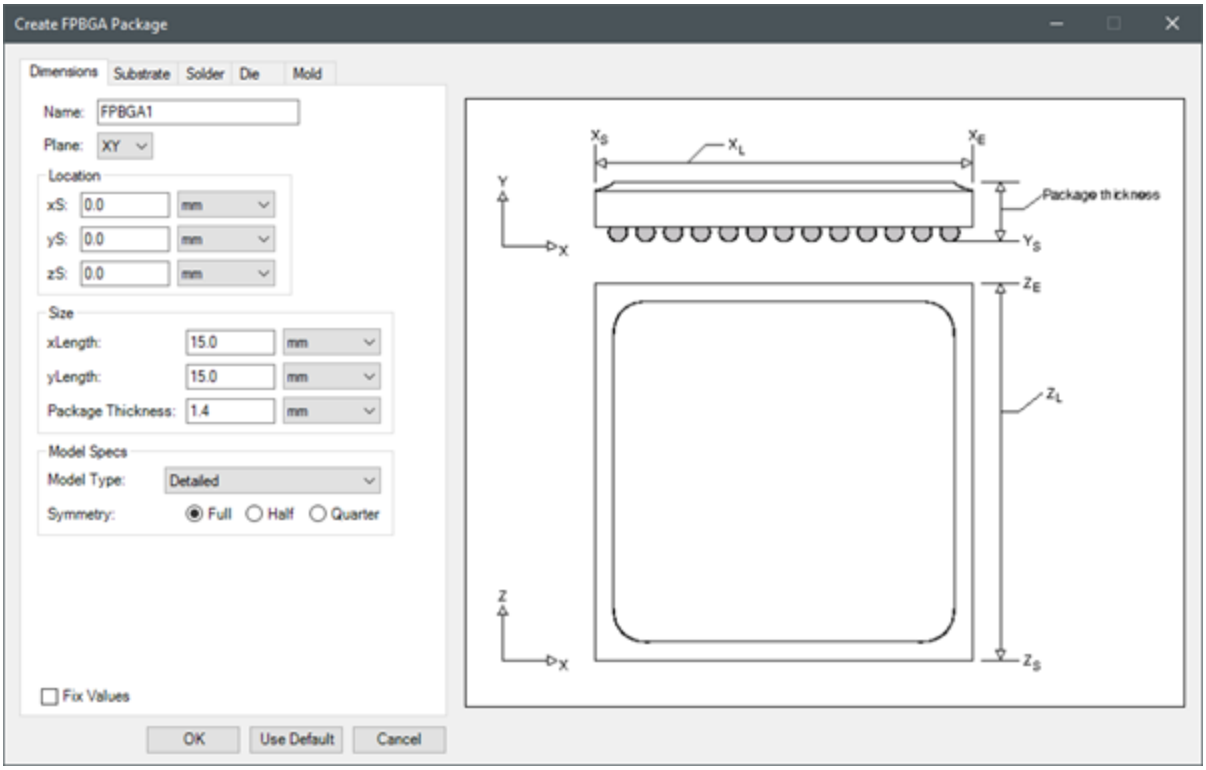
```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.015',
        'Length2:0.015',
        'Package Thickness:0.0016',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Substrate Thickness:0.00036',
        'Substrate Material:Epoxy Resin-Typical',
        'Top Trace:55.0',
        'Bottom Trace:55.0',
        '1st Int. Layer Coverage:0.0',
        '2nd Int. Layer Coverage:0.0',
        'Trace Thickness:0.000033',
        'Trace Material:Cu-Pure',
        'Number of Thermal Vias:0',
        'Via Diameter:0.0002',
        'Via Plate Thickness:0.00005',
        'Number of Rows 1:14',
        'Number of Rows 2:14',
        'Array Type:Full Array',
        'Suppressed Rows 1:0',
        'Suppressed Rows 2:0',
        'Central Rows 1:0',
        'Central Rows 2:0',
        'Ball Pitch:0.001',
        'Ball Diameter:0.0005',
        'Ball Height:0.0005',
        'Ball Material:Solder-pb50_sn50',
        'Die Length1:0.00856',
        'Die Length2:0.00856',
        'Power:1',
        'Die Material:Si-Typical',
        'Underfill Bump Size:0.00036',
        'Underfill Material:Flipchip_underfill',
        'Include Heatsink:Yes',
        'Heatsink Thickness:0.0001',
        'Heatsink Material:Cu-Pure'
        ]
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Flipchip_BGA",
[])
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Flipchip_BGA",
["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages/Flipchip_BGA",
data)
```

Packages - FPBGA

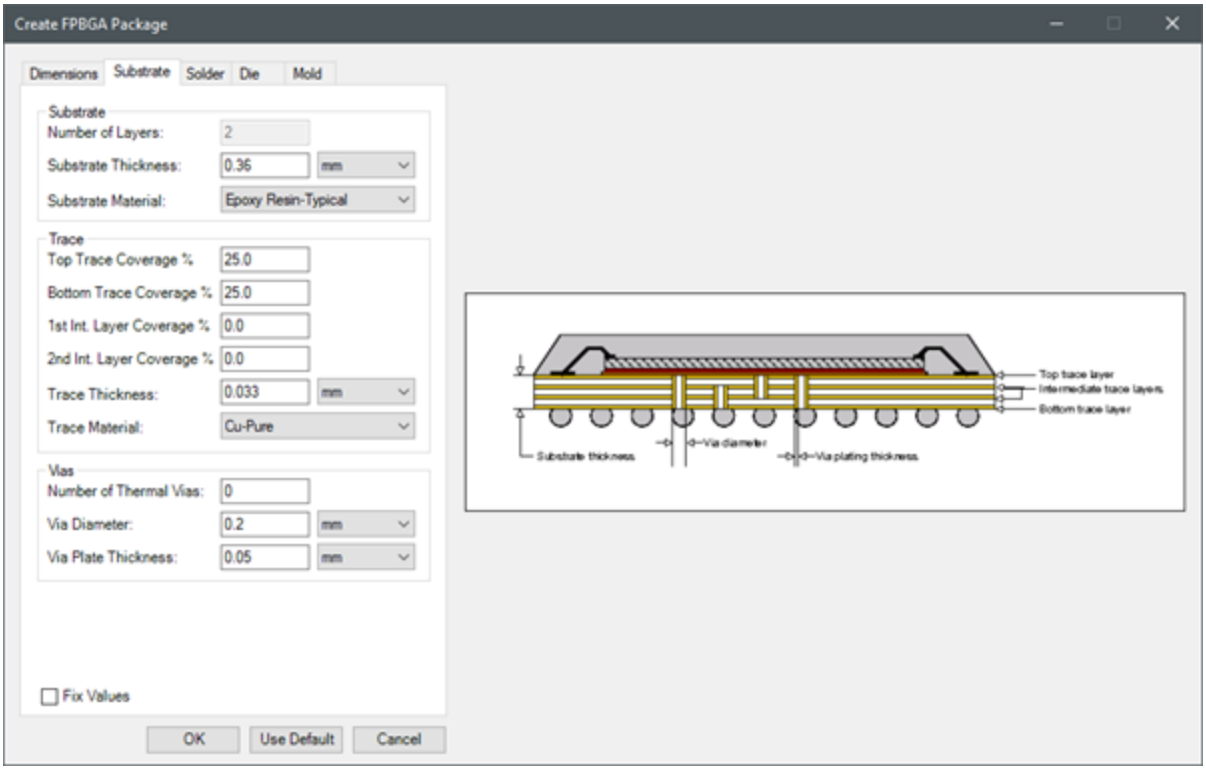
FPBGA enables you to create geometry and boundary conditions to represent a fine pitch ball grid array package.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.

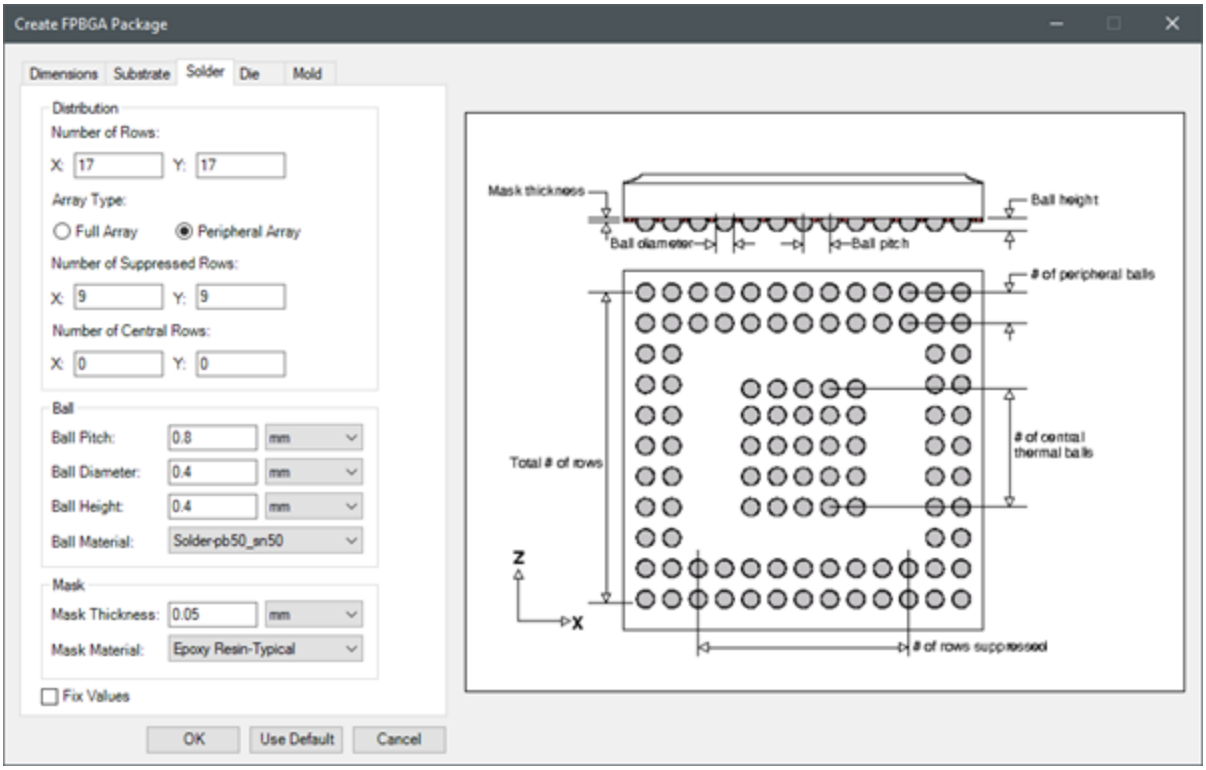
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.



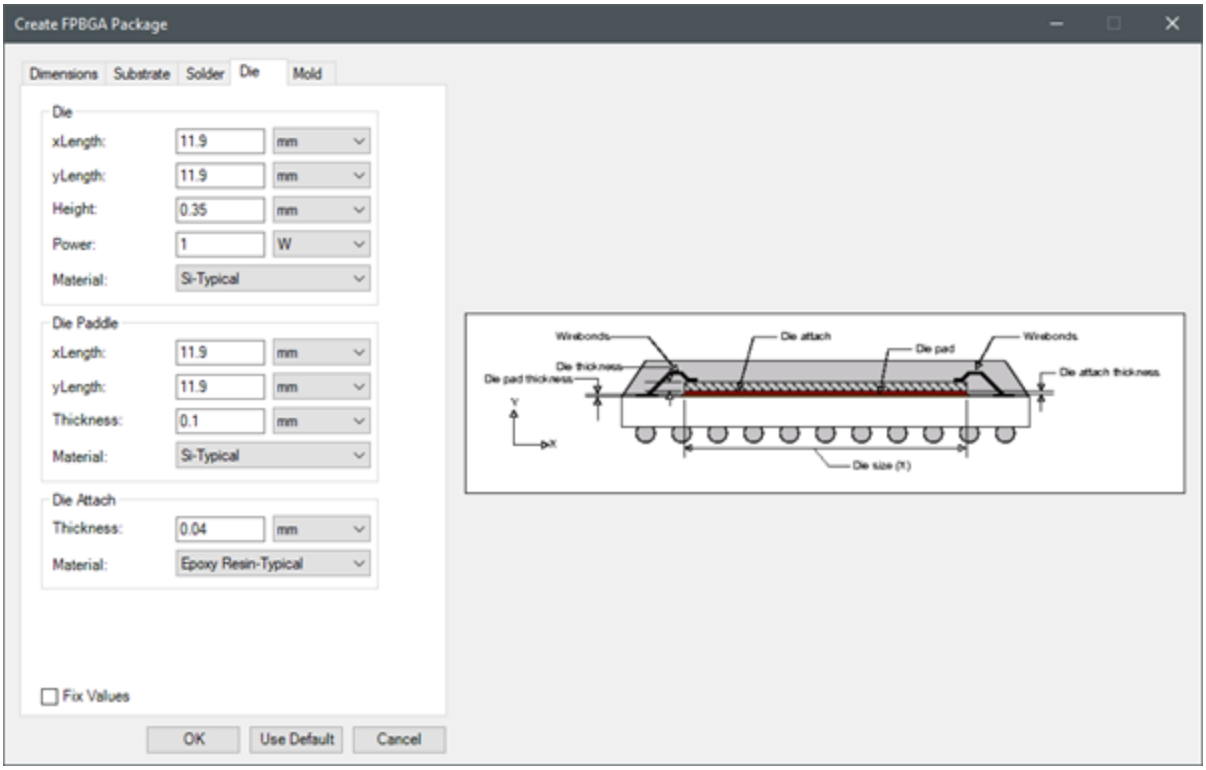
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



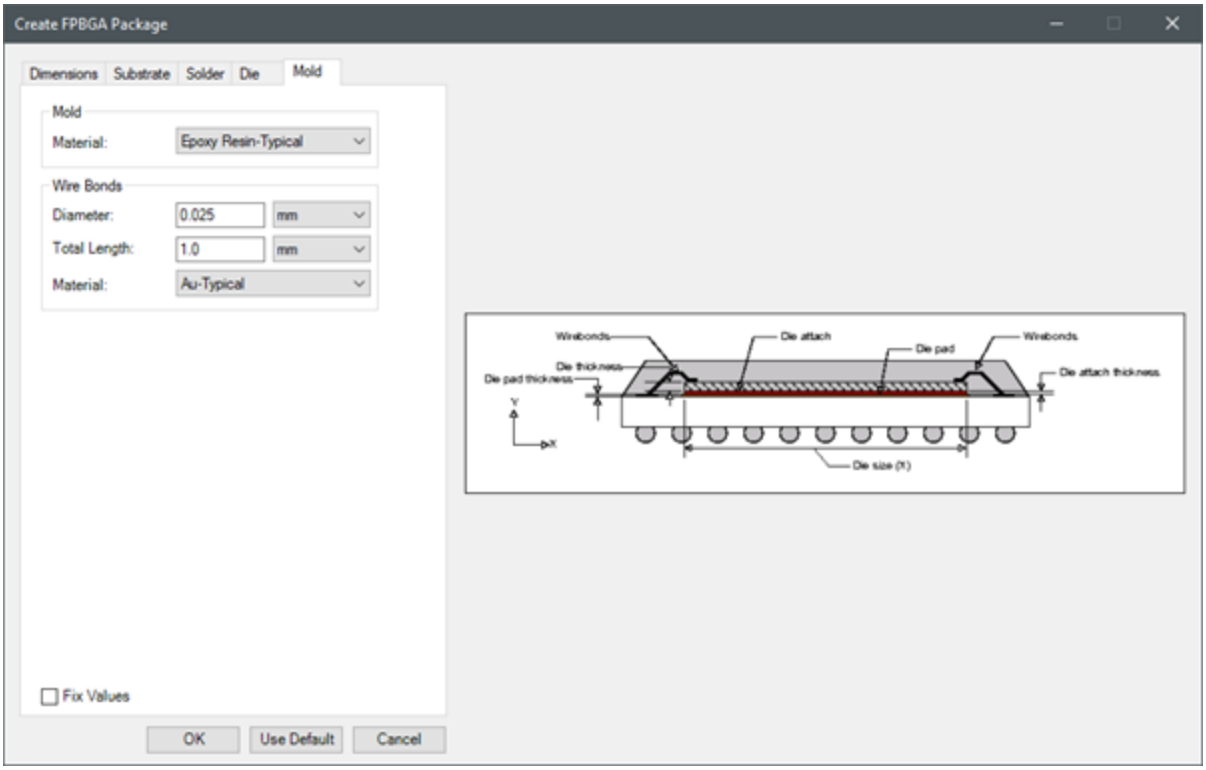
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed and number of central rows.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.
Mask	Under **Mask**, enter the solder ball mask thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

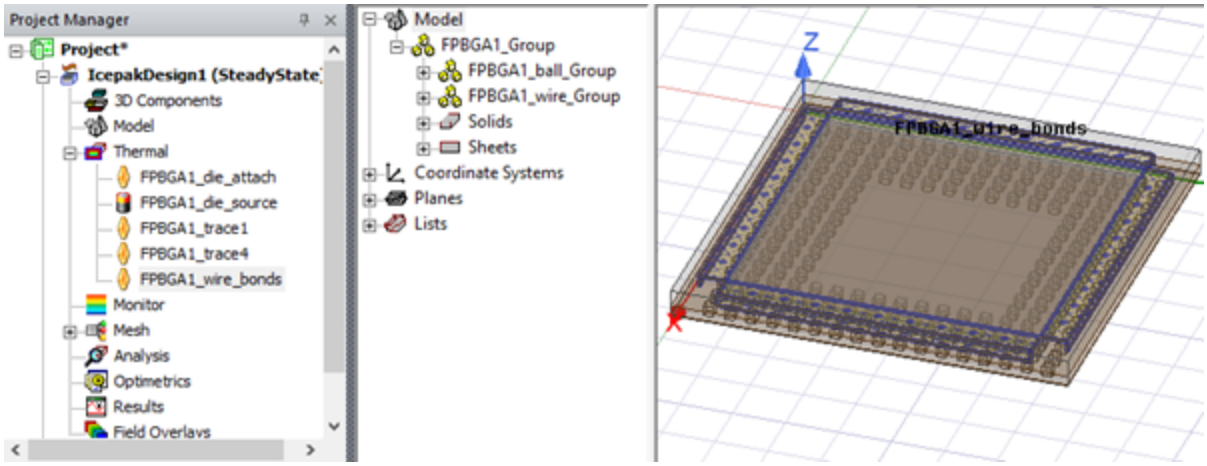


Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.



Iron Python Script – *FPBGA.py*:

The following is the toolkit's scripting information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.015',
        'Length2:0.015',
        'Package Thickness:0.0014',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Substrate Thickness:0.00036',
        'Substrate Material:Epoxy Resin-Typical',
        'Top Trace:25.0',
        'Bottom Trace:25.0',
        '1st Int. Layer Coverage:0.0',
        '2nd Int. Layer Coverage:0.0',
        'Trace Thickness:0.000033',
        'Trace Material:Cu-Pure',
        'Number of Thermal Vias:0',
        'Via Diameter:0.0002',
        'Via Plate Thickness:0.00005',
        'Number of Rows 1:17',
        'Number of Rows 2:17',
        'Array Type:Peripheral Array',
        'Suppressed Rows 1:9',
        'Suppressed Rows 2:9',
        'Central Rows 1:0',
        'Central Rows 2:0',
        'Ball Pitch:0.0008',
        'Ball Diameter:0.0004',
        'Ball Height:0.0004',
        'Ball Material:Solder-pb50_sn50',
        'Mask Thickness:0.00005',
        'Mask Material:Epoxy Resin-Typical',
        'Die Length1:0.0119',
        'Die Length2:0.0119',
        'Die Height:.00035',
        'Power:1',
        'Die Material:Si-Typical',
        'Pad Length1:0.0119',
        'Pad Length2:0.0119',
        'Pad Thickness:0.0001',
        'Pad Material:Si-Typical',
        'Attach Thickness:0.0004',
        'Attach Material:Epoxy Resin-Typical',
```

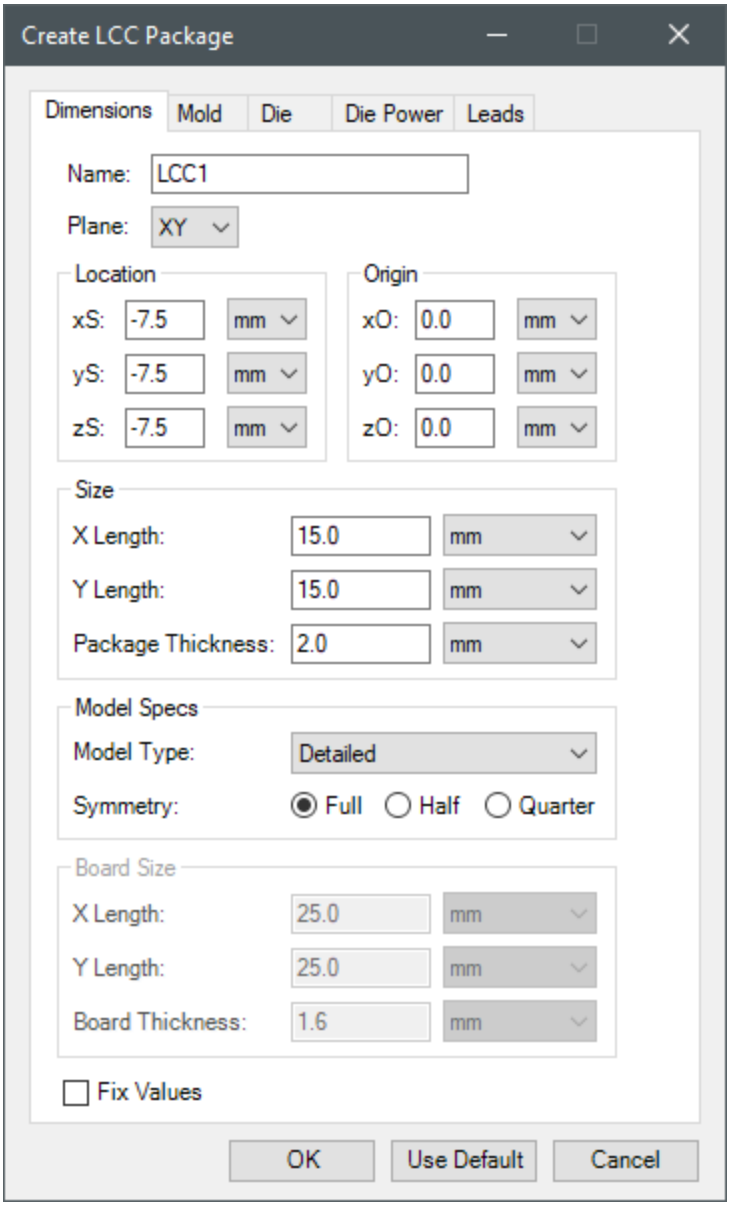

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'Mold Material:Epoxy Resin-Typical',
        'Wire Diameter:0.000025',
        'Wire Length:0.001',
        'Wire Material:Au-Typical',
        ]
oDesign.RunToolkit("SysLib", "Geometry/Packages/FPBGA", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/FPBGA", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Packages/FPBGA", data)
```

Packages - LCC

LCC enables you to create geometry and boundary conditions to represent an leadless chip carrier package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

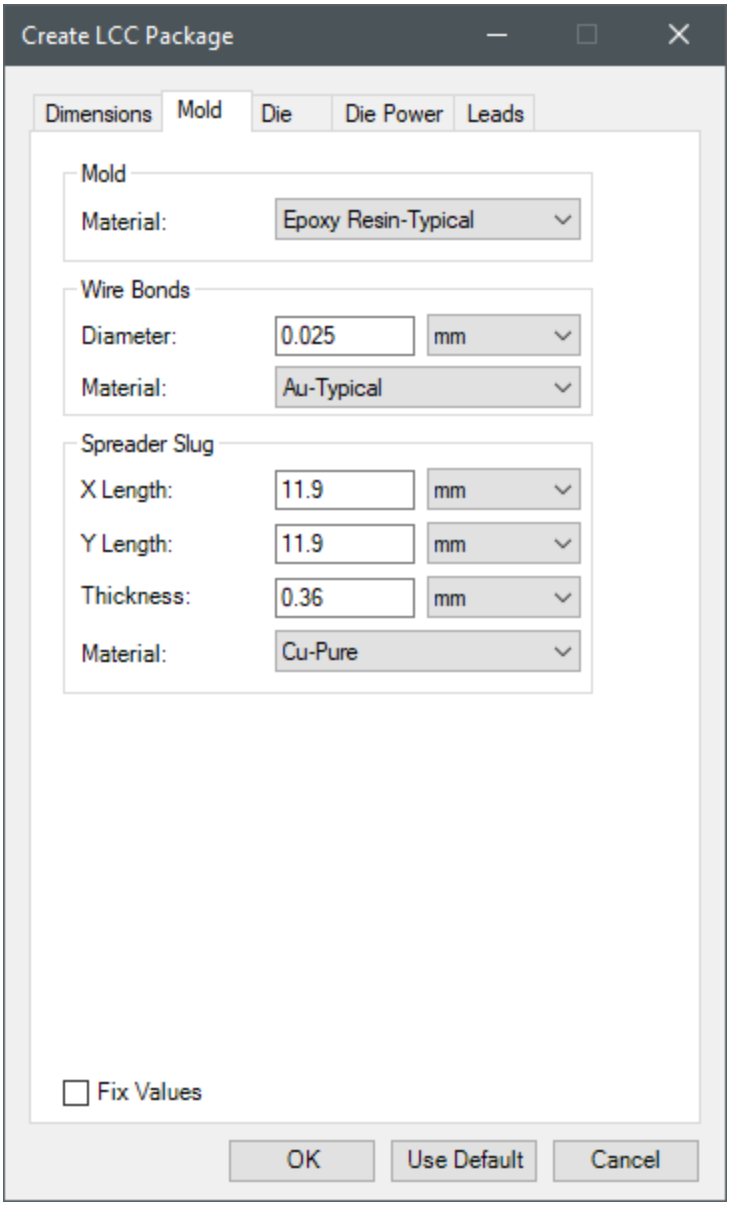


Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Origin	Under **Origin**, specify the coordinates to define the center from the **Location** of the package is offset. The default setting specifies the center of the **3D Modeler**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

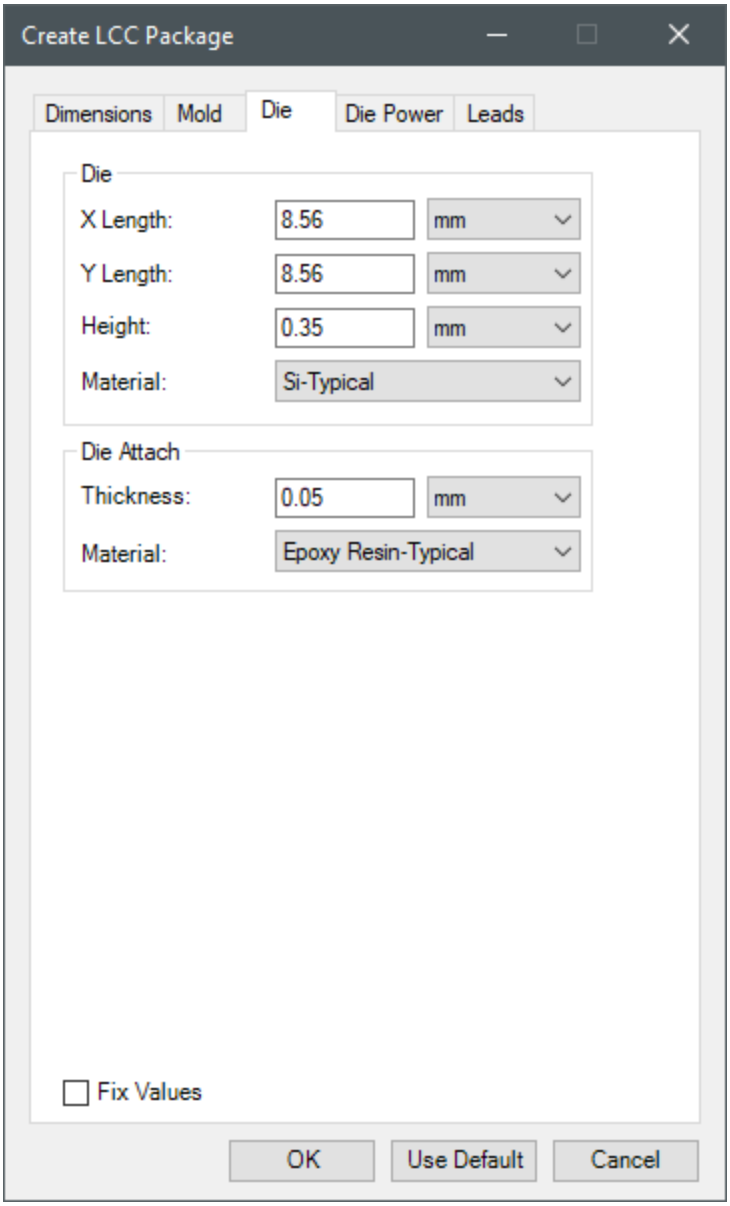
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.
Board Size	Under **Board Size**, when the **Model Type** is **Characterize JB**, specify the length of the board in each coordinate direction. The coordinate direction represented by **Board Thickness** depends on the selected **Plane**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



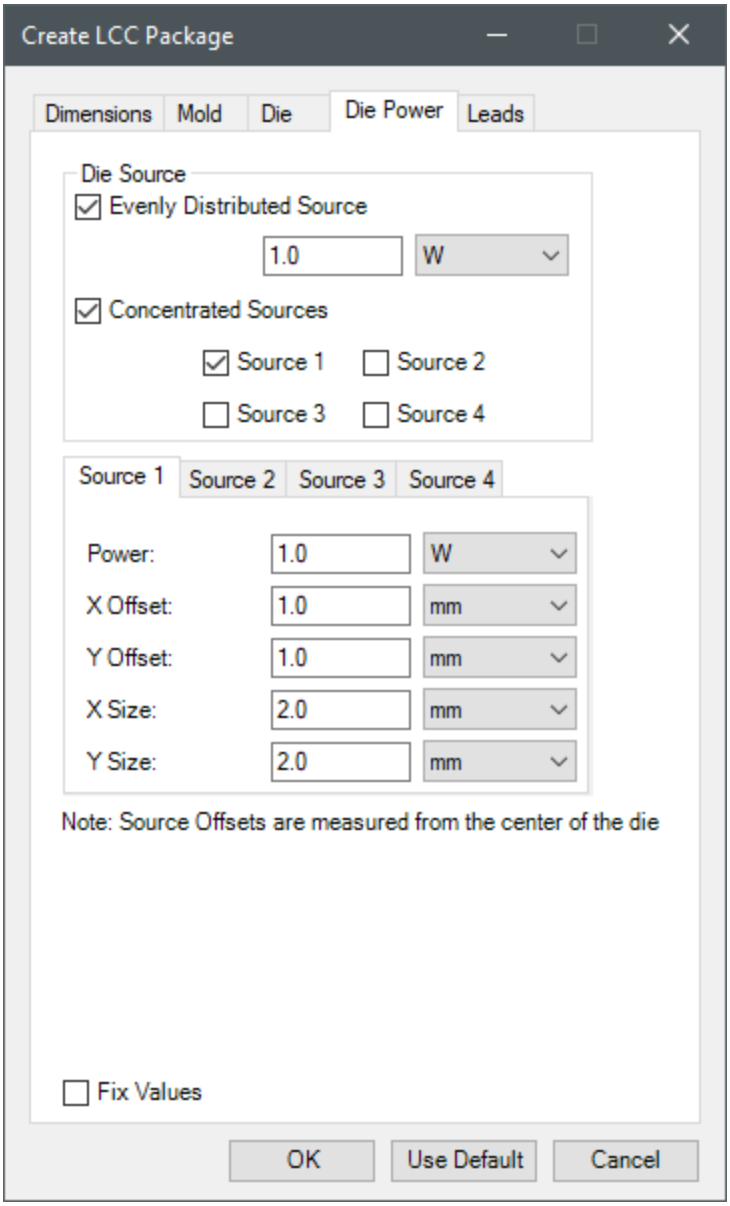
Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.
Spreader Slug	Under **Spreader Slug**, specify the length of the spreader slug in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



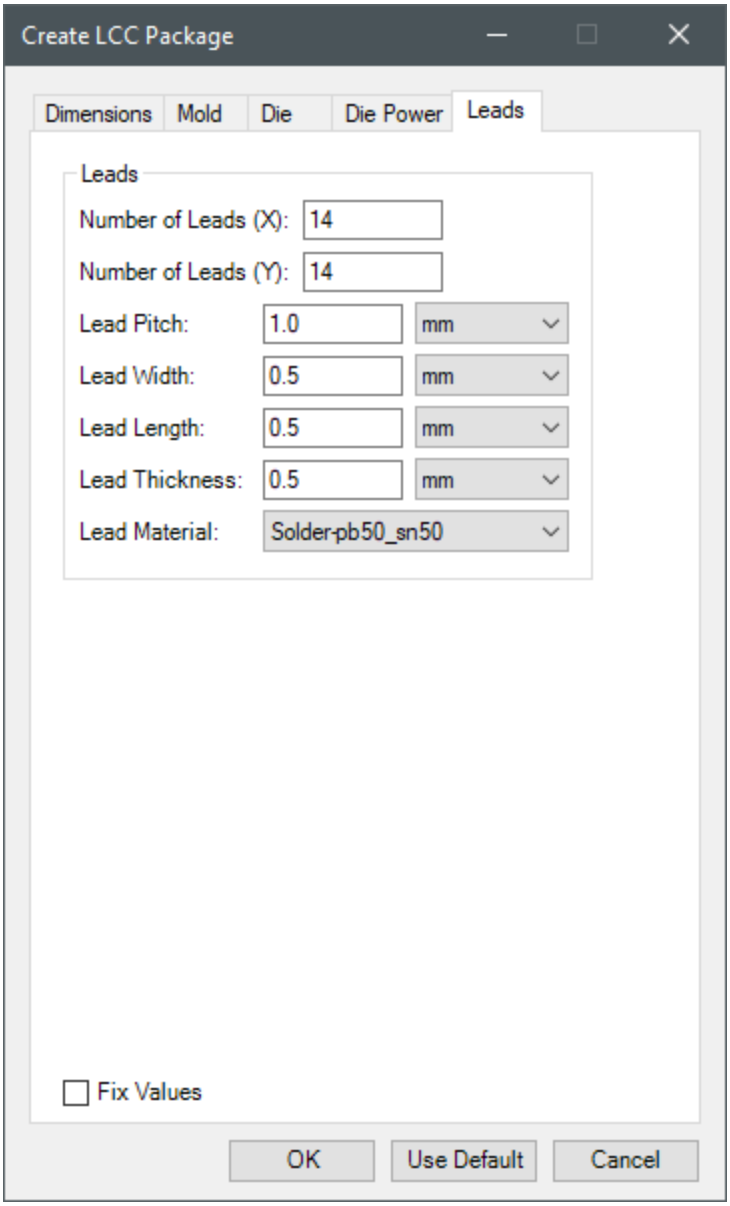
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



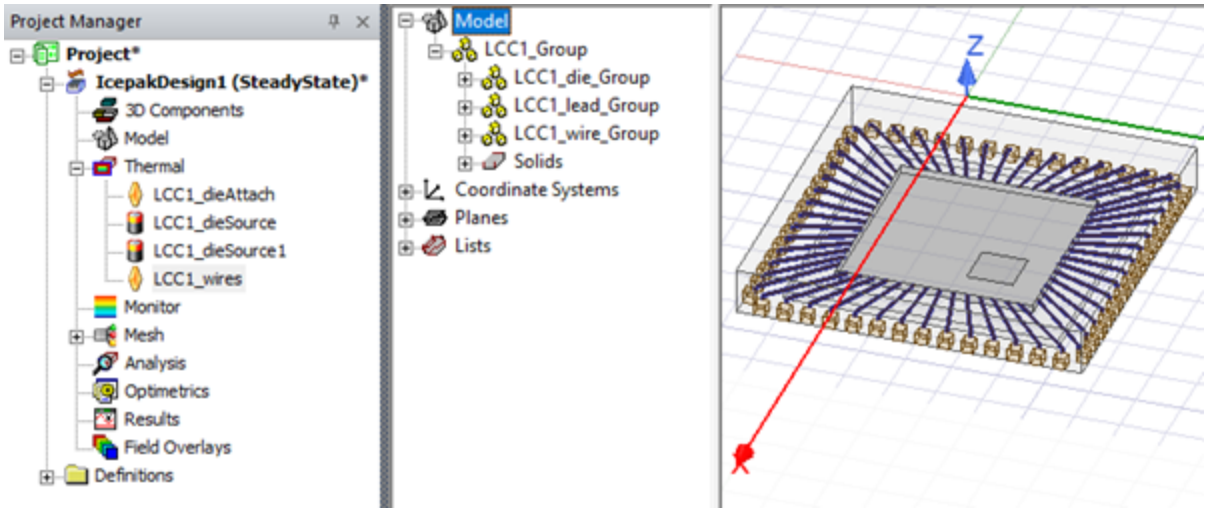
Die Source	Under **Die Source**, specify if the package has evenly distributed die sources and/or concentrated. Also, if **Evenly Distributed Source** is selected, specify the source power. If **Concentrated Sources** is selected, select how many sources to include in the package.
Source(s)	If **Concentrated Sources** is selected, on the **Source** tab(s), specify the source power, the offsets (from the center of the die), and the lengths of the sides.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads along the axes and the lead pitch, width, length, thickness, and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *LCC.py*:

The following is the toolkit's scripting information.

```
scriptInput = ["ng"]
scriptInput.append("Name:LCC1")
scriptInput.append("Plane:YZ")
scriptInput.append("length_unit:meter")
scriptInput.append("power_unit:W")
scriptInput.append("xS:-0.0075")
scriptInput.append("yS:-0.0075")
scriptInput.append("zS:-0.0075")
scriptInput.append("xO:0.0")
scriptInput.append("yO:0.0")
scriptInput.append("zO:0.0")
scriptInput.append("Len1:0.015")
scriptInput.append("Len2:0.015")
scriptInput.append("Thick:0.0016")
scriptInput.append("Type:Detailed")
scriptInput.append("boardLen1:0.0252")
scriptInput.append("boardLen2:0.0252")
scriptInput.append("boardThick:0.0016")
scriptInput.append("Symmetry:Full")
scriptInput.append("MoldMaterial:Epoxy Resin-Typical")
scriptInput.append("SlugMaterial:Cu-Pure")
scriptInput.append("WireMaterial:Au-Typical")
scriptInput.append("WireDiameter:0.000025")
scriptInput.append("slugLen1:0.0119")
scriptInput.append("slugLen2:0.0119")
scriptInput.append("slugThick:0.00036")
```

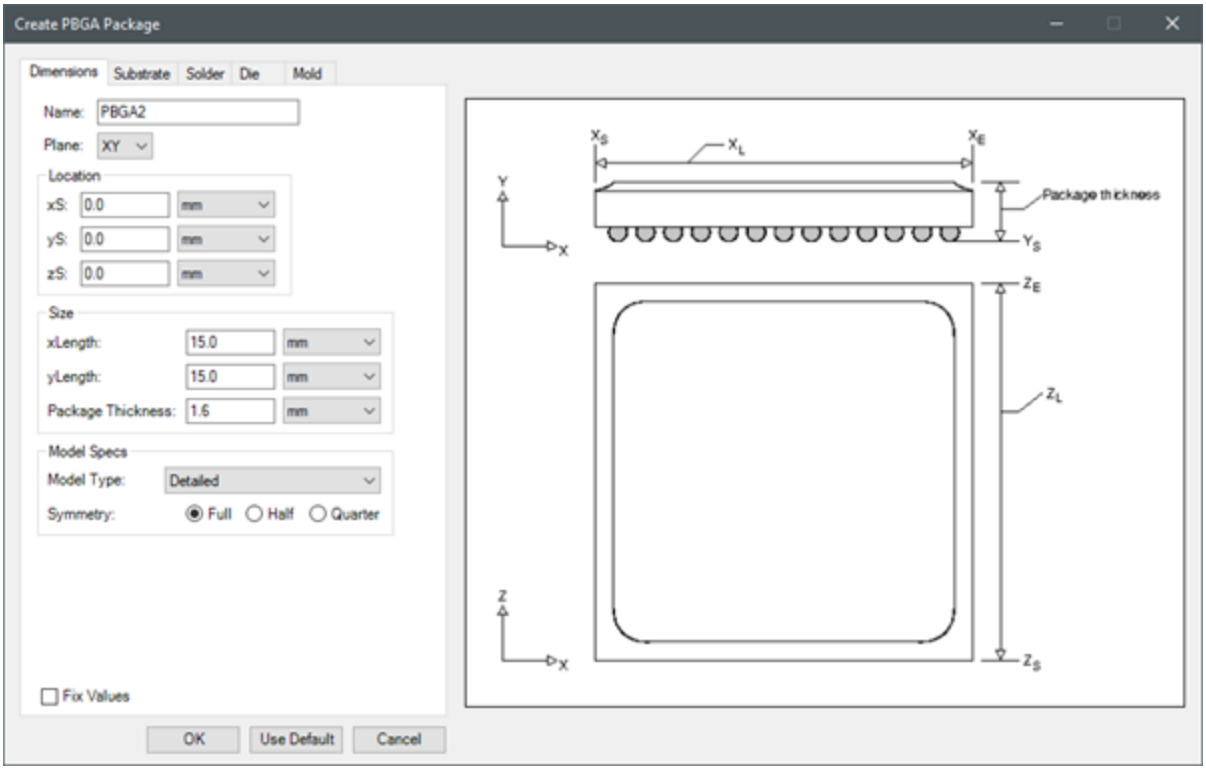

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 scriptInput.append("dieLen1:0.00856")
 scriptInput.append("dieLen2:0.00856")
 scriptInput.append("dieHeight:0.00035")
 scriptInput.append("dieMat:Si-Typical")
 scriptInput.append("attachThick:0.00005")
 scriptInput.append("attachMat:Epoxy Resin-Typical")
 scriptInput.append("DistributedSource:None")
 scriptInput.append("SourcePower:1.0")
 scriptInput.append("ConcentratedSource:Checked")
 scriptInput.append("Source1:Checked")
 scriptInput.append("Source2:None")
 scriptInput.append("Source3:Checked")
 scriptInput.append("Source4:None")
 # enter source info: sourcePower,offset1,offset2,size1,size2
 scriptInput.append("SourcePower1:1.0,0.001,0.001,0.002,0.002")
 scriptInput.append("SourcePower2:1.0,0.001,-0.003,0.002,0.002")
 scriptInput.append("SourcePower3:1.0,-0.003,0.001,0.002,0.002")
 scriptInput.append("SourcePower4:1.0,-0.003,-
0.003,0.002,0.002")
 scriptInput.append("LeadCount1:10")
 scriptInput.append("LeadCount2:7")
 scriptInput.append("LeadPitch:0.001")
 scriptInput.append("LeadWidth:0.0005")
 scriptInput.append("LeadLength:0.0005")
 scriptInput.append("LeadThick:0.0005")
 scriptInput.append("LeadMaterial:Solder-pb50_sn50")
 oDesign.RunToolkit("SysLib", "Geometry/Packages/LCC", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages/LCC", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages/LCC",
scriptInput)
```

Packages - PBGA

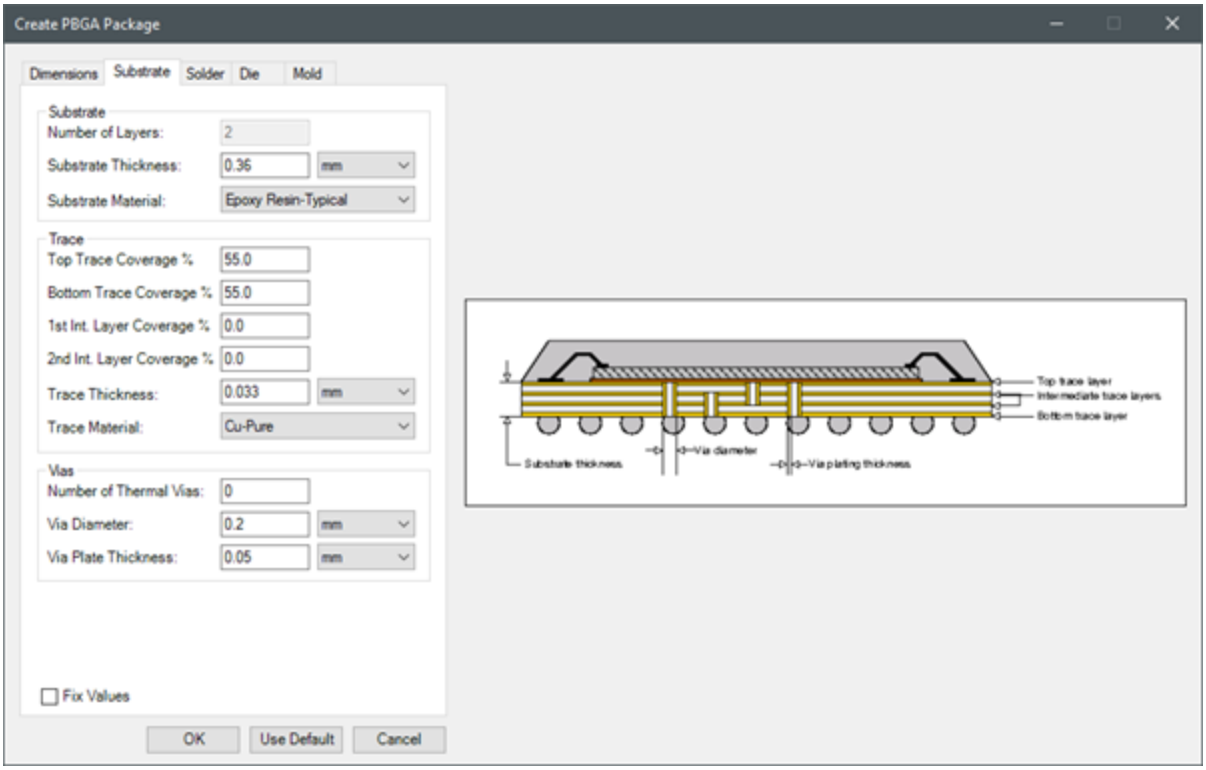
PBGA enables you to create geometry and boundary conditions to represent an plastic ball grid array package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



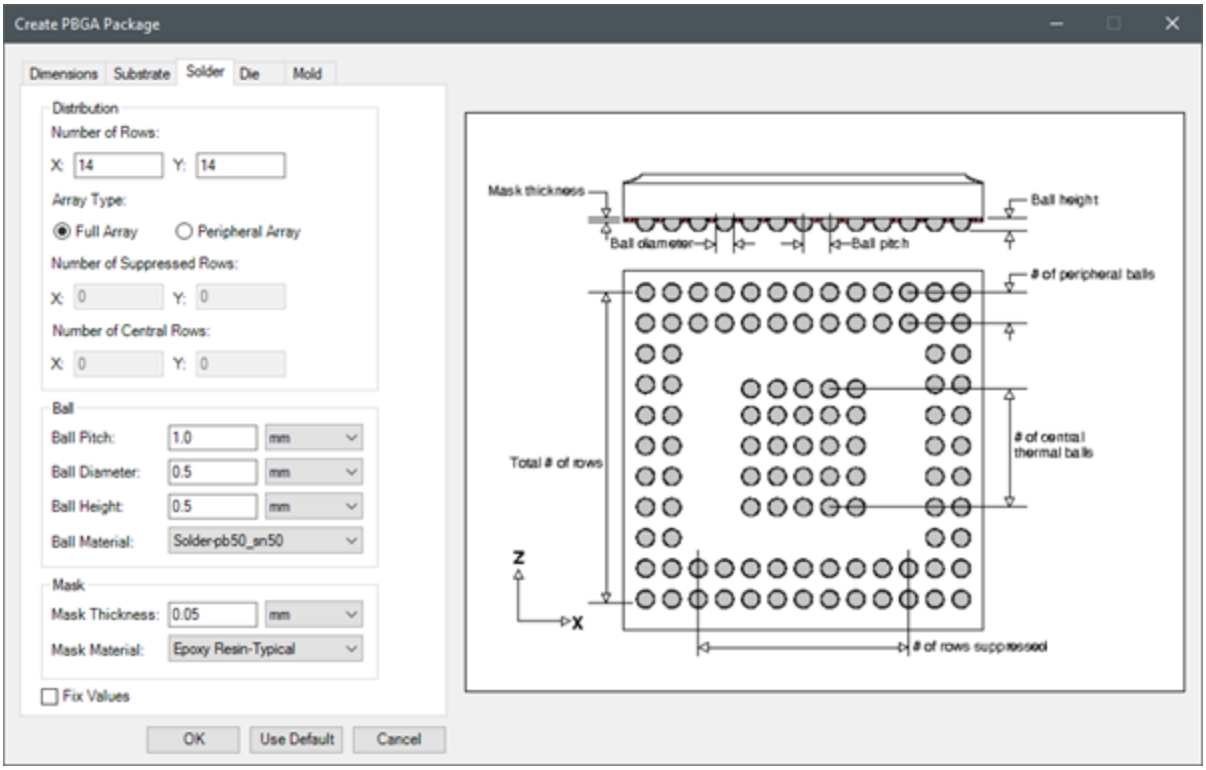
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



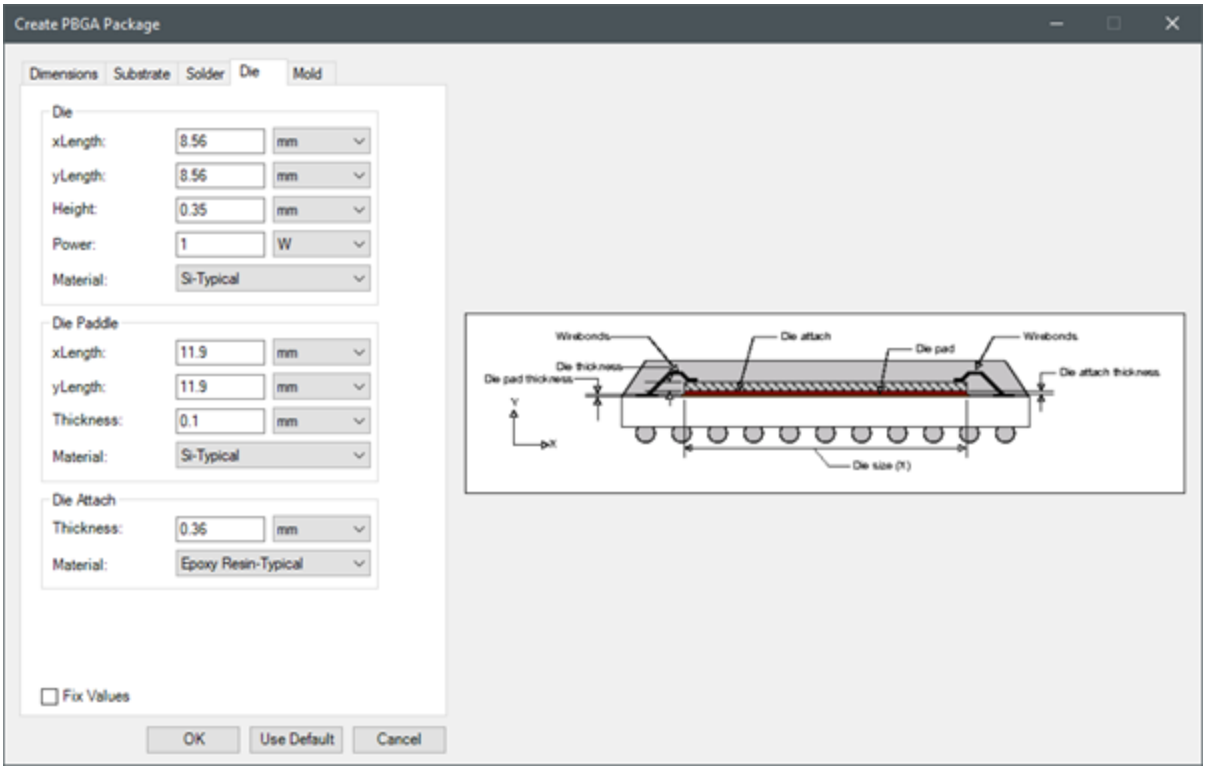
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



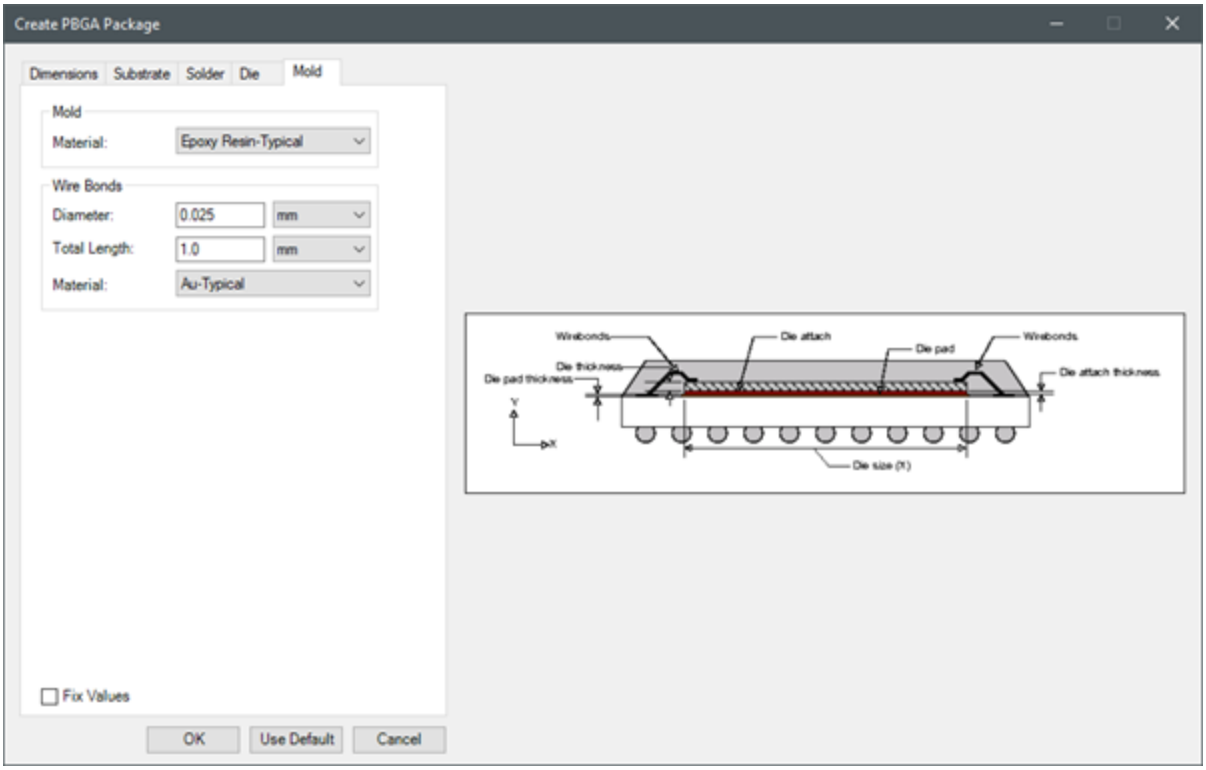
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed and number of central rows.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.
Mask	Under **Mask**, enter the solder ball mask thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

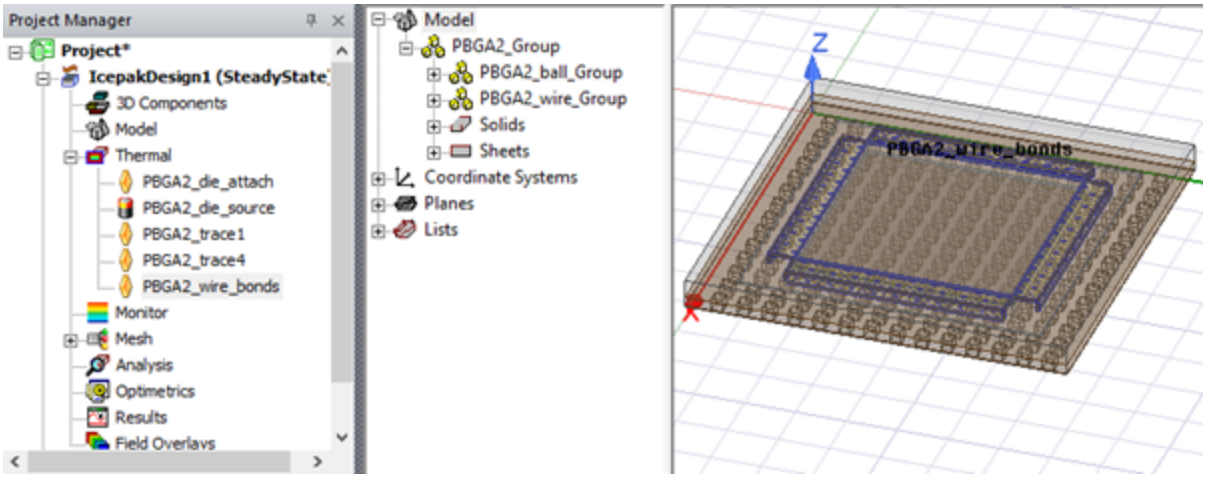


Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.



Iron Python Script – *PBGA.py*:

The following is the toolkit's scripting information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

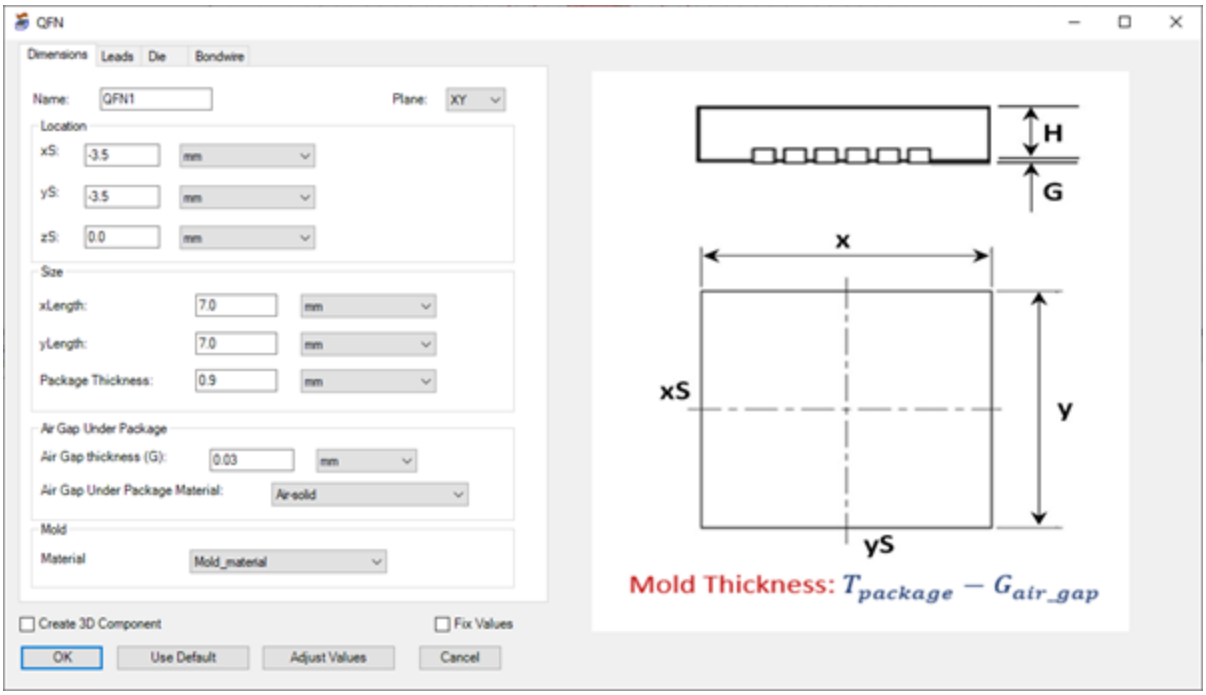
```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.015',
        'Length2:0.015',
        'Package Thickness:0.0016',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Substrate Thickness:0.00036',
        'Substrate Material:Epoxy Resin-Typical',
        'Top Trace:55.0',
        'Bottom Trace:55.0',
        '1st Int. Layer Coverage:0.0',
        '2nd Int. Layer Coverage:0.0',
        'Trace Thickness:0.000033',
        'Trace Material:Cu-Pure',
        'Number of Thermal Vias:0',
        'Via Diameter:0.0002',
        'Via Plate Thickness:0.00005',
        'Number of Rows 1:14',
        'Number of Rows 2:14',
        'Array Type:Full Array',
        'Suppressed Rows 1:0',
        'Suppressed Rows 2:0',
        'Central Rows 1:0',
        'Central Rows 2:0',
        'Ball Pitch:0.001',
        'Ball Diameter:0.0005',
        'Ball Height:0.0005',
        'Ball Material:Solder-pb50_sn50',
        'Mask Thickness:0.00005',
        'Mask Material:Epoxy Resin-Typical',
        'Die Length1:0.00856',
        'Die Length2:0.00856',
        'Die Height:.00035',
        'Power:1',
        'Die Material:Si-Typical',
        'Pad Length1:0.0119',
        'Pad Length2:0.0119',
        'Pad Thickness:0.0001',
        'Pad Material:Si-Typical',
        'Attach Thickness:0.00036',
        'Attach Material:Epoxy Resin-Typical',
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'Mold Material:Epoxy Resin-Typical',
        'Wire Diameter:0.000025',
        'Wire Length:0.001',
        'Wire Material:Au-Typical'
        ]
oDesign.RunToolkit("SysLib", "Geometry/Packages/PBGA", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/PBGA", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Packages/PBGA", data)
```

Packages - QFN

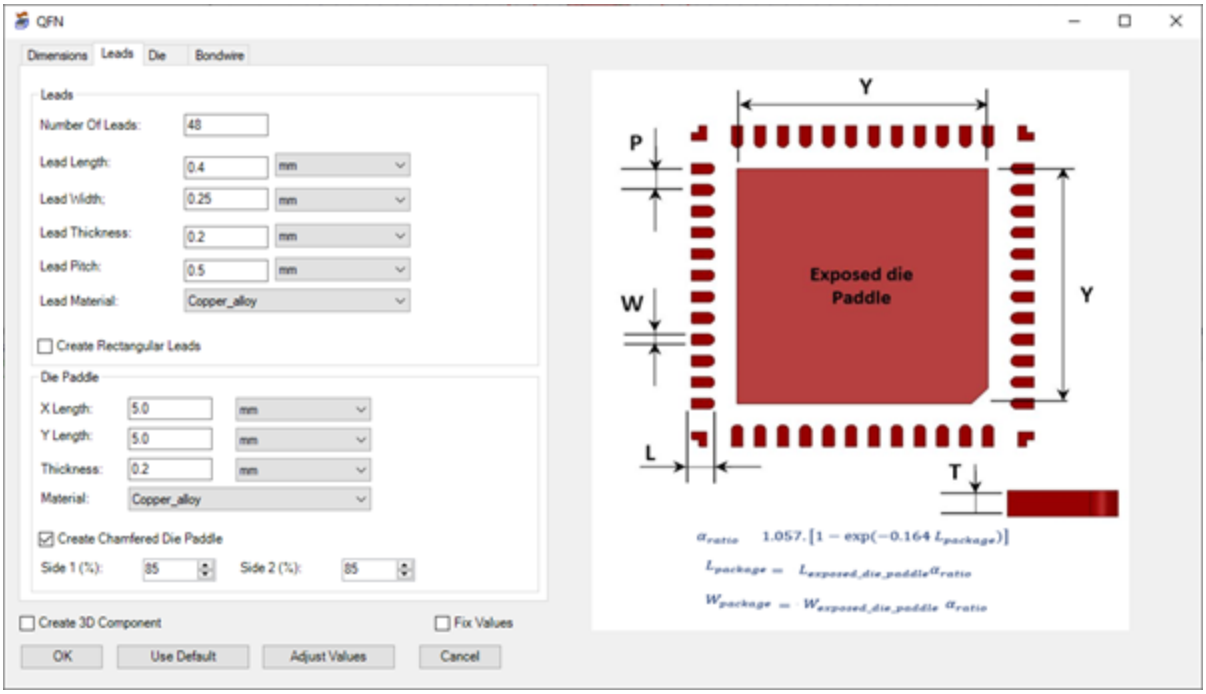
QFN enables you to create geometry and boundary conditions to represent a quad flat no-lead package.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Air Gap Under	Under **Air Gap Under Package**, specify the **Air Gap thickness** and

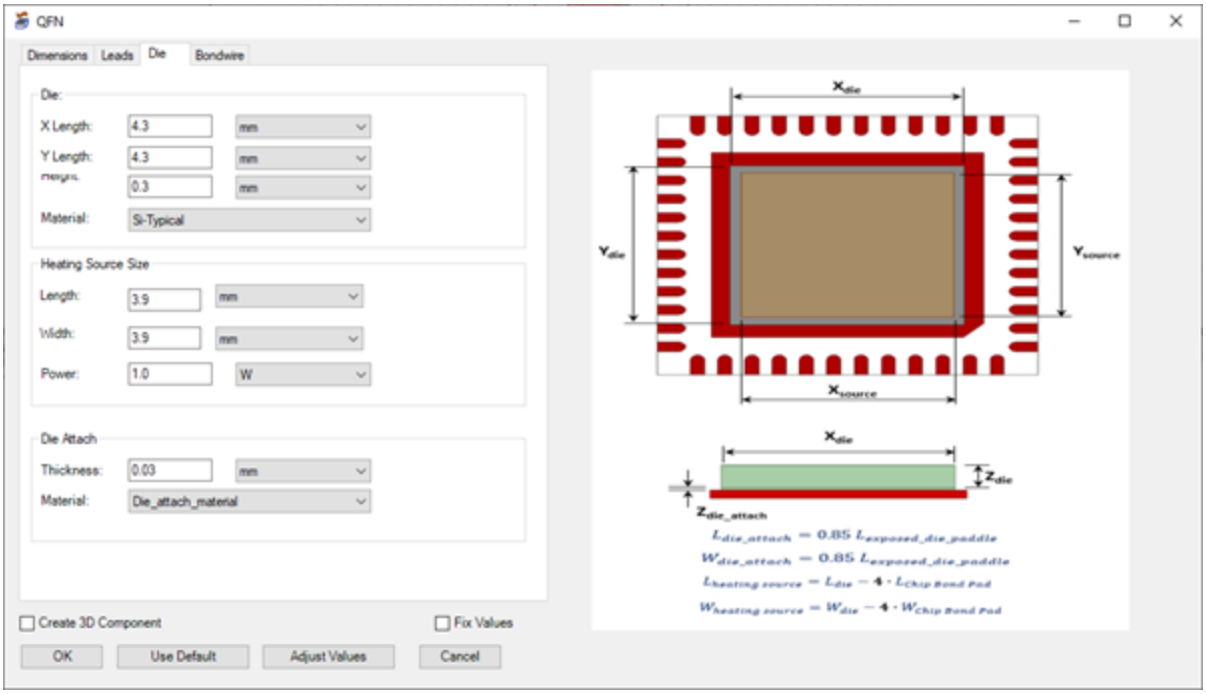
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Package	the **Air Gap Under Package Material**. The thickness of the mold is calculated by subtracting the **Air Gap thickness** from the **Package Thickness**.
Mold	Under **Mold**, specify the **Material** for the package mold.
OK	Click **OK** to run the toolkit.
Use Default	Click **Use Default** to restore the default settings.
Adjust Values	If the package length or number of leads is modified, click **Adjust Values** to automatically adjust the other QFN component size values.
Cancel	Click **Cancel** to close the QFN dialog box without running the toolkit.



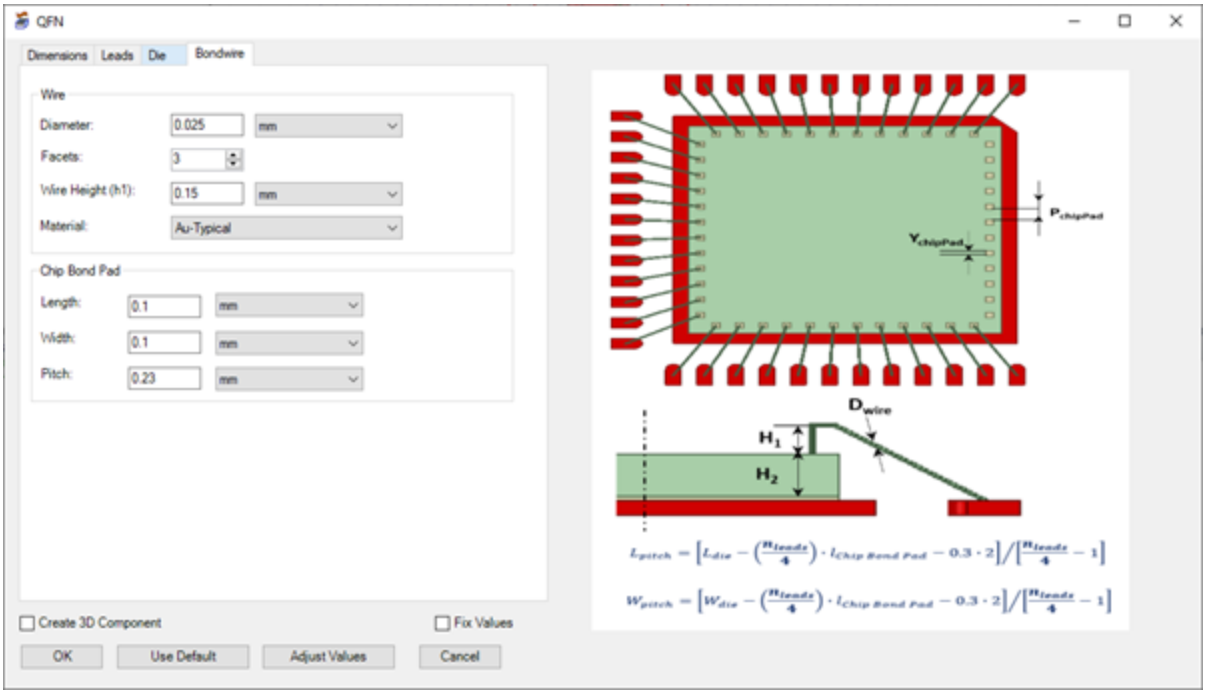
Leads	Under **Leads**, specify the number of leads. Also specify the lead length, width, thickness, pitch, and material. If needed, enable **Create Rectangular Leads** to specify the shape of the leads.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material. If needed, enable **Create Chamfered Die Paddle** and specify **Side 1 (%)** and **Side 2 (%)** to specify the shape of the die.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

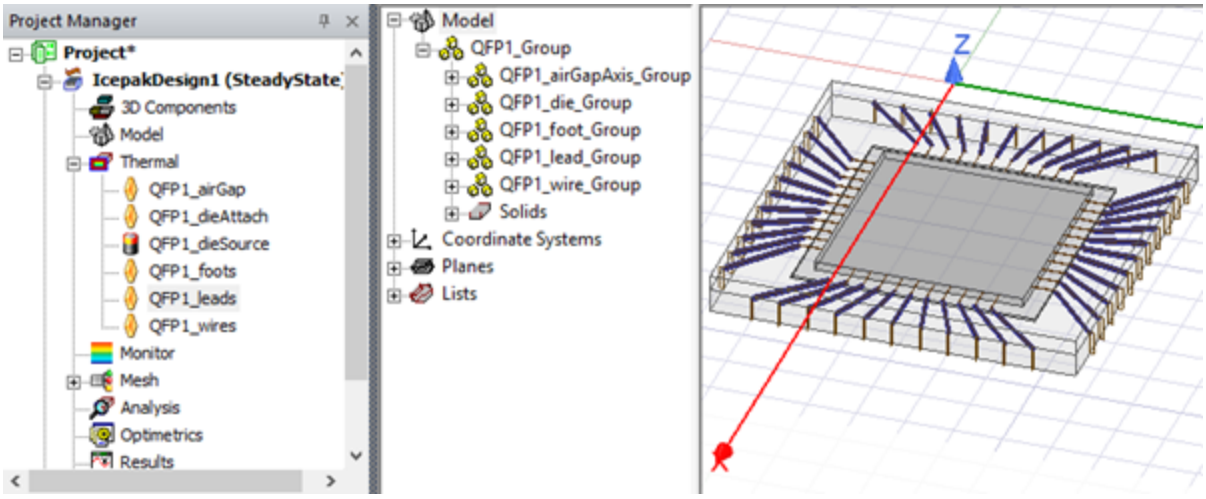


Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die material.
Heating Source Size	Under **Heating Source Size**, specify the length and width of the die. Also, specify the power of the heating source.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Wire	Under **Wire**, specify the wire diameter, number of facets, height, and material.
Chip Bond Pad	Under **Chip Bond Pad**, specify the length, width, and pitch of the chip bond pad.



Iron Python Script – *QFN.py*:

The following is the toolkit's scripting information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

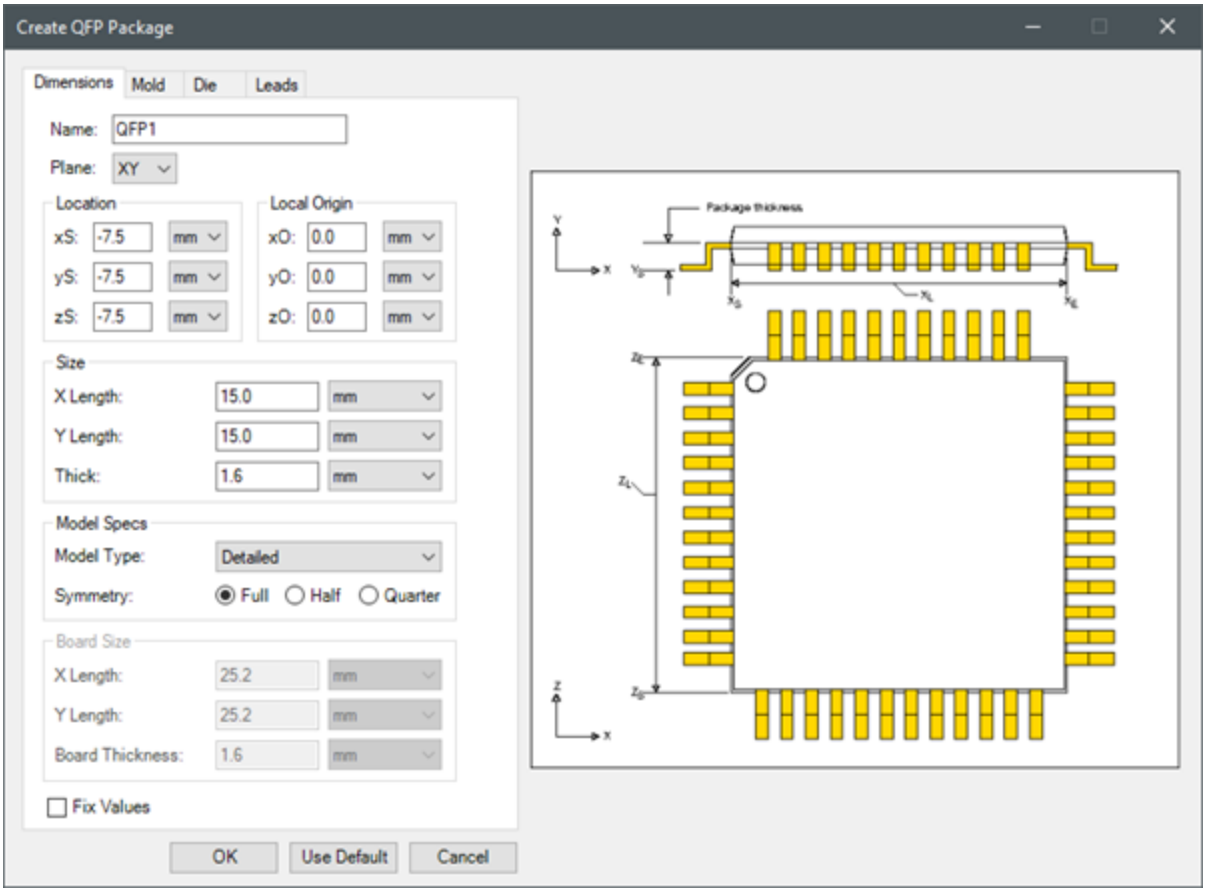
```
        scriptInput = ["ng"]
        scriptInput.append("Name:QFN1")
        scriptInput.append("Plane:XY")
        scriptInput.append("length_Unit:meter")
        scriptInput.append("power_unit:W")
        scriptInput.append("xS:0.0075")
        scriptInput.append("yS:0.0075")
        scriptInput.append("zS:0.0")
        scriptInput.append("xLength:0.015")
        scriptInput.append("yLength:0.015")
        scriptInput.append("packThickness:0.0016")
        scriptInput.append("WBDiameter:0.00005")
        scriptInput.append("Facets:3")
        scriptInput.append("WBHeight:0.00015")
        scriptInput.append("CBPLength:0.0005")
        scriptInput.append("CBPWidth:0.0005")
        scriptInput.append("CBPPitch:0.0007")
        scriptInput.append("HSSLength:0.3")
        scriptInput.append("HSSWidth:0.3")
        scriptInput.append("DieXLength:0.00856")
        scriptInput.append("DieYLength:0.00856")
        scriptInput.append("DieHeight:0.00035")
        scriptInput.append("DiePower:1.0")
        scriptInput.append("DiePXLength:0.01")
        scriptInput.append("DiePYLength:0.01")
        scriptInput.append("DiePisChamfered:True")
        scriptInput.append("Chamferside1:85")
        scriptInput.append("Chamferside2:85")
        scriptInput.append("DieThickness:0.0001")
        scriptInput.append("DieAThickness:0.0005")
        scriptInput.append("leadCount:40")
        scriptInput.append("LeadLength:0.0005")
        scriptInput.append("LeadWidth:0.0005")
        scriptInput.append("LeadThick:0.0005")
        scriptInput.append("LeadPitch:0.001")
        scriptInput.append("LeadisRectangle:False")
        scriptInput.append("AGUP:0.00001")
        scriptInput.append("AGUPMAterial:Air-solid")
        scriptInput.append("MoldMaterial:Mold_material")
        scriptInput.append("WBMaterial:Au-Typical")
        scriptInput.append("DieMaterial:Si-Typical")
        scriptInput.append("DiePMaterial:Cu-Brass")
        scriptInput.append("DieAMaterial:Die_attach_material")
        scriptInput.append("LeadMaterial:Cu-Brass")
        oDesign.RunToolkit("SysLib", "Geometry/Packages/QFN", [])
        oDesign.RunToolkit("SysLib", "Geometry/Packages/QFN", ["ng"])
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        oDesign.RunToolkit("SysLib", "Geometry/Packages/QFN",
scriptInput)
```

Packages - QFP

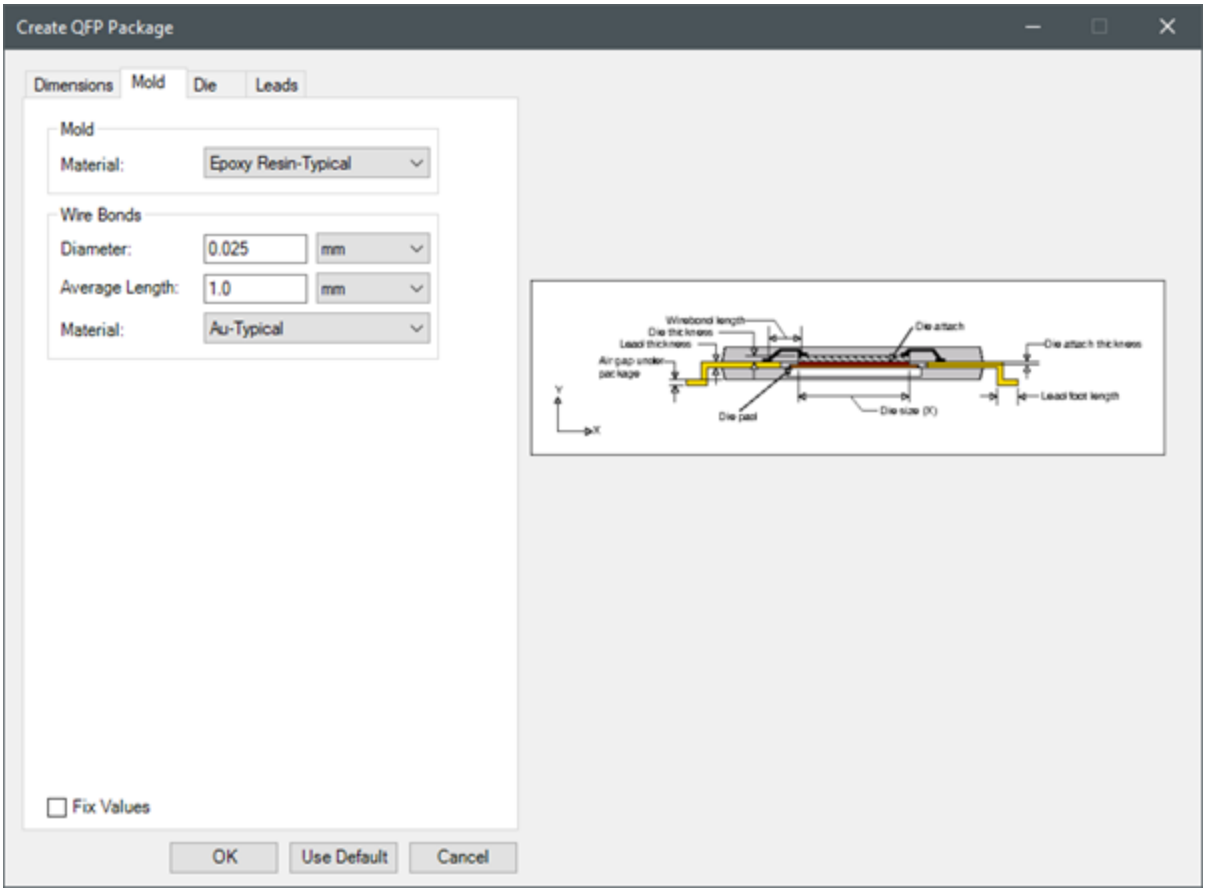
QFP enables you to create geometry and boundary conditions to represent a quad flat package.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Local Origin	Under **Local Origin**, specify the coordinates to define the center from the **Location** of the package is offset. The default setting specifies the center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Thick** depends on

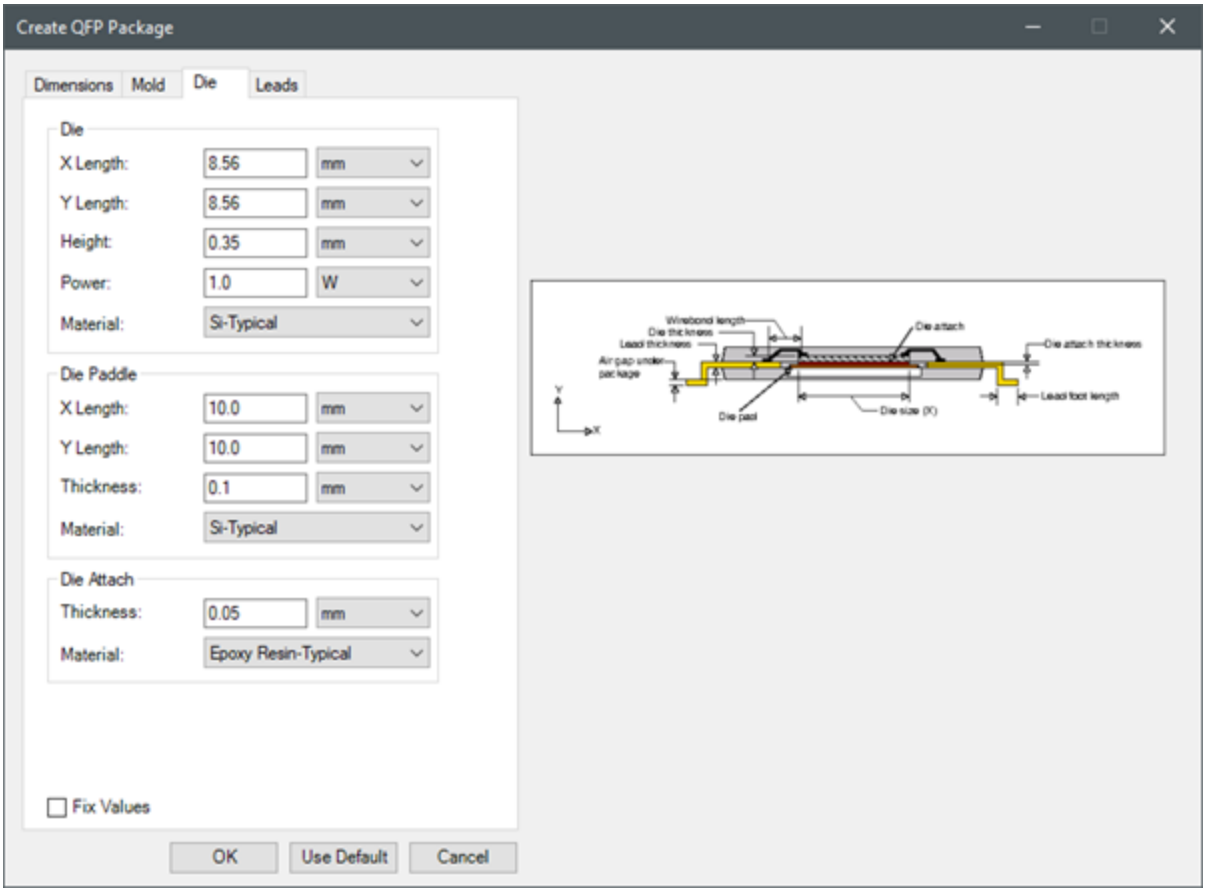
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.
Board Size	Under **Board Size**, when the **Model Type** is **Characterize JB**, specify the length of the board in each coordinate direction. The coordinate direction represented by **Board Thickness** depends on the selected **Plane**.



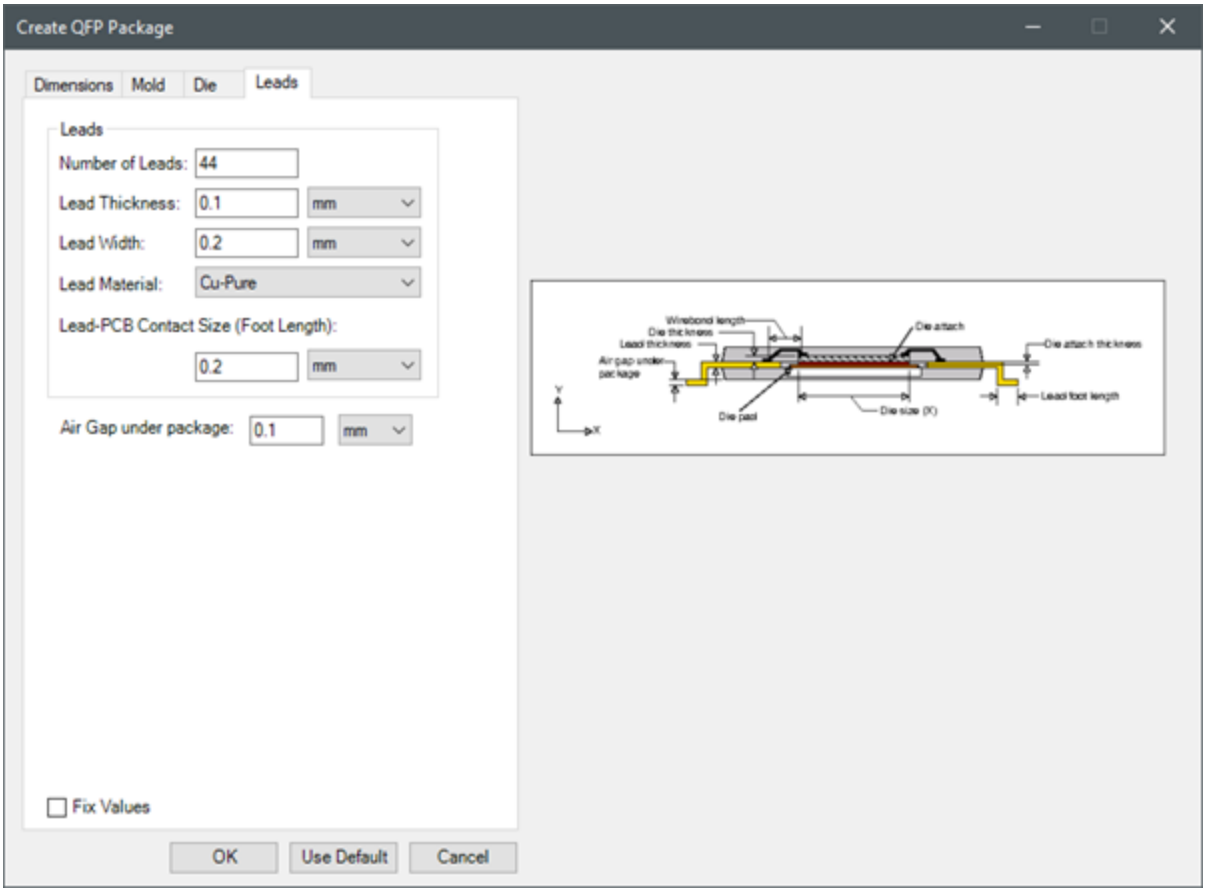
Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

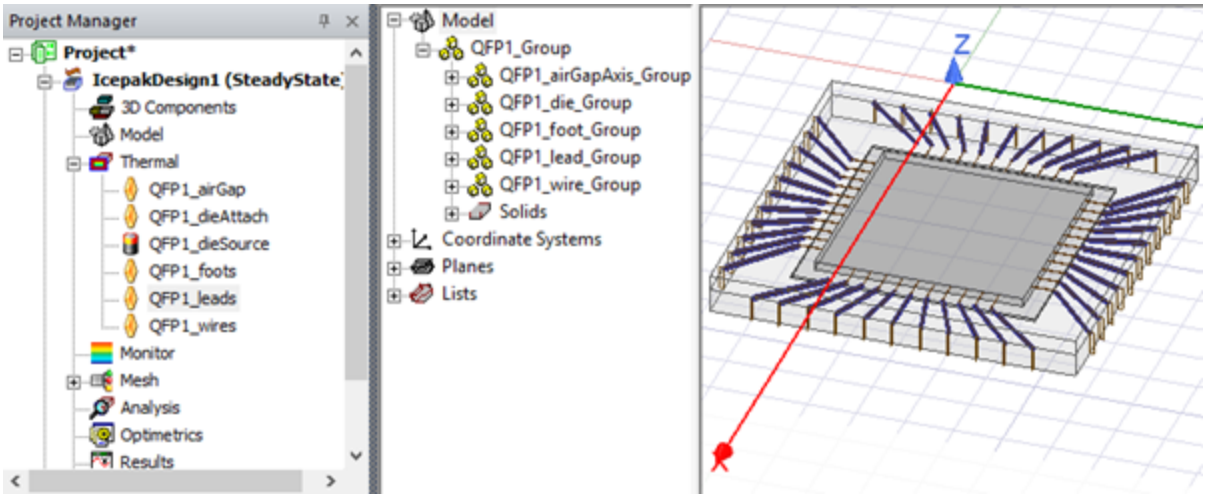


Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads, the lead thickness and width, material, foot length, and airgap under the package.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iron Python Script – *QFP.py*:

The following is the toolkit's scripting information.

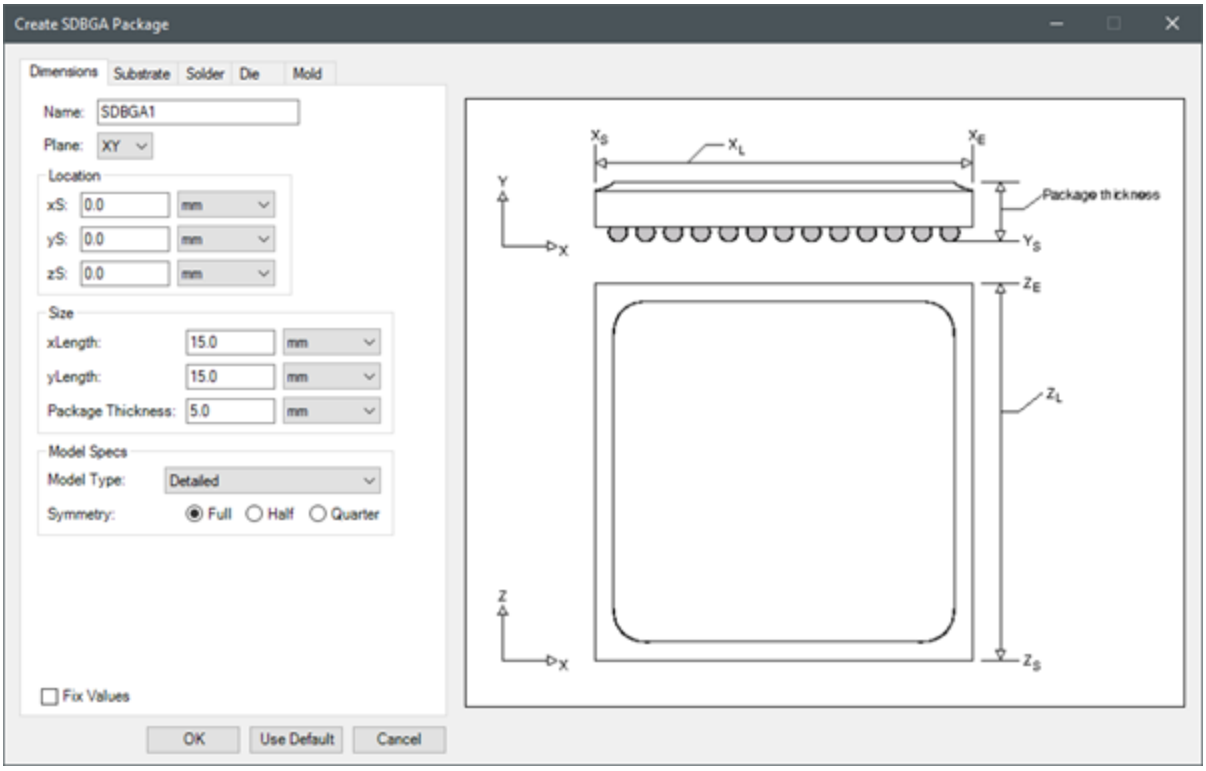
```
scriptInput = ["ng"]
scriptInput.append("Name:QFP1")
scriptInput.append("Plane:XY")
scriptInput.append("length_unit:meter")
scriptInput.append("power_unit:W")
scriptInput.append("xS:-0.0075")
scriptInput.append("yS:-0.0075")
scriptInput.append("zS:-0.0075")
scriptInput.append("xO:0.0")
scriptInput.append("yO:0.0")
scriptInput.append("zO:0.0")
scriptInput.append("Len1:0.015")
scriptInput.append("Len2:0.015")
scriptInput.append("Thick:0.0016")
scriptInput.append("Type:Detailed")
scriptInput.append("boardLen1:0.0252")
scriptInput.append("boardLen2:0.0252")
scriptInput.append("boardThick:0.0016")
scriptInput.append("Symmetry:Full")
scriptInput.append("dieLen1:0.00856")
scriptInput.append("dieLen2:0.00856")
scriptInput.append("dieHeight:0.00035")
scriptInput.append("diePow:1.0")
scriptInput.append("padLen1:0.01")
scriptInput.append("padLen2:0.01")
scriptInput.append("padThick:0.0001")
scriptInput.append("attachThick:0.00005")
scriptInput.append("WireDiameter:0.000025")
scriptInput.append("WireLength:0.001")
scriptInput.append("NumberLeads:12")
scriptInput.append("LeadThick:0.0001")
scriptInput.append("LeadWidth:0.0002")
scriptInput.append("LeadFootLength:0.0002")
scriptInput.append("AirGap:0.0001")
scriptInput.append("attachMat:Epoxy Resin-Typical")
scriptInput.append("moldMat:Epoxy Resin-Typical")
scriptInput.append("dieMat:Si-Typical")
scriptInput.append("padMat:Si-Typical")
scriptInput.append("wireMat:Au-Typical")
scriptInput.append("leadMat:Cu-Pure")
oDesign.RunToolkit("SysLib", "Geometry/Packages/QFP", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/QFP", ["ng"])
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign.RunToolkit("SysLib", "Geometry/Packages/QFP",
scriptInput)
```

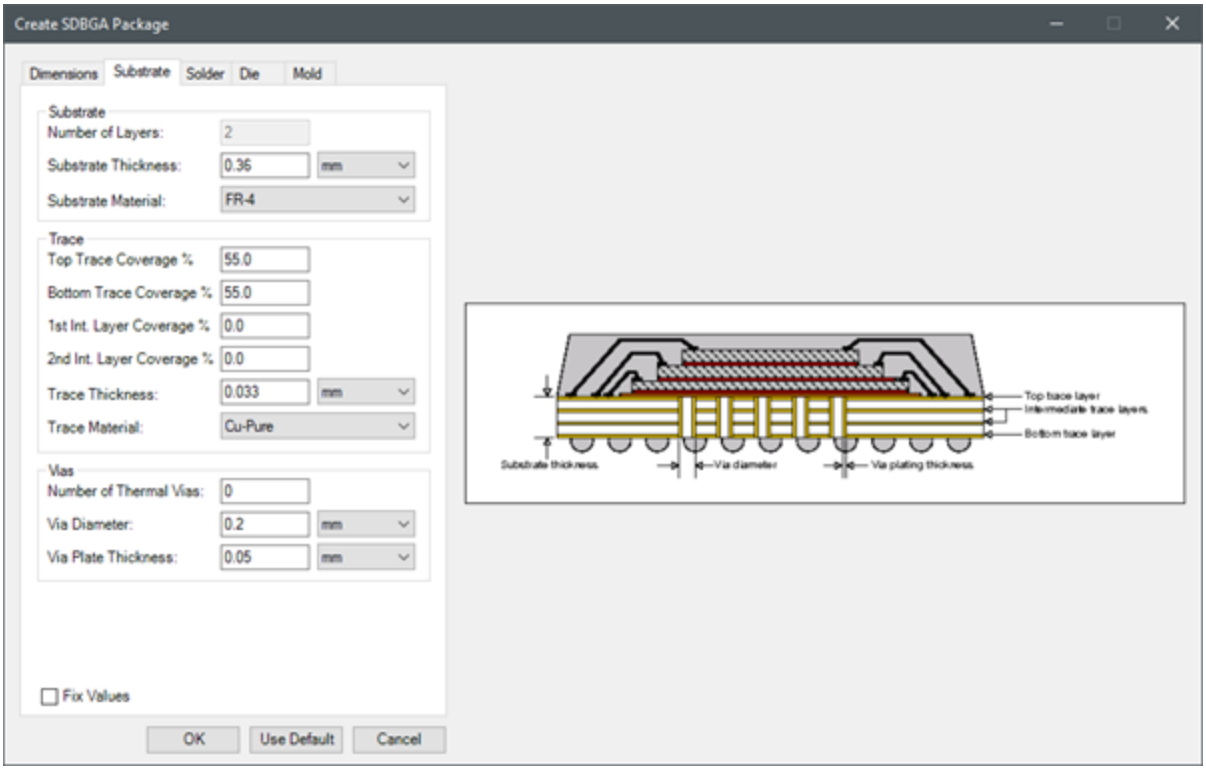
Packages - SDBGA

SDBGA enables you to create geometry and boundary conditions to represent an stacked die ball grid array package.



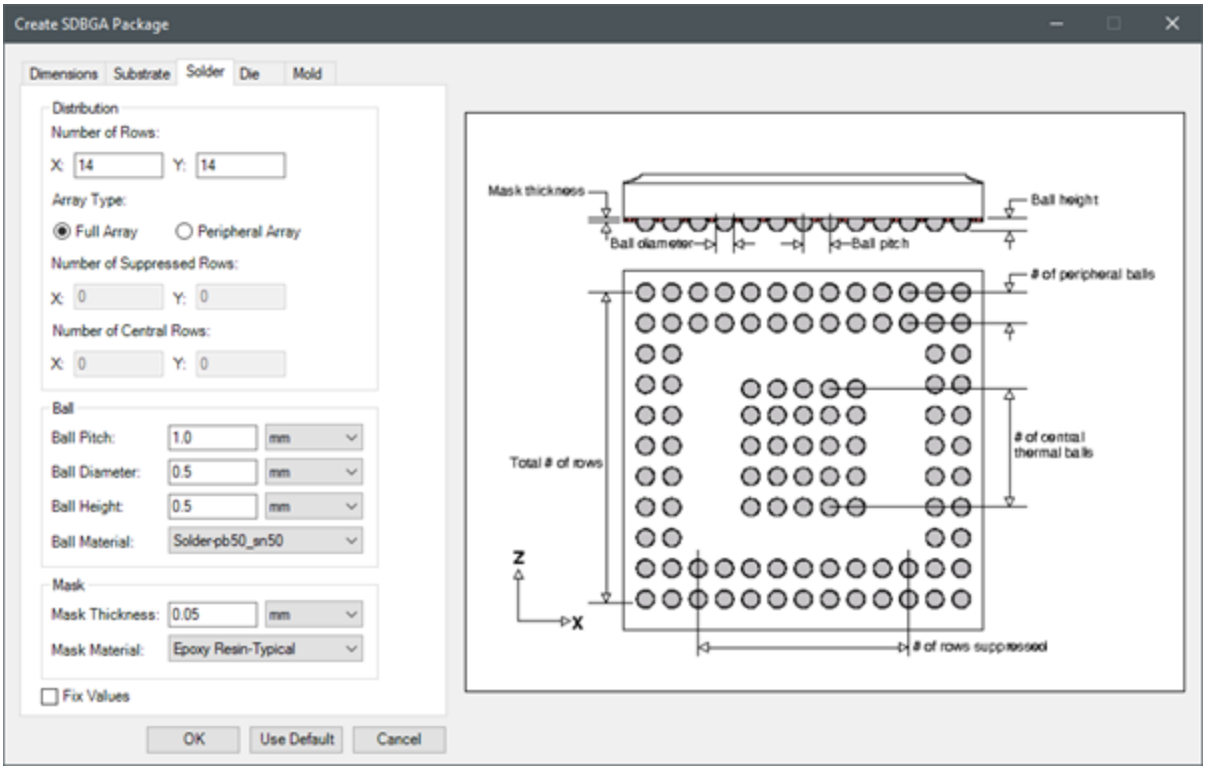
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Package Thickness** depends on the selected **Plane**.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed** or **Compact Conduction Model (CCM)** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



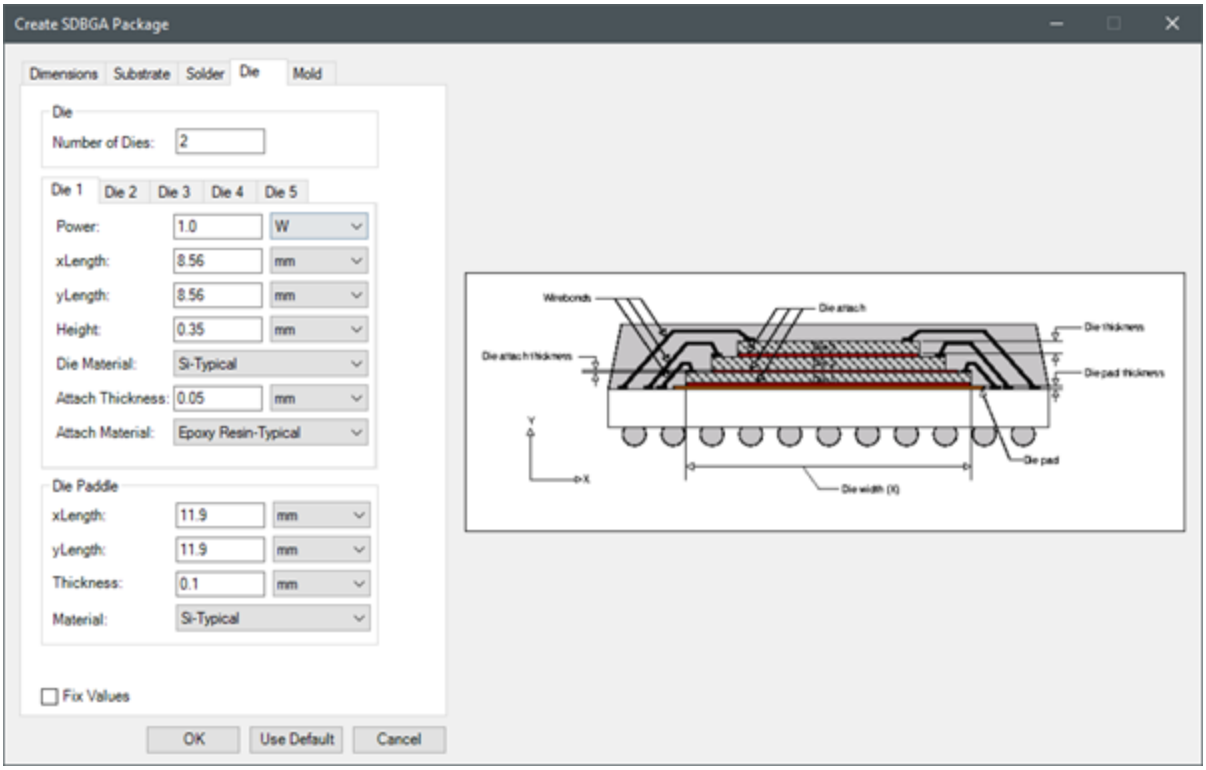
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



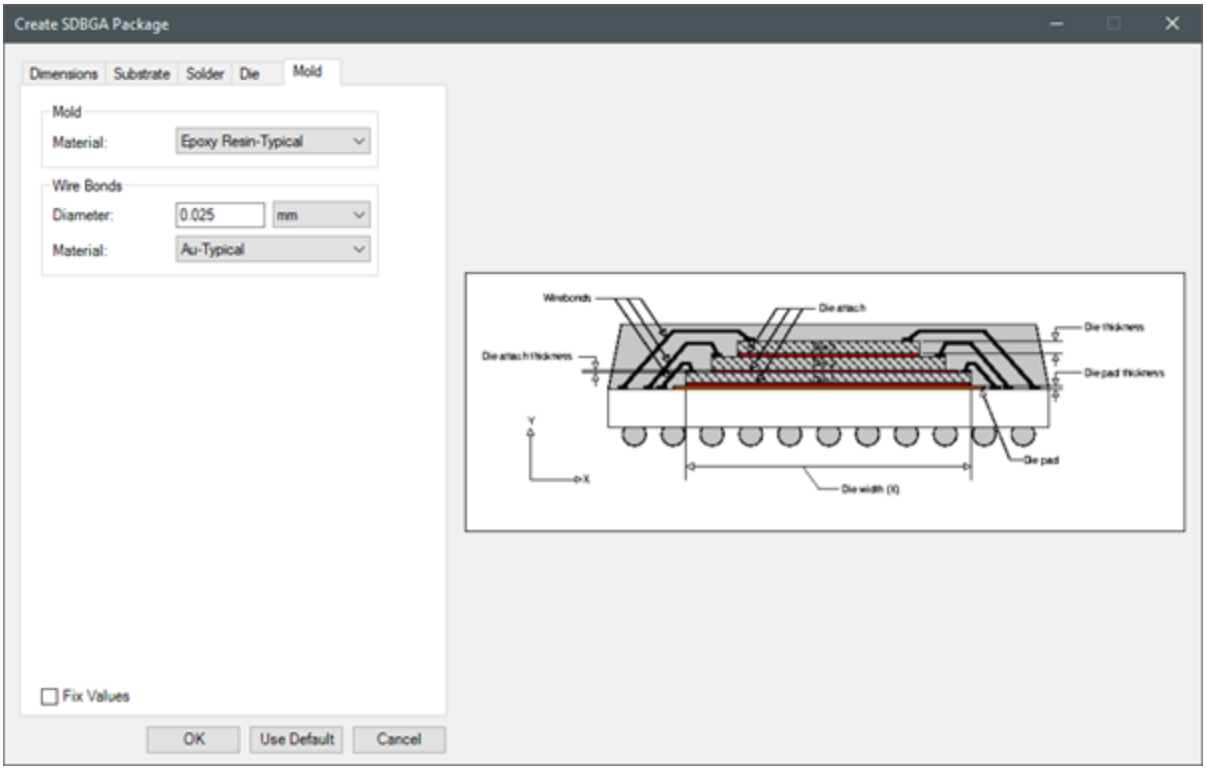
Distribution	Under **Distribution**, enter the number of rows for solder balls, the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed and number of central rows.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.
Mask	Under **Mask**, enter the solder ball mask thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

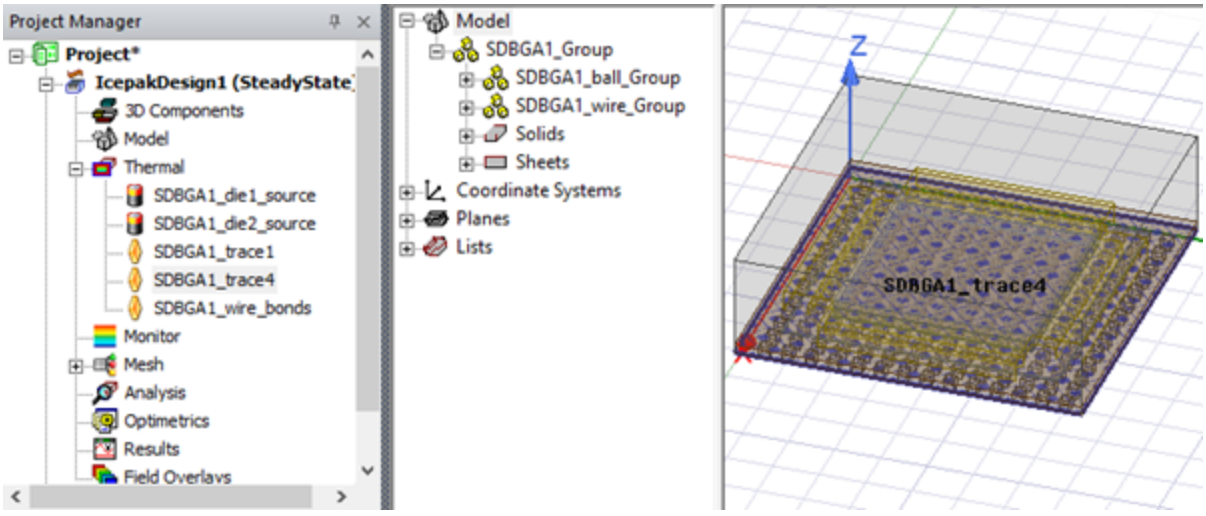


Die	Under **Die**, specify the number of dies on the package. You can specify up to five dies.
Die 1-5	On the **Die** tab(s), specify the die power, the length of the die in each coordinate direction, and die material. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the thickness and material of the die attach.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter and material of the of the wire bonds.



Iron Python Script – *SDBGA.py*:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length1:0.015',
        'Length2:0.015',
        'Package Thickness:0.005',
        'Model Type:Detailed',
        'Symmetry:Full',
        'Substrate Thickness:0.00036',
        'Substrate Material:FR-4',
        'Top Trace:55.0',
        'Bottom Trace:55.0',
        '1st Int. Layer Coverage:0.0',
        '2nd Int. Layer Coverage:0.0',
        'Trace Thickness:0.000033',
        'Trace Material:Cu-Pure',
        'Number of Thermal Vias:0',
        'Via Diameter:0.0002',
        'Via Plate Thickness:0.00005',
        'Number of Rows 1:14',
        'Number of Rows 2:14',
        'Array Type:Full Array',
        'Suppressed Rows 1:0',
        'Suppressed Rows 2:0',
        'Central Rows 1:0',
        'Central Rows 2:0',
        'Ball Pitch:0.001',
        'Ball Diameter:0.0005',
        'Ball Height:0.0005',
        'Ball Material:Solder-pb50_sn50',
        'Mask Thickness:0.00005',
        'Mask Material:Epoxy Resin-Typical',
        'Number of Dies:2',
        'Die 1 Length1:0.00856',
        'Die 1 Length2:0.00856',
        'Die 1 Height:.00035',
        'Die 1 Power:1',
        'Die 1 Material:Si-Typical',
        'Attach 1 Thickness:0.00005',
        'Attach 1 Material:Epoxy Resin-Typical',
        'Die 2 Length1:0.00856',
```

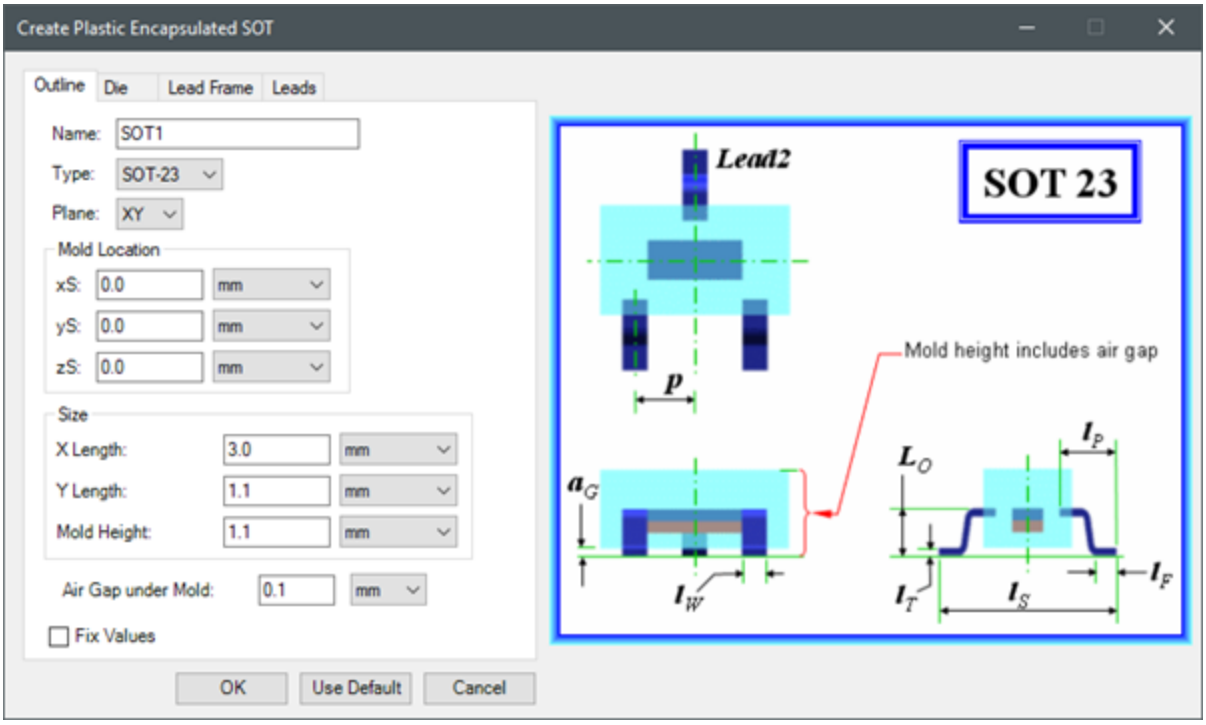

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'Die 2 Length2:0.00856',
        'Die 2 Height:.00035',
        'Die 2 Power:1',
        'Die 2 Material:Si-Typical',
        'Attach 2 Thickness:0.00005',
        'Attach 2 Material:Epoxy Resin-Typical',
        'Die 3 Length1:0.00856',
        'Die 3 Length2:0.00856',
        'Die 3 Height:.00035',
        'Die 3 Power:1',
        'Die 3 Material:Si-Typical',
        'Attach 3 Thickness:0.00005',
        'Attach 3 Material:Epoxy Resin-Typical',
        'Die 4 Length1:0.00856',
        'Die 4 Length2:0.00856',
        'Die 4 Height:.00035',
        'Die 4 Power:1',
        'Die 4 Material:Si-Typical',
        'Attach 4 Thickness:0.00005',
        'Attach 4 Material:Epoxy Resin-Typical',
        'Die 5 Length1:0.00856',
        'Die 5 Length2:0.00856',
        'Die 5 Height:.00035',
        'Die 5 Power:1',
        'Die 5 Material:Si-Typical',
        'Attach 5 Thickness:0.00005',
        'Attach 5 Material:Epoxy Resin-Typical',
        'Pad Length1:0.0119',
        'Pad Length2:0.0119',
        'Pad Thickness:0.0001',
        'Pad Material:Si-Typical',
        'Mold Material:Epoxy Resin-Typical',
        'Wire Diameter:0.000025',
        'Wire Material:Au-Typical'
        ]
oDesign.RunToolkit("SysLib", "Geometry/Packages/SDBGA", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/SDBGA", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Packages/SDBGA", data)
```

Packages - SOT

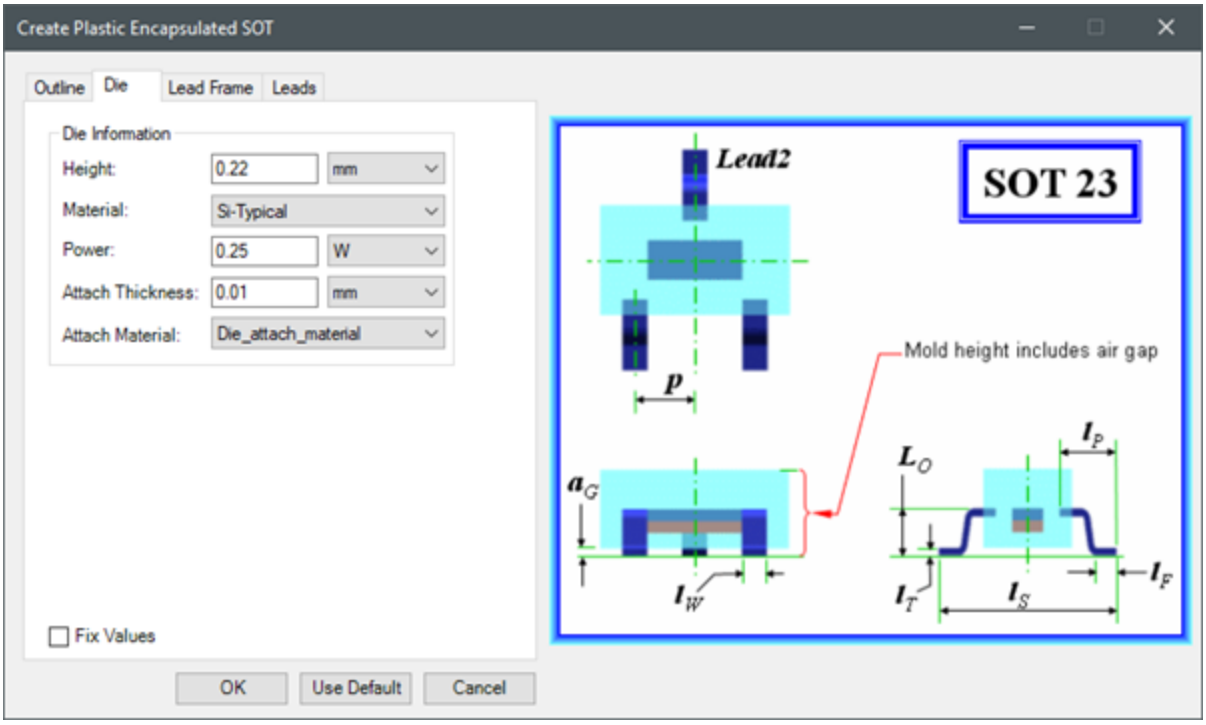
SOT enables you to create geometry and boundary conditions to represent an plastic encapsulated small outline transistor.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



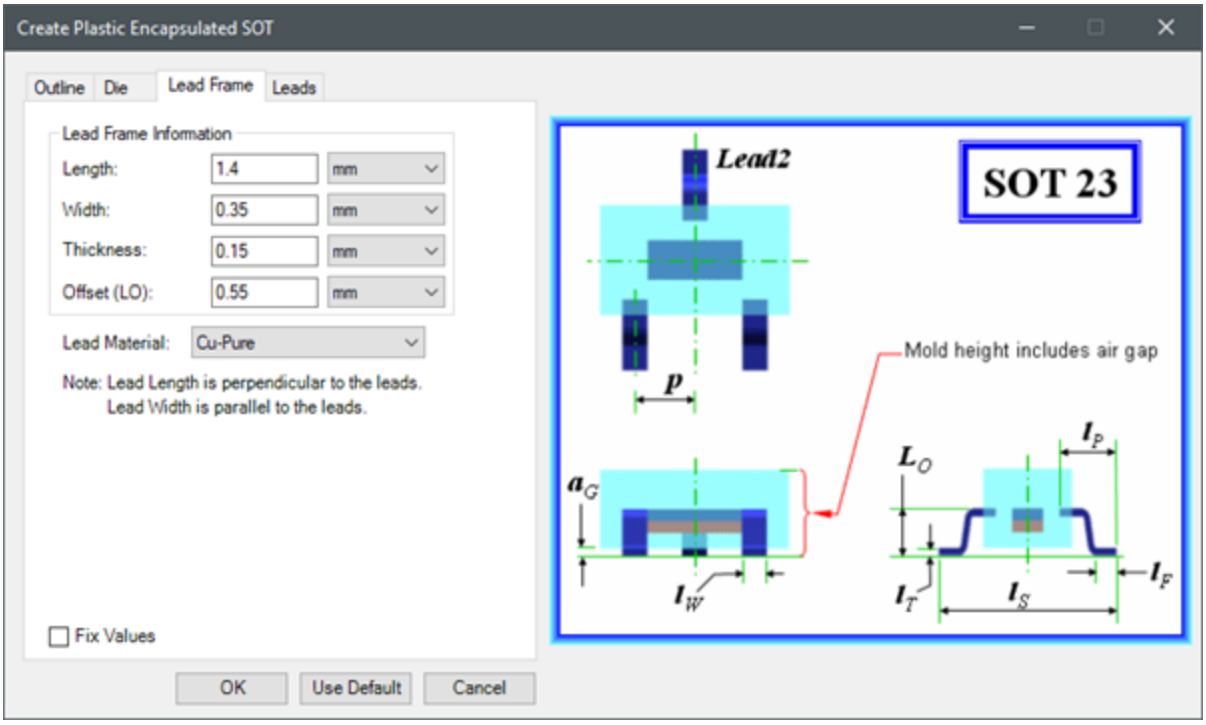
Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Type	**Type** specifies whether the package is type SOT-23, SOT-223, or SOT-89.
Plane	**Plane** specifies the plane on which to create the package.
Mold Location	Under **Location**, specify the starting coordinates for the mold geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length of the package in each coordinate direction. The coordinate direction represented by **Mold Height** depends on the selected **Plane**.
Air Gap under Mold	**Air Gap under Mold** specifies the airgap under the mold.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



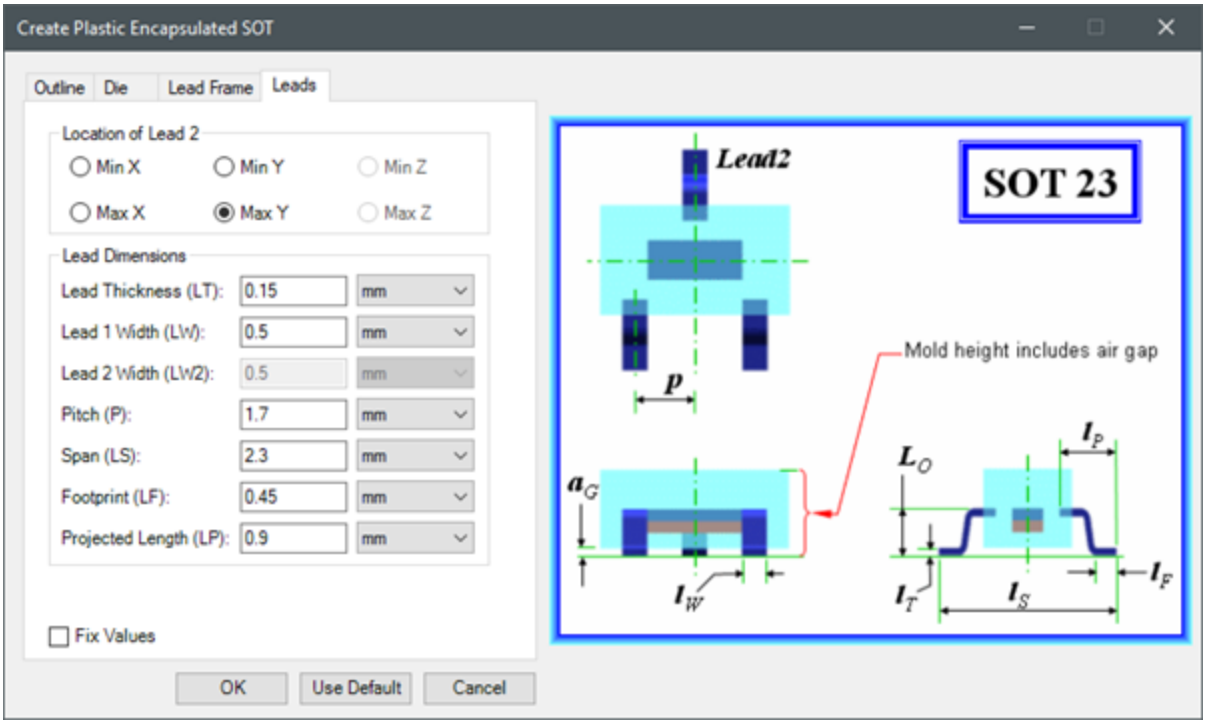
Die Information	Under **Die Information**, specify the die height, material, and power. Also, specify the thickness and material of the die attach.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

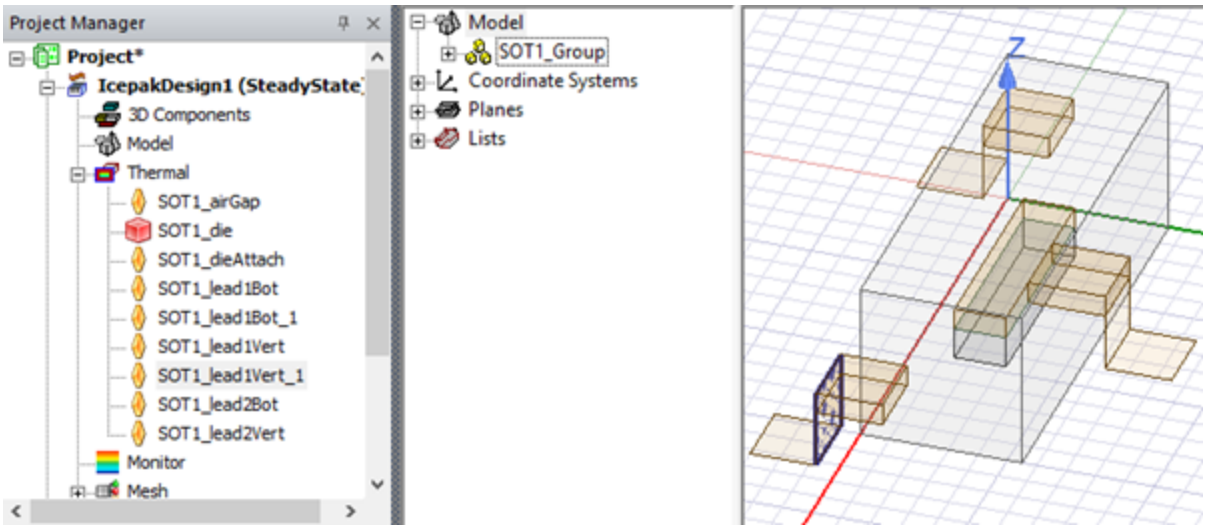


Lead Frame Information	Under **Lead Frame Information**, specify the lead frame length, width, thickness, and offset.
Lead Material	**Lead Material** specifies the material of the leads.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Location of Lead 2	Under **Location of Lead 2**, specify the coordinate direction for the location of lead 2.
Lead Dimensions	Under **Lead Dimensions**, specify the lead thickness, widths, pitch, span, footprint, and projected length.



Iron Python Script – *SOT.py*:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

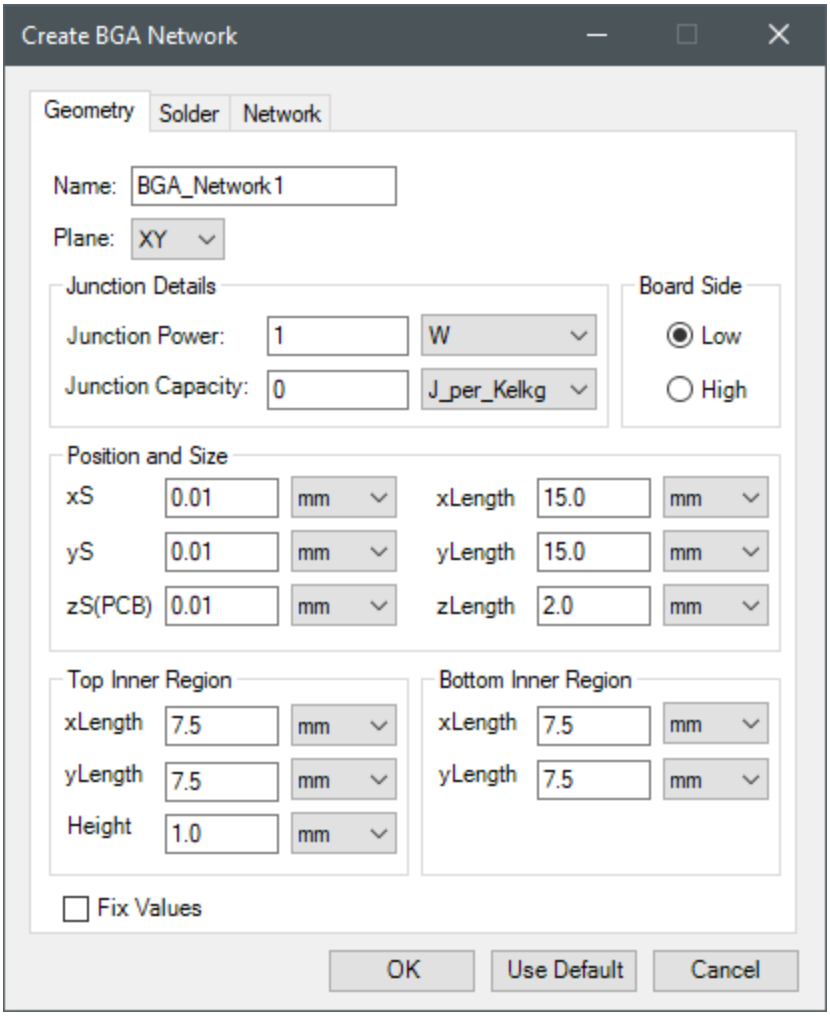
The following is the toolkit's scripting information.

```
scriptInput = ["ng"]
scriptInput.append("Name:SOT1")
scriptInput.append("Type:SOT-23")
scriptInput.append("Plane:XY")
scriptInput.append("length_unit:meter")
scriptInput.append("power_unit:W")
scriptInput.append("xS:0.0")
scriptInput.append("yS:0.0")
scriptInput.append("zS:0.0")
scriptInput.append("Len1:0.003")
scriptInput.append("Len2:0.0011")
scriptInput.append("Thick:0.0011")
scriptInput.append("airGap:0.0001")
scriptInput.append("dieHeight:0.00022")
scriptInput.append("DieMaterial:Si-Typical")
scriptInput.append("attachThick:0.00001")
scriptInput.append("AttachMaterial:Die_attach_material")
scriptInput.append("DiePower:1.0")
scriptInput.append("FrameLength:0.0014")
scriptInput.append("FrameWidth:0.00035")
scriptInput.append("FrameThick: 0.00015")
scriptInput.append("FrameOffset:0.00055")
scriptInput.append("Lead2Axis:Min Y")
scriptInput.append("LeadThick:0.00015")
scriptInput.append("Lead1Width:0.0005")
scriptInput.append("Lead2Width:0.0005")
scriptInput.append("LeadPitch:0.0017")
scriptInput.append("LeadSpan:0.0023")
scriptInput.append("LeadFootprint:0.00045")
scriptInput.append("LeadProjected:0.0009")
scriptInput.append("LeadMaterial:Cu-Pure")
oDesign.RunToolkit("SysLib", "Geometry/Packages/SOT", [])
oDesign.RunToolkit("SysLib", "Geometry/Packages/SOT", ["ng"])
oDesign.RunToolkit("SysLib", "Geometry/Packages/SOT",
scriptInput)
```

Packages - Network Model - BGA_Network

Packages - Network Model - BGA_Network enables you to create geometry and boundary conditions to represent a ball grid array network.

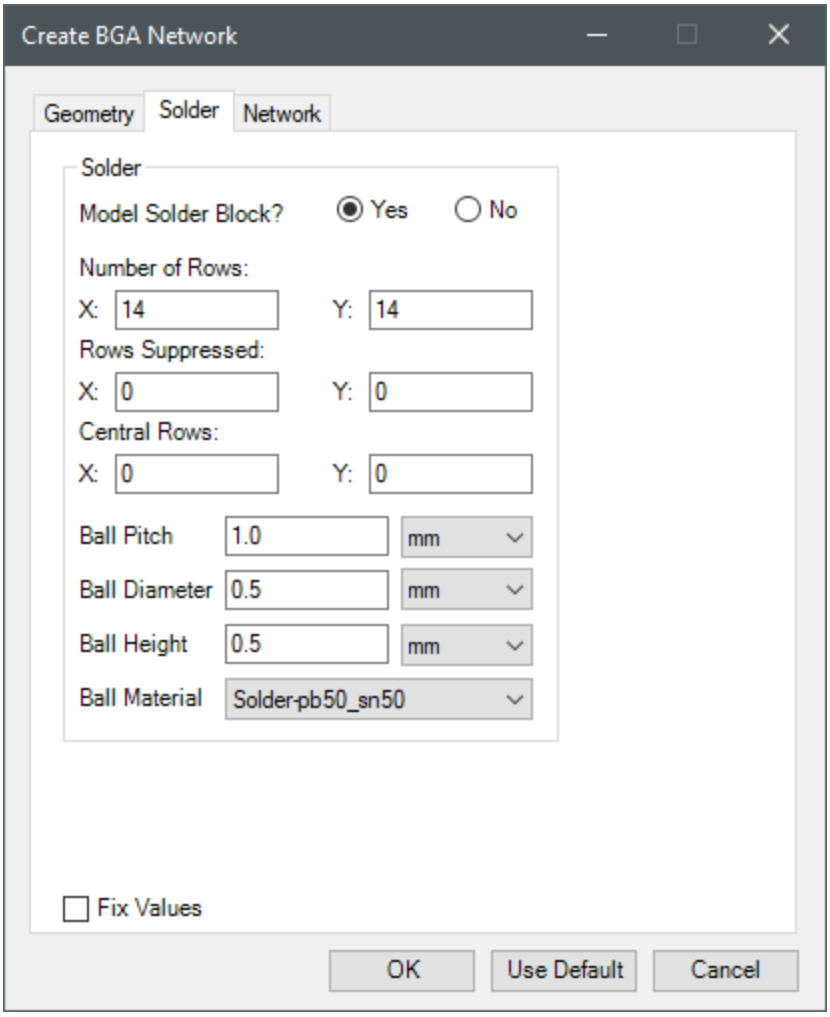
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Junction Details	Under **Junction Details**, specify the junction power and capacity.
Board Side	Under **Board Side**, specify which side of the package the board is located.
Position and Size	Under **Position and Size**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**. Also specify the length of the package in each coordinate direction.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on

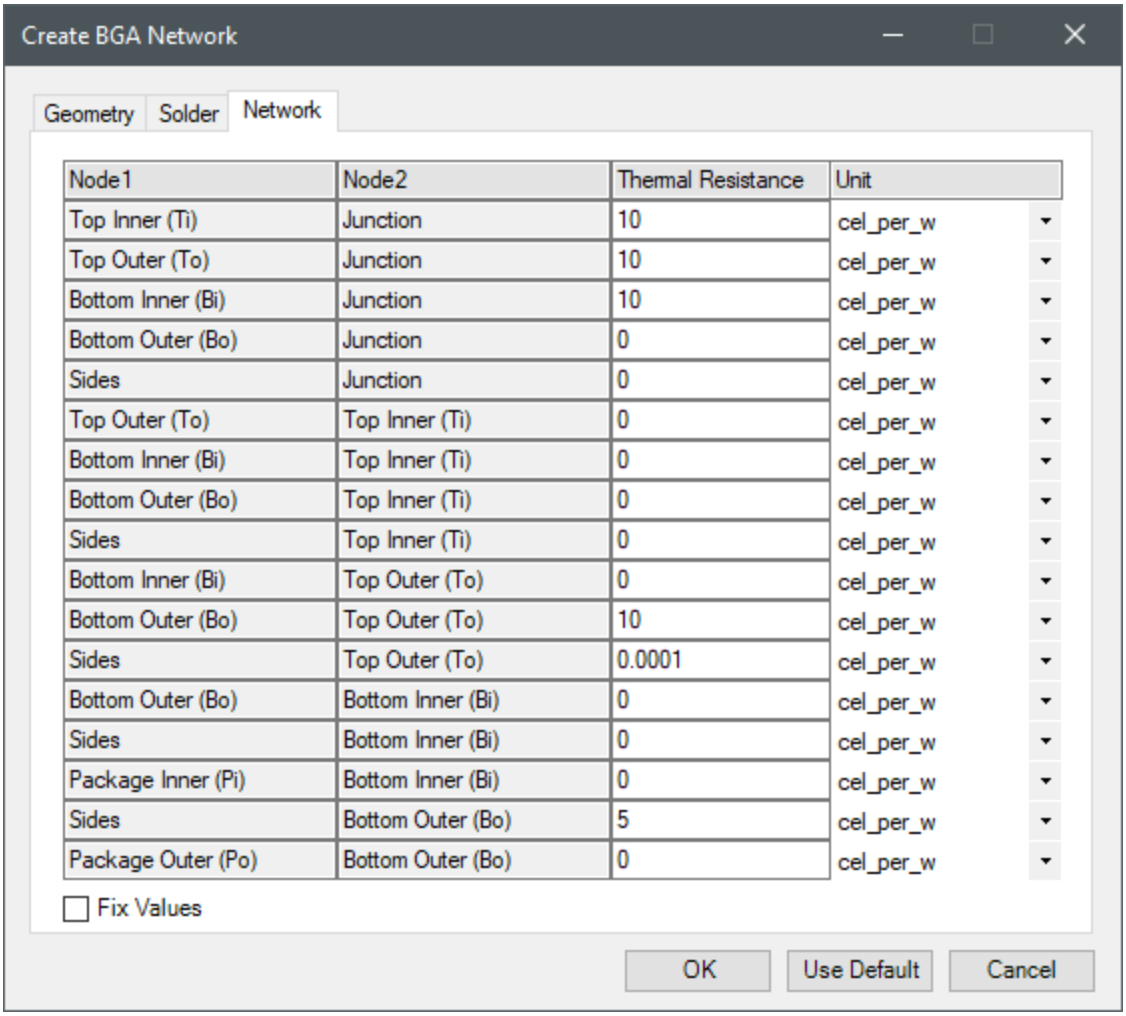
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the bottom side of the package.



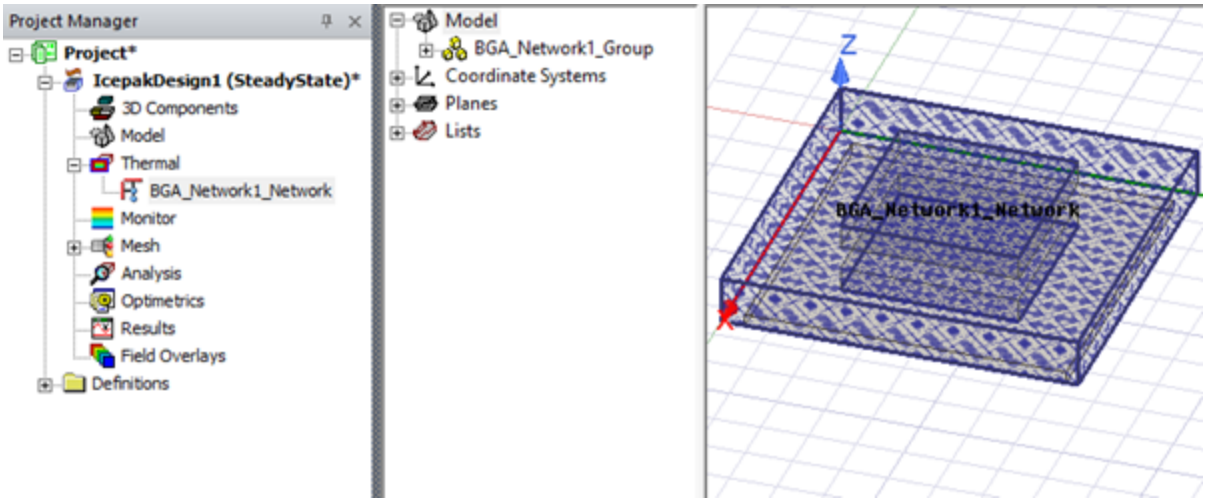
Solder	Under **Solder**, specify whether or not to model a solder block and specify the number of rows, rows to be suppressed, and number of central rows. Also, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Network	On the **Network** tab, enter the thermal resistances for the network nodes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *BGA_Network.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'capacity_unit:J_per_Kelkg',
        'resistance_unit:cel_per_w',
        'Plane:XY',
        'Junction Power:1',
        'Junction Capacity:0',
        'Board Side:Low',
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'xLength:0.015',
        'yLength:0.015',
        'zLength:0.002',
        'Top Inner Length1:0.0075',
        'Top Inner Length2:0.0075',
        'Top Inner Height:0.001',
        'Bottom Inner Length1:0.0075',
        'Bottom Inner Length2:0.0075',
           'Resistances:
[10,10,10,0,0,0,0,0,0,0,10,0.0001,0,0,0,5,0]',
       # this must be in the same order as the UI from top to
bottom
        'Model Solder Block:Yes',
        'Number of Rows 1:14',
        'Number of Rows 2:14',
```

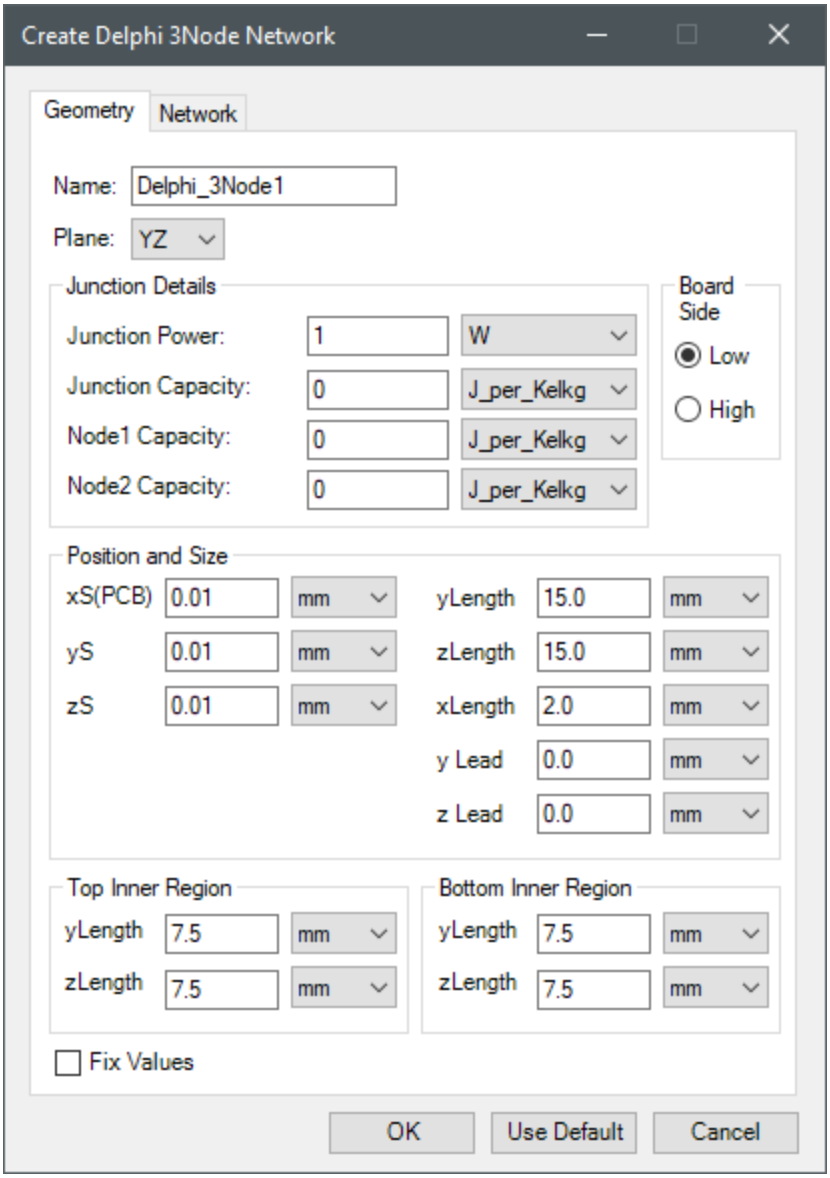

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'Suppressed Rows 1:0',
        'Suppressed Rows 2:0',
        'Central Rows 1:0',
        'Central Rows 2:0',
        'Ball Pitch:0.001',
        'Ball Diameter:0.0005',
        'Ball Height:0.0005',
        'Ball Material:Solder-pb50_sn50'
        ]
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/BGA_Network", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/BGA_Network", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/BGA_Network", data)
```

Packages - Network Model - Delphi_3Node_Network

Packages - Network Model - Delphi_3Node_Network enables you to create geometry and boundary conditions to represent a Delphi 3 node network.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

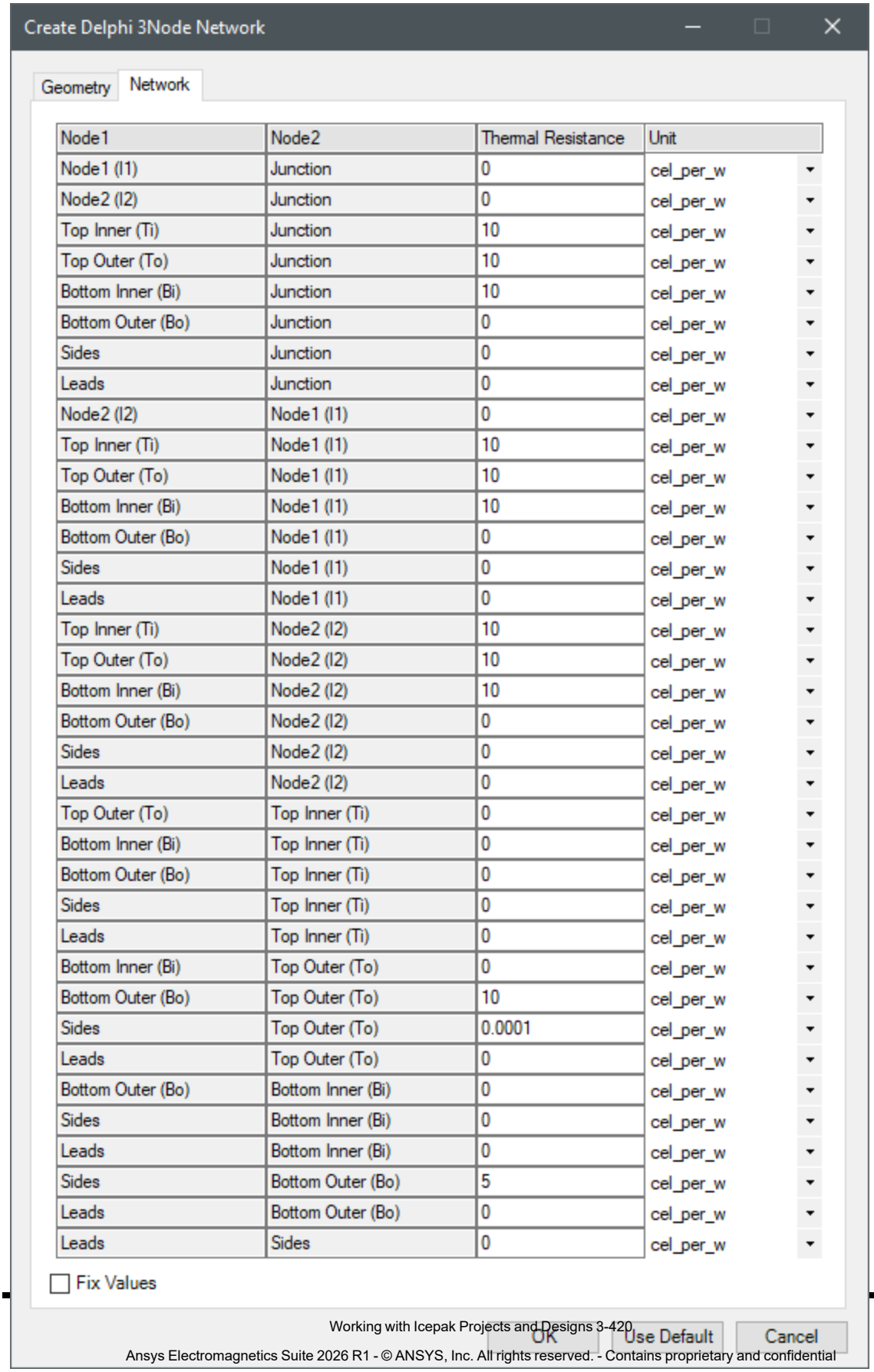


Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Junction Details	Under **Junction Details**, specify the junction power and the capacity of the junction and nodes.
Board Side	Under **Board Side**, specify which side of the package the board is located.
Position and Size	Under **Position and Size**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**. Also specify

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

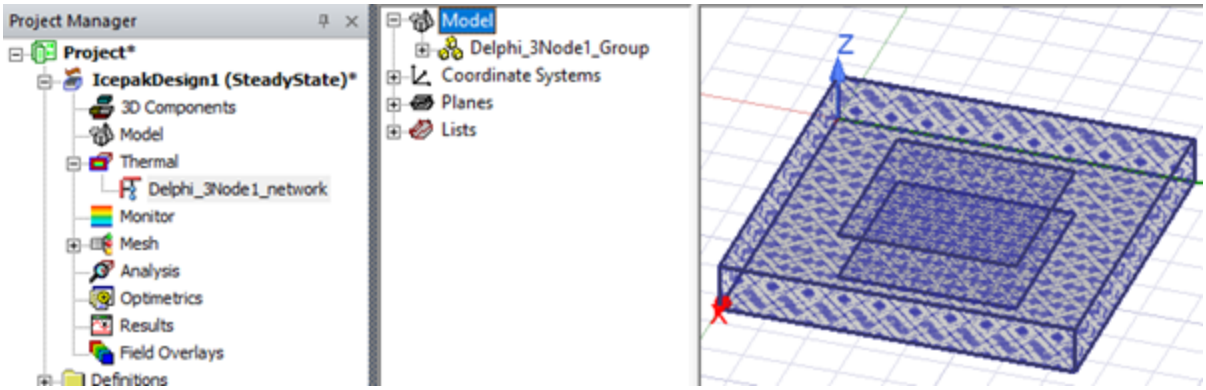
	the length of the package and leads in each coordinate direction.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on the bottom side of the package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network	On the **Network** tab, enter the thermal resistances for the network nodes.



Iron Python Script – *Delphi_3Node_Network.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'capacity_unit:J_per_Kelkg',
        'resistance_unit:cel_per_w',
        'Plane:XY',
        'Junction Power:1',
        'Junction Capacity:0',
        'Node1 Capacity:0',
        'Node2 Capacity:0',
        'Board Side:Low',
        'xStart:0.0',
        'yStart:0.0',
        'zStart:0.0',
        'xLength:0.015',
        'yLength:0.015',
        'zLength:0.002',
        'Lead1:0.0',
        'Lead2:0.0',
        'Top Inner Length1:0.0075',
        'Top Inner Length2:0.0075',
        'Bottom Inner Length1:0.0075',
        'Bottom Inner Length2:0.0075',
           'Resistances:
[0,0,10,10,10,0,0,0,0,10,10,10,0,0,0,10,10,10,0,0,0,0,0,0,0,0,0,
       10,0.0001,0,0,0,0,5,0,0]'
```

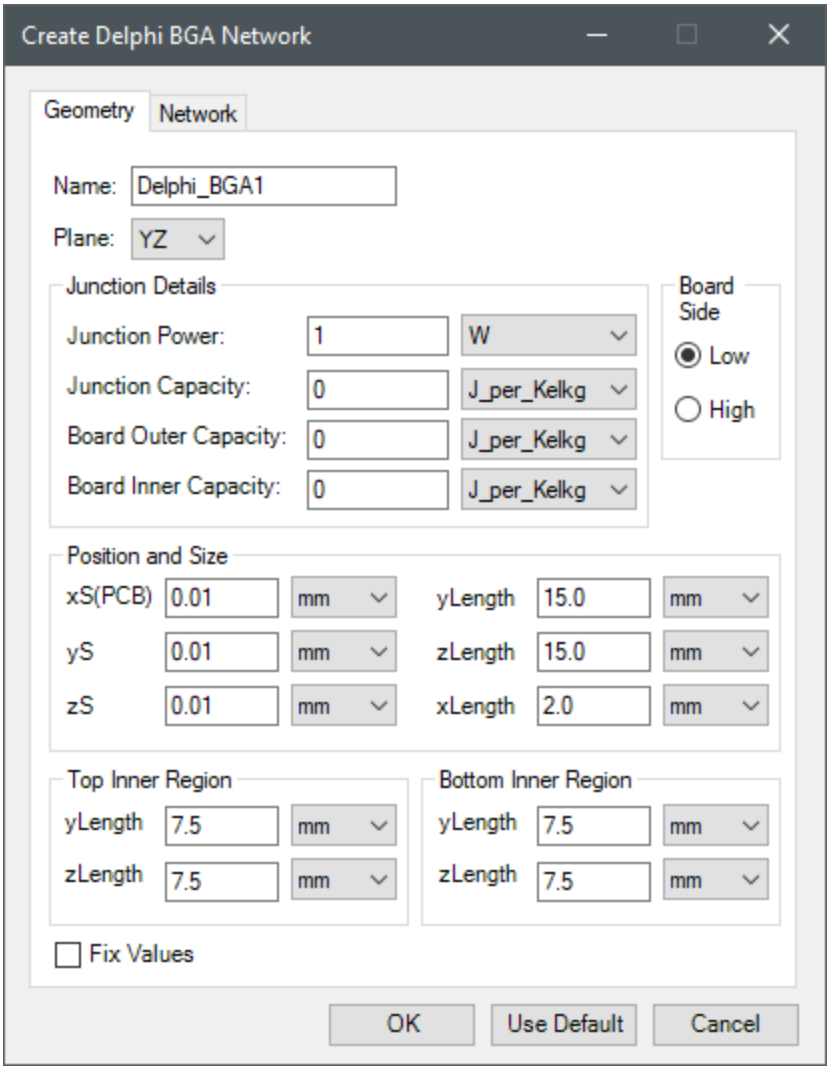

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        # this must be in the same order as the UI from top to
bottom
         ]
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_3Node_Network", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_3Node_Network", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_3Node_Network", data)
```

Packages - Network Model - Delphi_BGA_Network

Packages - Network Model - Delphi_BGA_Network enables you to create geometry and boundary conditions to represent Delphi ball grid array network.

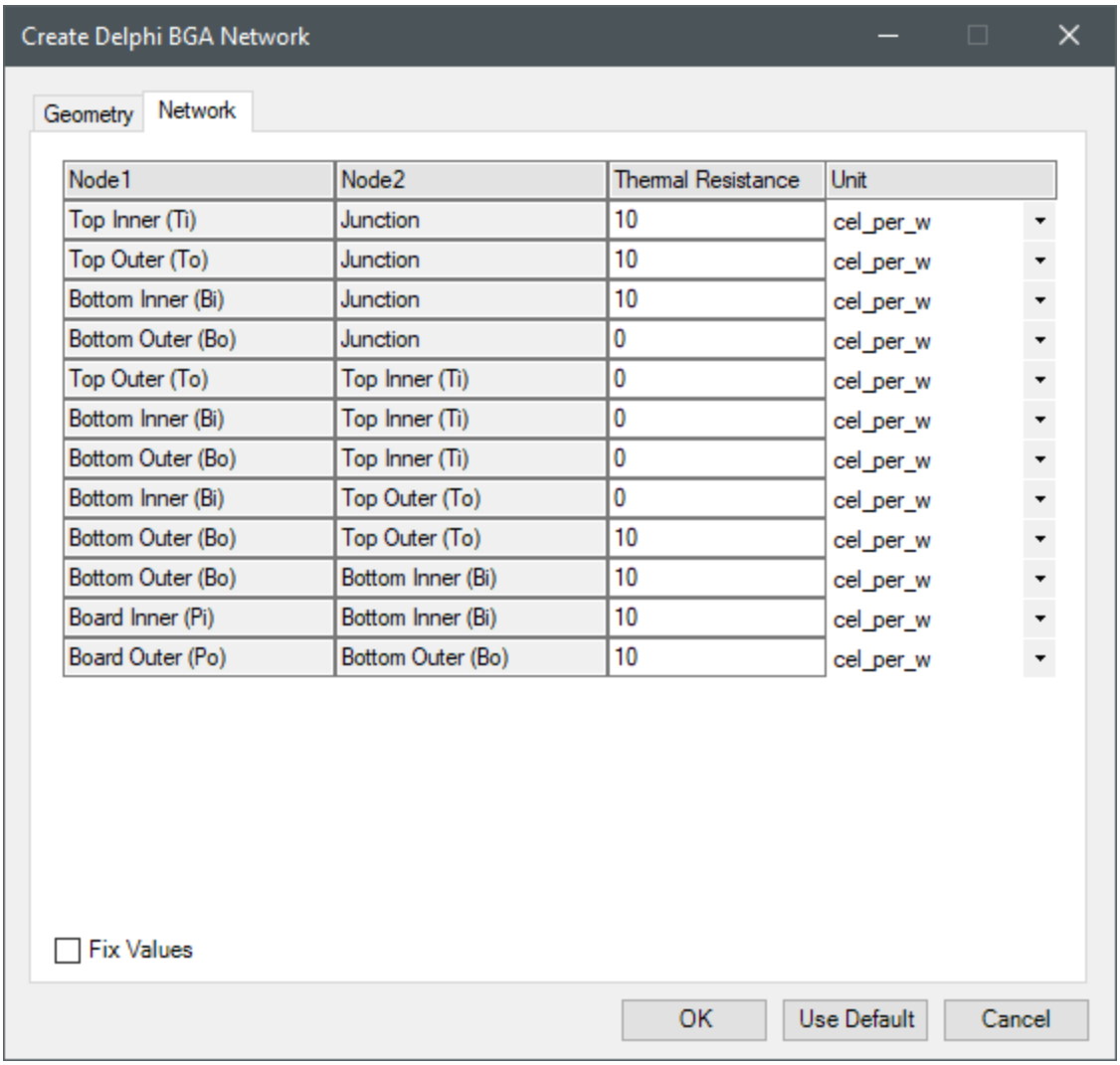
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Junction Details	Under **Junction Details**, specify the junction power and the capacity of the junction and board.
Board Side	Under **Board Side**, specify which side of the package the board is located.
Position and Size	Under **Position and Size**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**. Also specify the length of the package and leads in each coordinate direction.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the

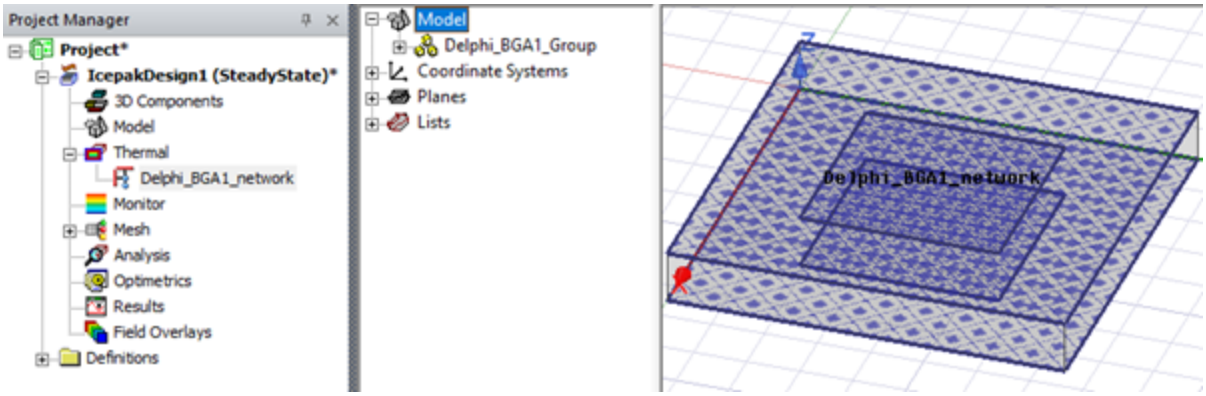
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on the bottom side of the package.



Network	On the **Network** tab, enter the thermal resistances for the network nodes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *Delphi_BGA_Network.py*:

The following is the toolkit's scripting information.

```
 data = ["ng",
         'length_unit:meter',
         'power_unit:W',
         'capacity_unit:J_per_Kelkg',
         'resistance_unit:cel_per_w',
         'Plane:XY',
         'Junction Power:1',
         'Junction Capacity:0',
         'Board Outer Capacity:0',
         'Board Inner Capacity:0',
         'Board Side:Low',
         'xStart:0.0',
         'yStart:0.0',
         'zStart:0.0',
         'xLength:0.015',
         'yLength:0.015',
         'zLength:0.002',
         'Top Inner Length1:0.0075',
         'Top Inner Length2:0.0075',
         'Bottom Inner Length1:0.0075',
         'Bottom Inner Length2:0.0075',
            'Resistances:[10,10,10,0,0,0,0,0,10,10,10,10]'
        # this must be in the same order as the UI from top to
bottom
         ]
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_BGA_Network", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_BGA_Network", ["ng"])
```

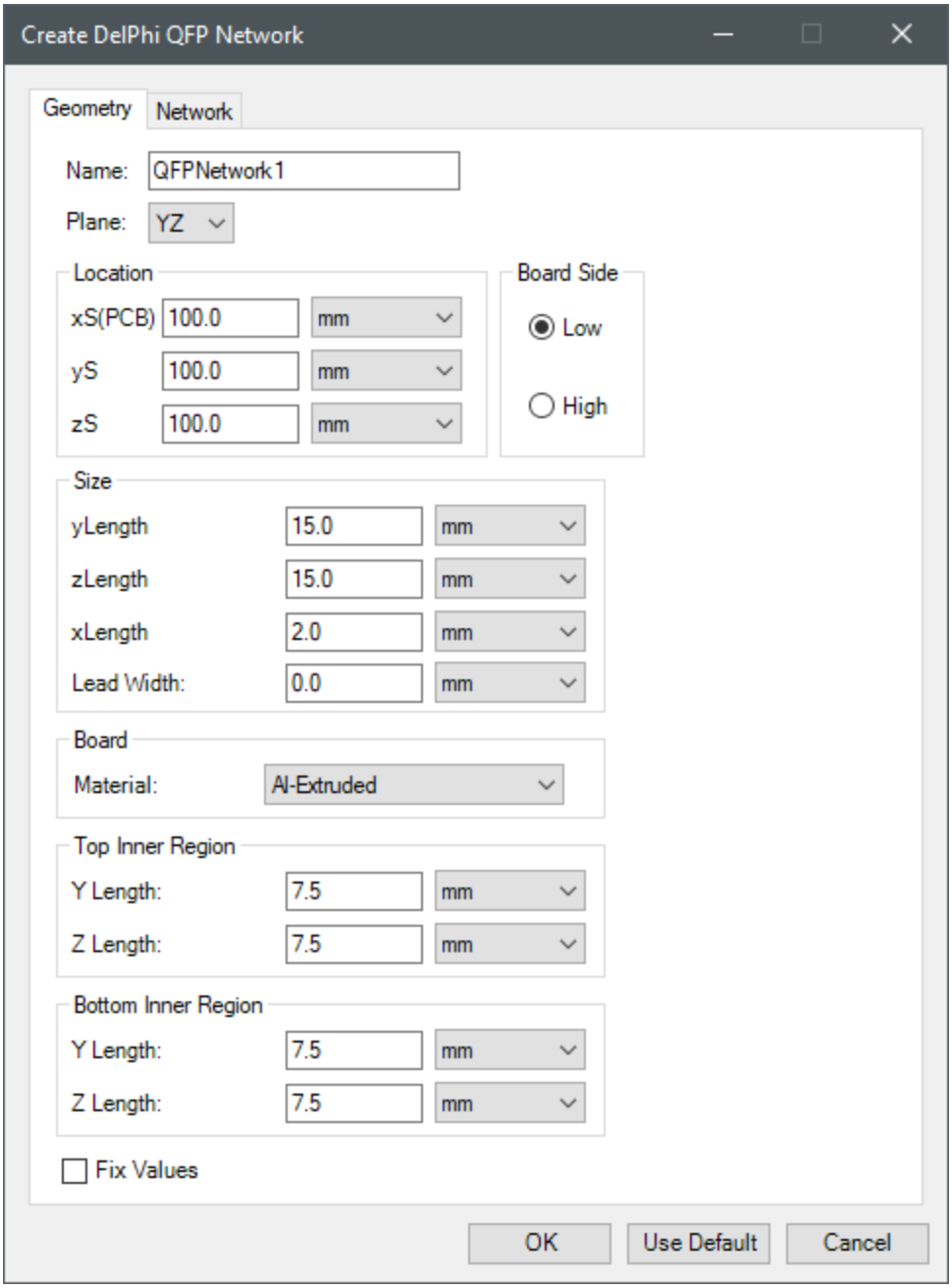

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_BGA_Network", data)
```

Packages - Network Model - Delphi_QFP_Network

Packages - Network Model - Delphi_QFP_Network enables you to create geometry and boundary conditions to represent a Delphi quad flat package network.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

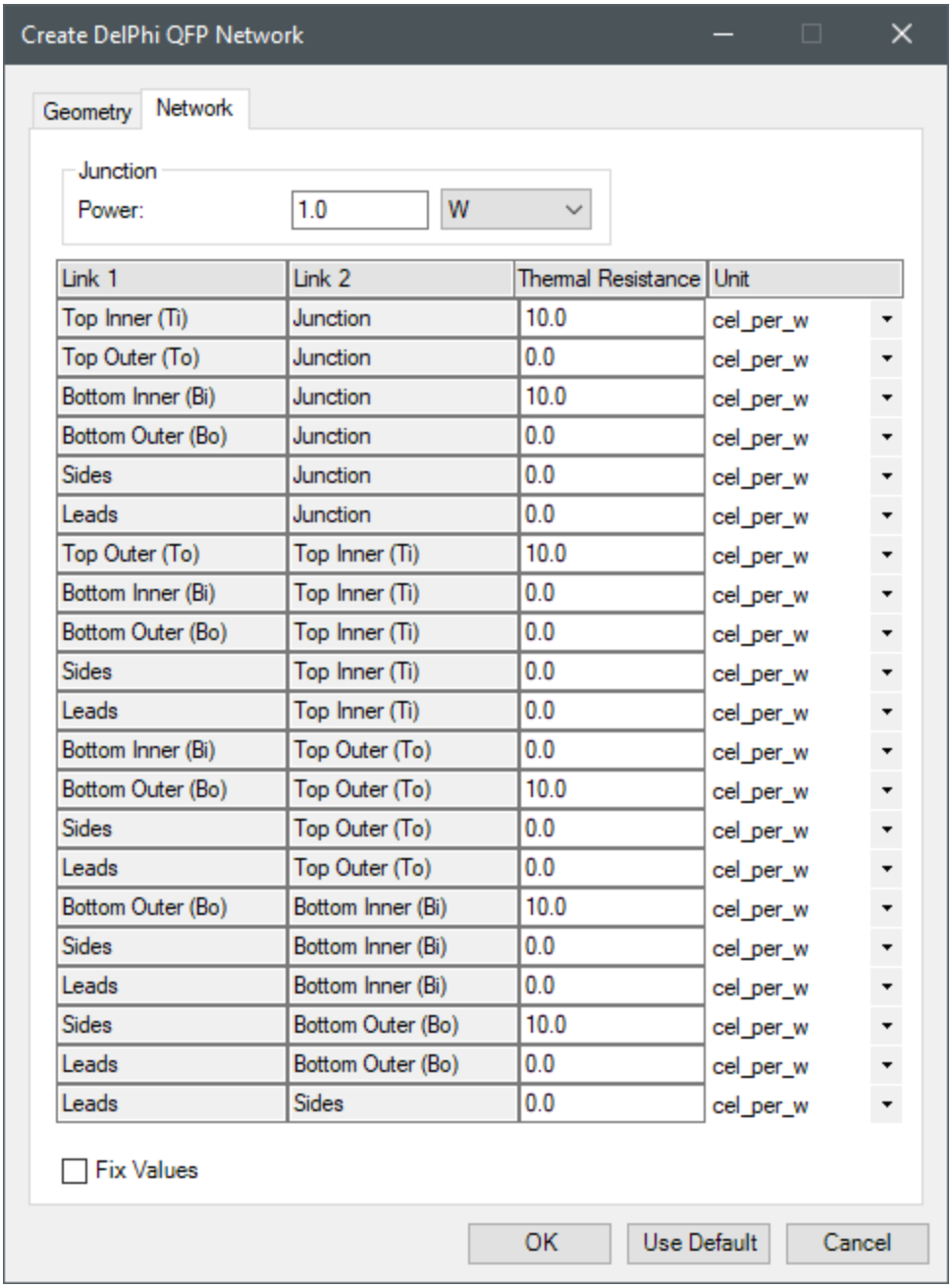


Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

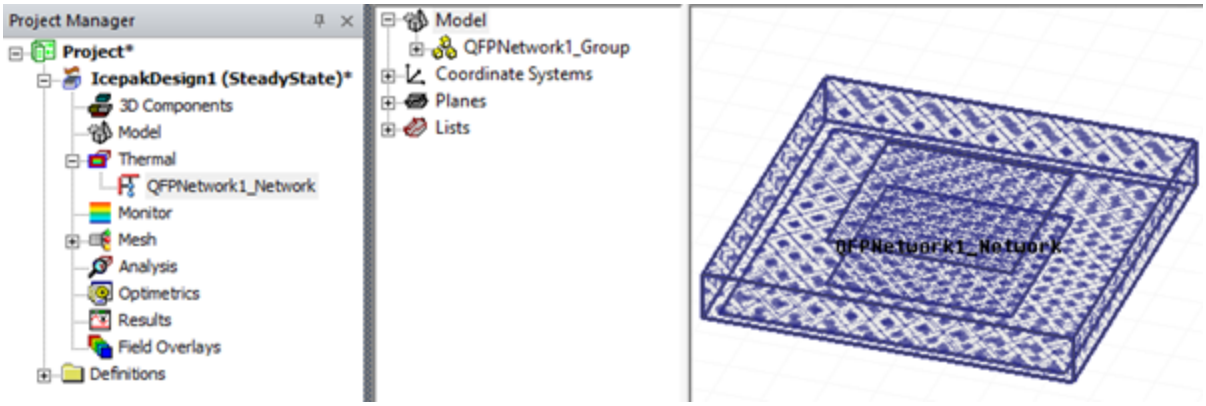
Board Side	Under **Board Side**, specify which side of the package the board is located.
Size	Under **Size**, specify the length of the package in each coordinate direction and the lead width.
Board	Under **Board**, select the board material.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on the bottom side of the package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Junction	Under **Junction**, specify the junction power.
Network	On the **Network** tab, enter the thermal resistances for the network nodes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *Delphi_QFP_Network.py*:

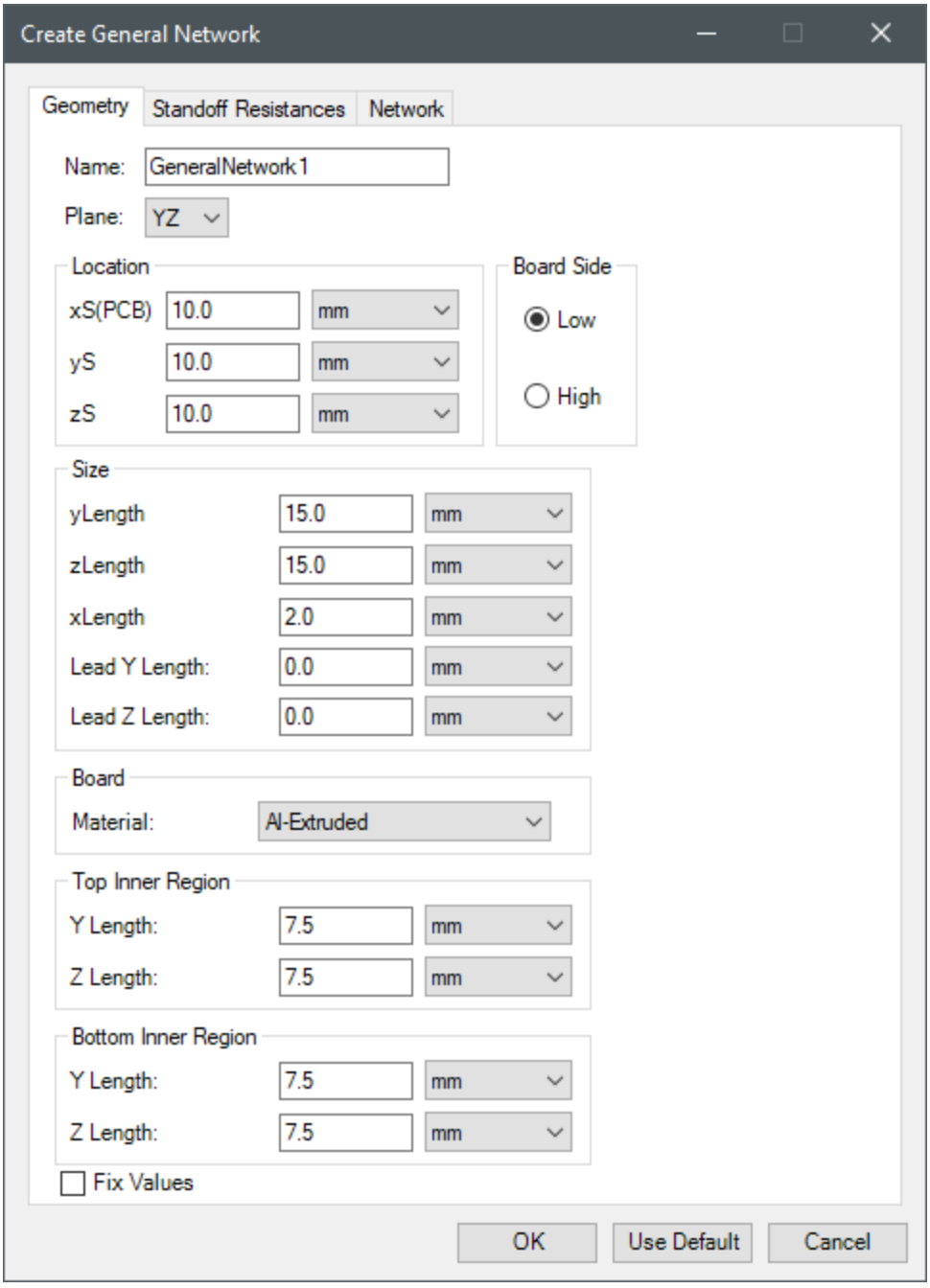
The following is the toolkit's scripting information.

```
 scriptInput = ["ng"]
 scriptInput.append("Name:QFPNetworkA")
 scriptInput.append("Plane:XY")
 scriptInput.append("length_unit:meter")
 scriptInput.append("power_unit:W")
 scriptInput.append("resistance_unit:cel_per_w")
 scriptInput.append("xS:0.1")
 scriptInput.append("yS:0.1")
 scriptInput.append("zS:0.1")
 scriptInput.append("BoardSide:Low")
 scriptInput.append("Len1:0.015")
 scriptInput.append("Len2:0.015")
 scriptInput.append("Thick:0.002")
 scriptInput.append("LeadWidth:0.001")
 scriptInput.append("BoardMaterial:Al-Extruded")
 scriptInput.append("TopLength1:0.0075")
 scriptInput.append("TopLength2:0.0075")
 scriptInput.append("BottomLength1:0.0075")
 scriptInput.append("BottomLength2:0.0075")
 scriptInput.append("JunctionPower:1.0")
 scriptInput.append("Links:
[10,0,10,0,0,0,0,0,10,0,0,0,0,10,0,0,0,0,10,0,0]")
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_QFP_Network", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_QFP_Network", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/Delphi_QFP_Network", scriptInput)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Packages - Network Model - GeneralNetwork

Packages - Network Model - GeneralNetwork enables you to create geometry and boundary conditions to represent a general network.

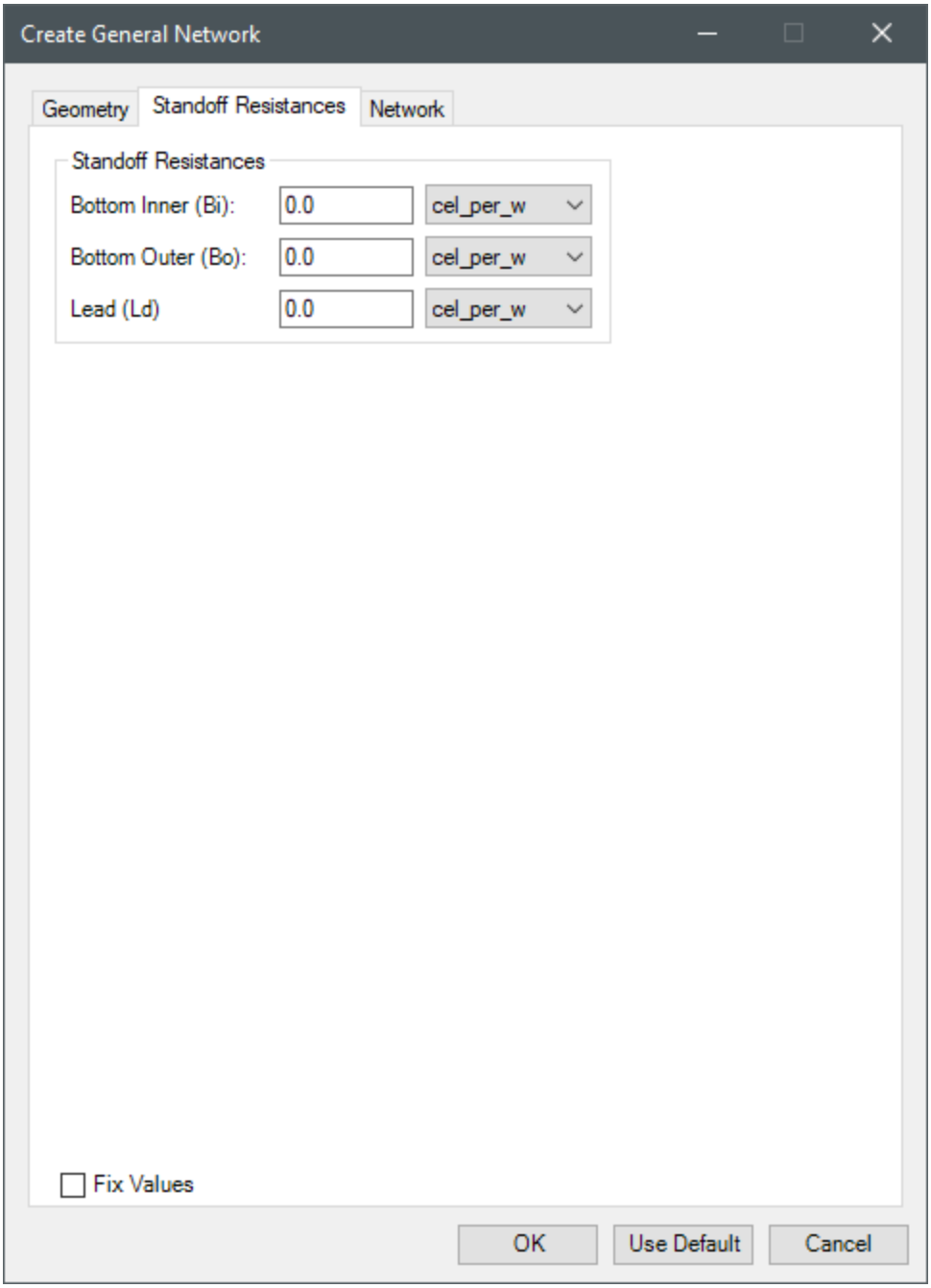


Name	The **Name** appears in the **Project Manager** window, history tree, and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

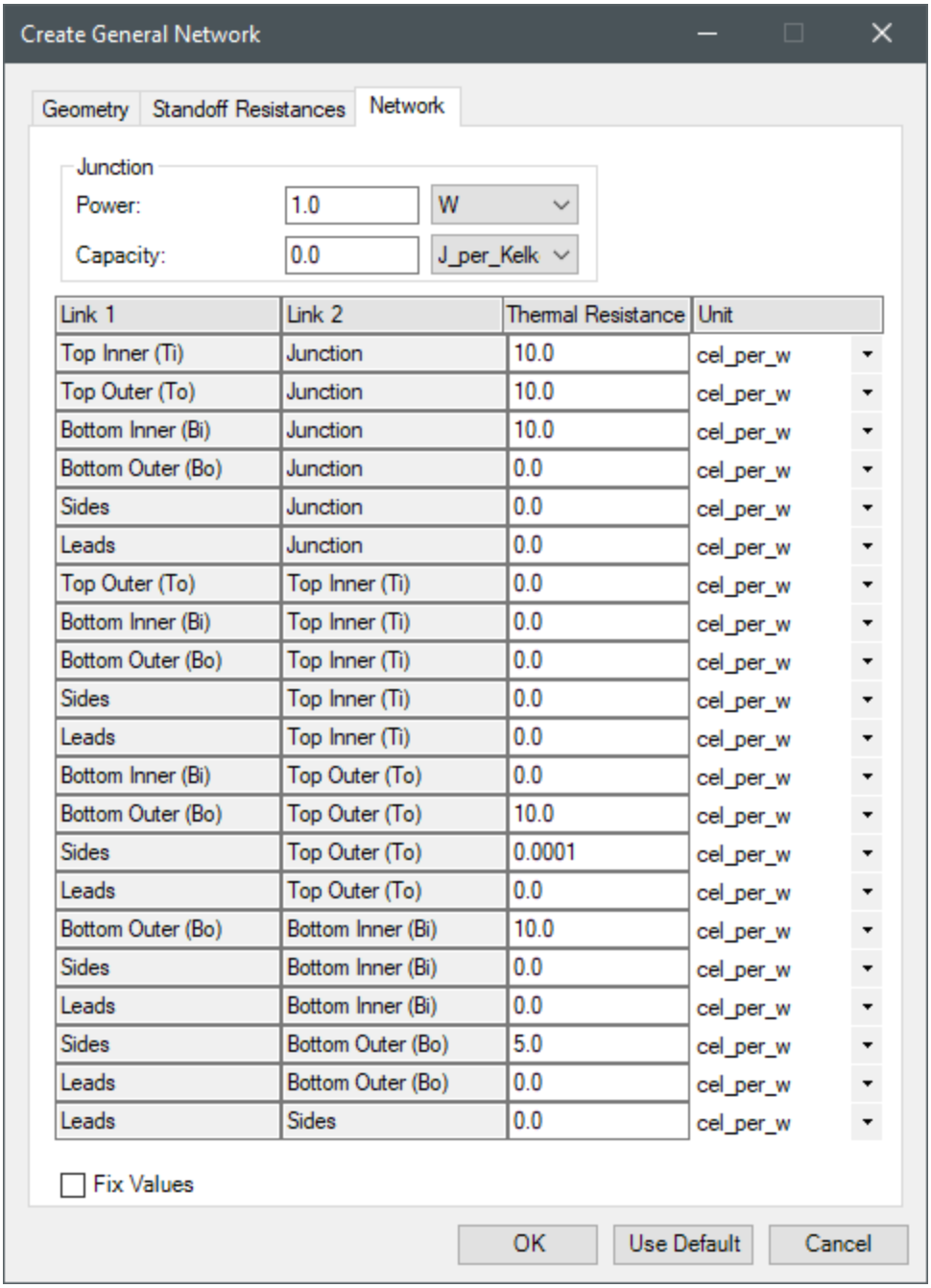
	3D Modeler window.
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Board Side	Under **Board Side**, specify which side of the package the board is located.
Size	Under **Size**, specify the length of the package and leads in each coordinate direction.
Board	Under **Board**, select the board material.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on the bottom side of the package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



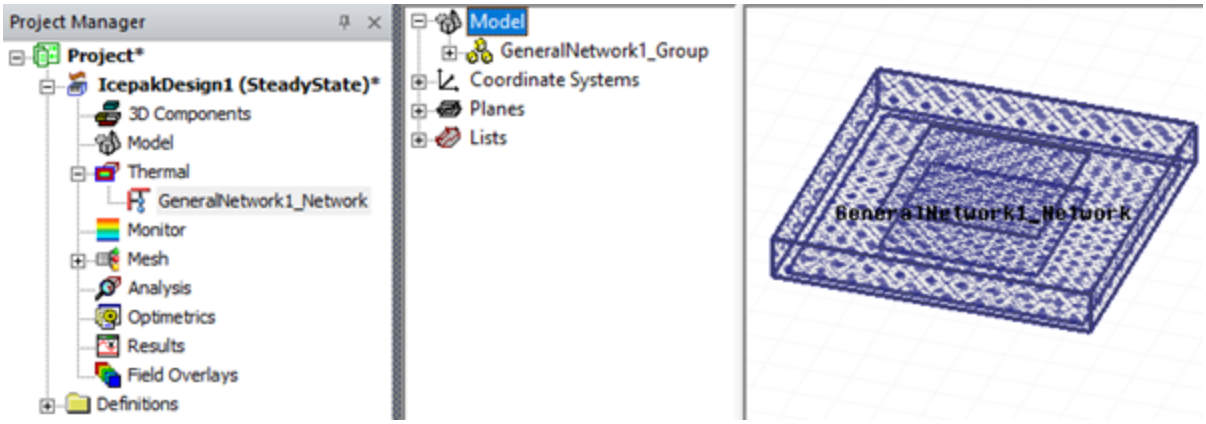
Standoff Resistances	Under **Standoff Resistances**, specify the resistance of the bottom, inner, and lead standoff resistances.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Junction	Under **Junction**, specify the junction power and capacity.
Network	On the **Network** tab, enter the thermal resistances for the network nodes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *General_3Node_Network.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'capacity_unit:J_per_Kelkg',
        'resistance_unit:cel_per_w',
        'Plane:XY',
        'Junction Power:1',
        'Junction Capacity:0',
        'Node1 Capacity:0',
        'Node2 Capacity:0',
        'Board Side:Low',
        'Bottom Inner Standoff Resistance:0',
        'Bottom Outer Standoff Resistance:0',
        'Leads Standoff Resistance:0',
        'xStart:0.0',
        'yStart:0.0',
        'zStart:0.0',
        'xLength:0.015',
        'yLength:0.015',
        'zLength:0.002',
        'Lead1:0.0',
        'Lead2:0.0',
        'Top Inner Length1:0.0075',
        'Top Inner Length2:0.0075',
        'Bottom Inner Length1:0.0075',
        'Bottom Inner Length2:0.0075',
           'Resistances:
[0,0,10,10,10,0,0,0,0,10,10,10,0,0,0,10,10,10,0,0,0,0,0,0,0,0,0,
       10,0.0001,0,0,0,0,5,0,0]'
```

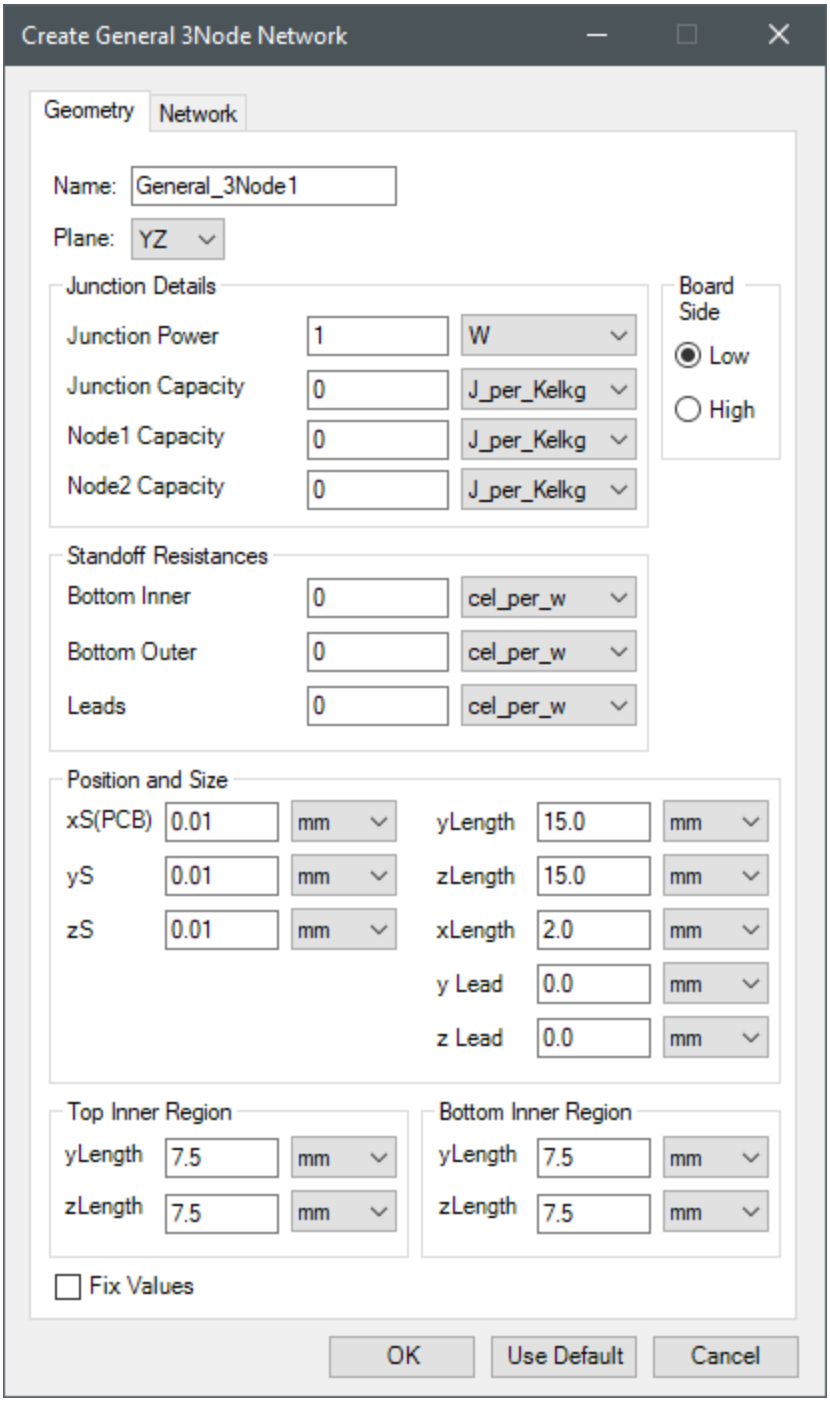

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        # this must be in the same order as the UI from top to
bottom
         ]
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/General_3Node_Network", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/General_3Node_Network", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/General_3Node_Network", data)
```

Packages - Network Model - General_3Node_Network

Packages - Network Model - General_3Node_Network enables you to create geometry and boundary conditions to represent a general 3 node network.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

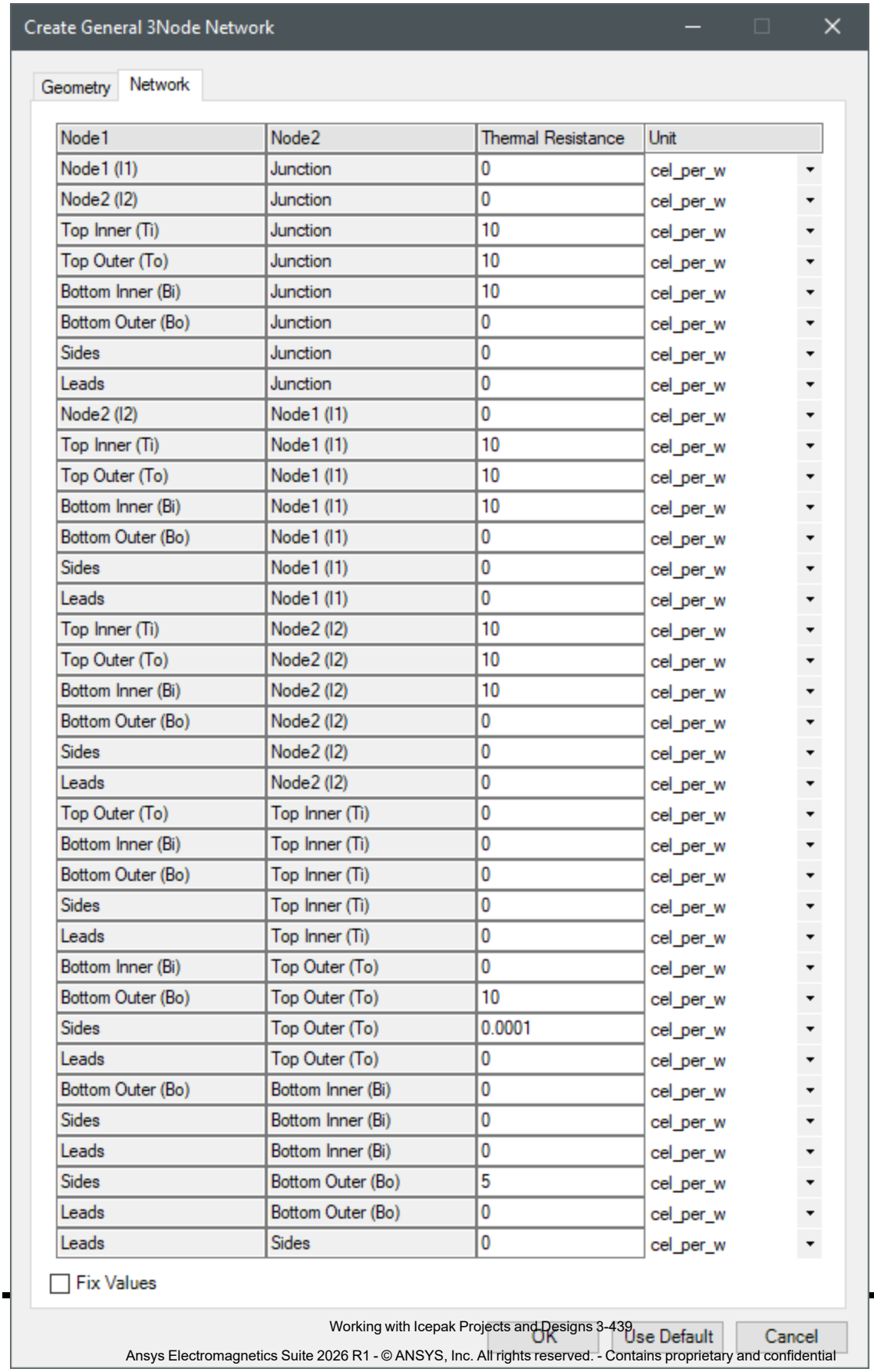


Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane	**Plane** specifies the plane on which to create the package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Junction Details	Under **Junction Details**, specify the junction power and the capacity of the junction and nodes.
Standoff Resistances	Under **Standoff Resistances**, specify the resistance of the bottom, inner, and lead standoff resistances.
Board Side	Under **Board Side**, specify which side of the package the board is located.
Position and Size	Under **Position and Size**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**. Also specify the length of the package and leads in each coordinate direction.
Top Inner Region	Under **Top Inner Region**, specify the length of the inner region on the top side of the package.
Bottom Inner Region	Under **Bottom Inner Region**, specify the length of the inner region on the bottom side of the package.

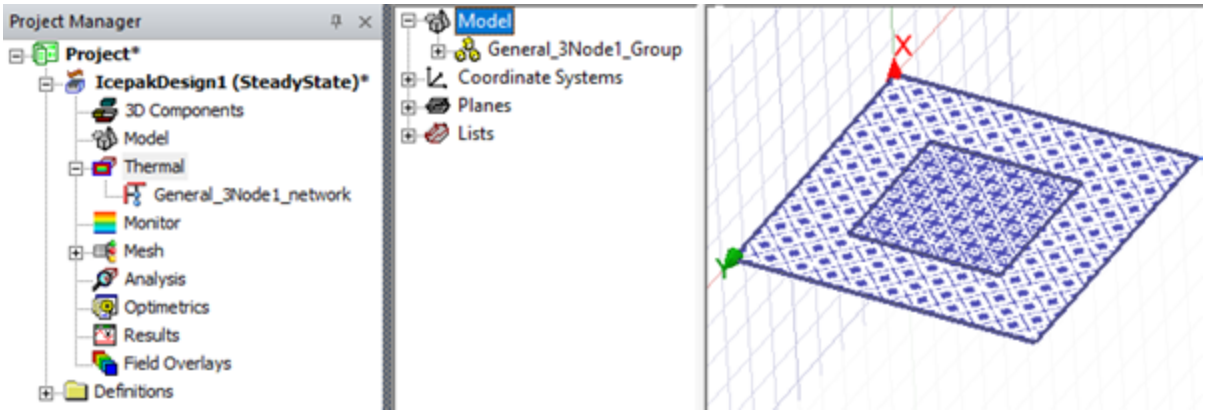
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Node1	Node2	Thermal Resistance	Unit
Node1 (I1)	Junction	0	cel_per_w
Node2 (I2)	Junction	0	cel_per_w
Top Inner (Ti)	Junction	10	cel_per_w
Top Outer (To)	Junction	10	cel_per_w
Bottom Inner (Bi)	Junction	10	cel_per_w
Bottom Outer (Bo)	Junction	0	cel_per_w
Sides	Junction	0	cel_per_w
Leads	Junction	0	cel_per_w
Node2 (I2)	Node1 (I1)	0	cel_per_w
Top Inner (Ti)	Node1 (I1)	10	cel_per_w
Top Outer (To)	Node1 (I1)	10	cel_per_w
Bottom Inner (Bi)	Node1 (I1)	10	cel_per_w
Bottom Outer (Bo)	Node1 (I1)	0	cel_per_w
Sides	Node1 (I1)	0	cel_per_w
Leads	Node1 (I1)	0	cel_per_w
Top Inner (Ti)	Node2 (I2)	10	cel_per_w
Top Outer (To)	Node2 (I2)	10	cel_per_w
Bottom Inner (Bi)	Node2 (I2)	10	cel_per_w
Bottom Outer (Bo)	Node2 (I2)	0	cel_per_w
Sides	Node2 (I2)	0	cel_per_w
Leads	Node2 (I2)	0	cel_per_w
Top Outer (To)	Top Inner (Ti)	0	cel_per_w
Bottom Inner (Bi)	Top Inner (Ti)	0	cel_per_w
Bottom Outer (Bo)	Top Inner (Ti)	0	cel_per_w
Sides	Top Inner (Ti)	0	cel_per_w
Leads	Top Inner (Ti)	0	cel_per_w
Bottom Inner (Bi)	Top Outer (To)	0	cel_per_w
Bottom Outer (Bo)	Top Outer (To)	10	cel_per_w
Sides	Top Outer (To)	0.0001	cel_per_w
Leads	Top Outer (To)	0	cel_per_w
Bottom Outer (Bo)	Bottom Inner (Bi)	0	cel_per_w
Sides	Bottom Inner (Bi)	0	cel_per_w
Leads	Bottom Inner (Bi)	0	cel_per_w
Sides	Bottom Outer (Bo)	5	cel_per_w
Leads	Bottom Outer (Bo)	0	cel_per_w
Leads	Sides	0	cel_per_w

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network	On the **Network** tab, enter the thermal resistances for the network nodes.



Iron Python Script – *GeneralNetwork.py*:

The following is the toolkit's scripting information.

```
scriptInput = ["ng"]
scriptInput.append("Name:GeneralNetwork1")
scriptInput.append("Plane:XY")
scriptInput.append("length_unit:meter")
scriptInput.append("power_unit:W")
scriptInput.append("capacity_unit:J_per_Kelkg")
scriptInput.append("resistance_unit:cel_per_w")
scriptInput.append("xS:0.1")
scriptInput.append("yS:0.1")
scriptInput.append("zS:0.1")
scriptInput.append("BoardSide:Low")
scriptInput.append("xL:0.015")
scriptInput.append("yL:0.015")
scriptInput.append("zL:0.002")
scriptInput.append("LeadLength1:0.001")
scriptInput.append("LeadLength2:0.001")
scriptInput.append("BoardMaterial:Al-Extruded")
scriptInput.append("TopLength1:0.0075")
scriptInput.append("TopLength2:0.0075")
scriptInput.append("BottomLength1:0.0075")
scriptInput.append("BottomLength2:0.0075")
scriptInput.append("BottomOuterStandOff:1.0")
scriptInput.append("BottomInnerStandOff:1.0")
scriptInput.append("LeadsStandOff:1.0")
scriptInput.append("JunctionPower:1.0")
scriptInput.append("JunctionCapacity:0.0")
```

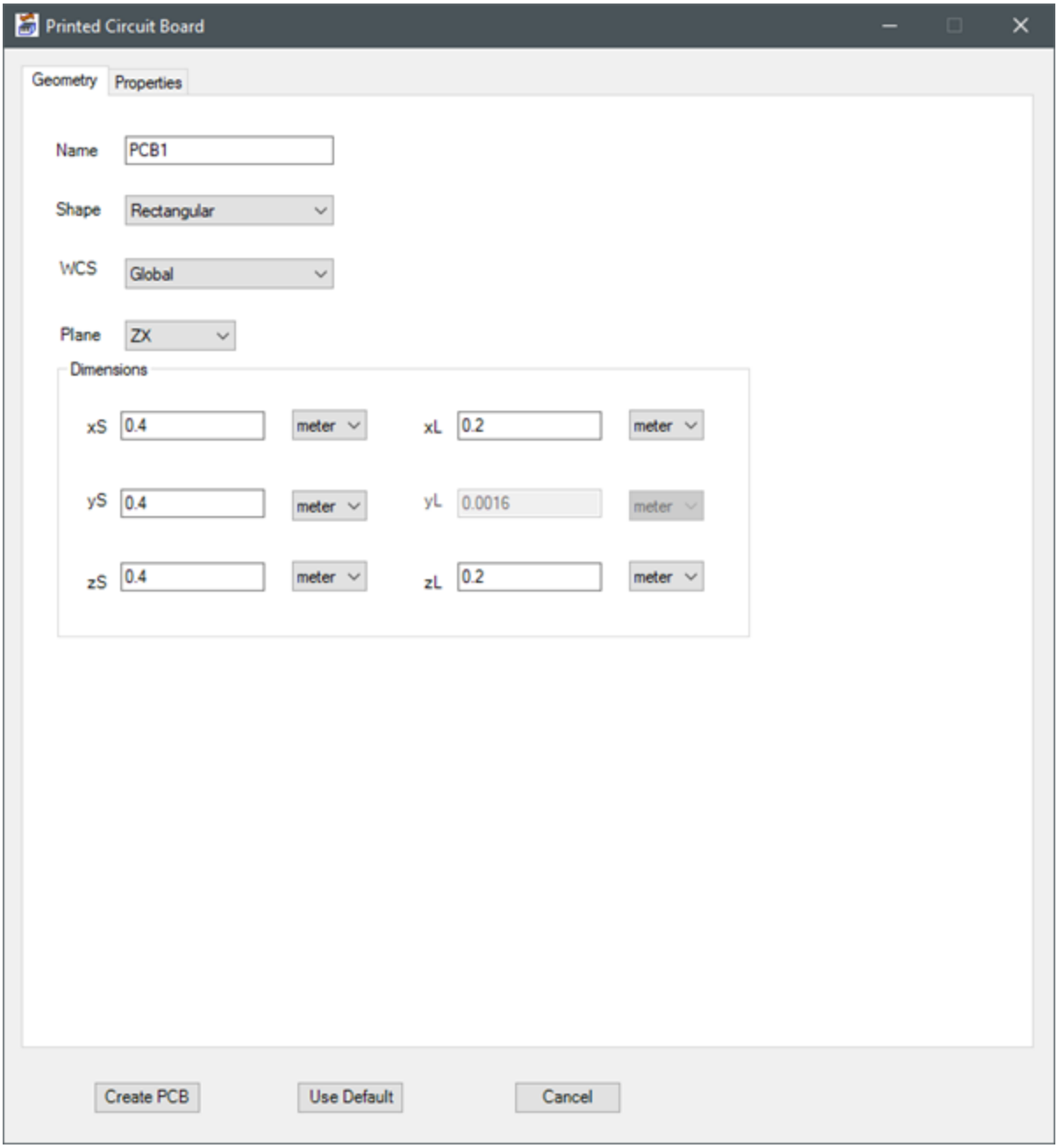

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 scriptInput.append("Links:
[10,10,10,0,0,0,0,0,0,0,0,10,0.001,0,10,0,0,5,0,0]")
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/GeneralNetwork", [])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/GeneralNetwork", ["ng"])
 oDesign.RunToolkit("SysLib", "Geometry/Packages - Network
Models/GeneralNetwork", scriptInput)
```

Printed Circuit Board (PCB)

PCB enables you to create geometry to represent an printed circuit board.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

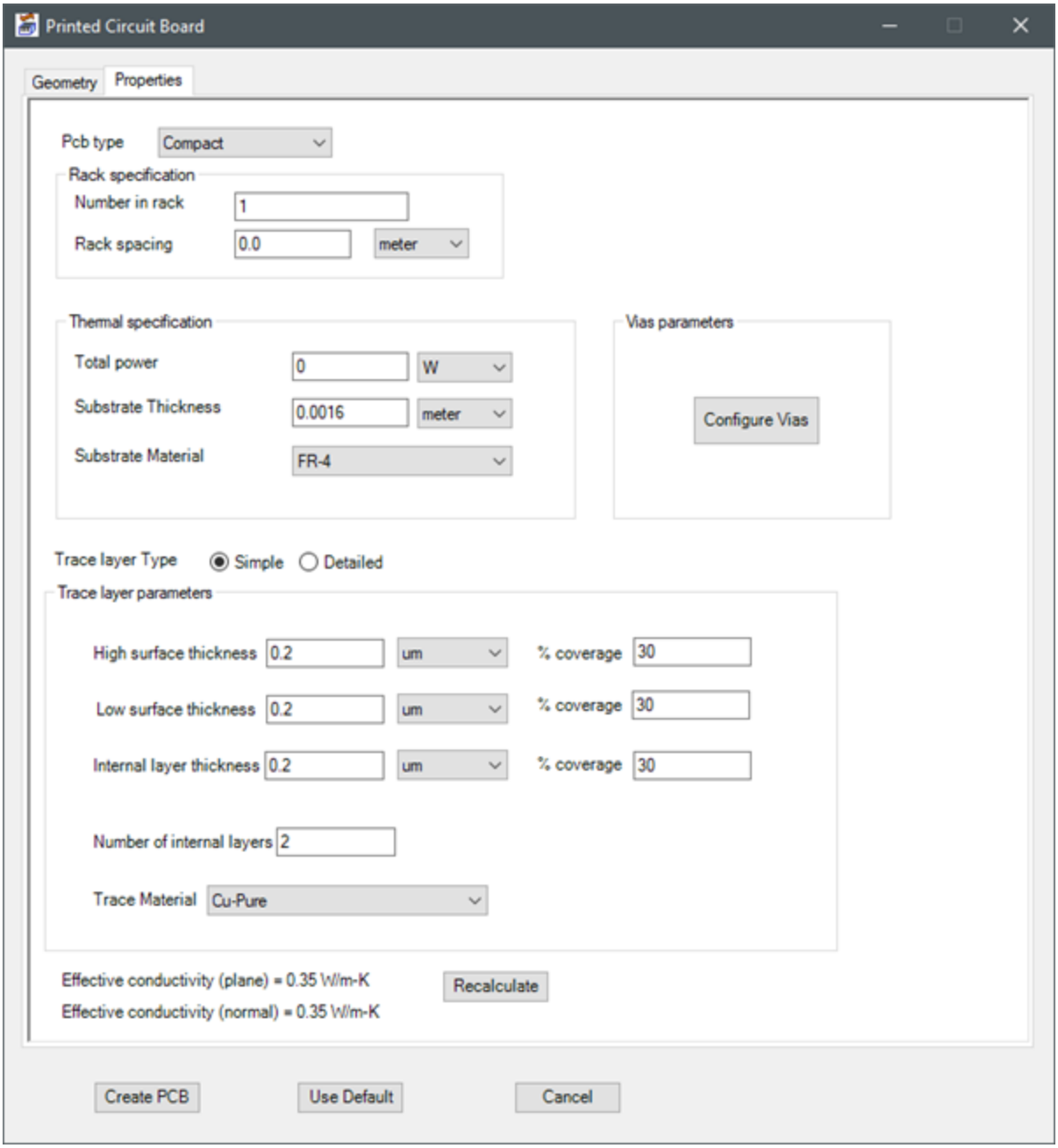


Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Shape	Specify whether the PCB is rectangular or polygonal.
Plane	Specify on which plane the PCB lies.
WCS	Specify on which coordinate system the PCB is oriented.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dimensions	For rectangular PCBs, define the start position relative to center and the length in the x, y, and z directions. For polygonal PCBs, specify the coordinates of the vertices (for example, vert 1, vert 2, vert 3, etc.) of the polygon. You can add and remove vertices using the **Add** and **Rem** buttons.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

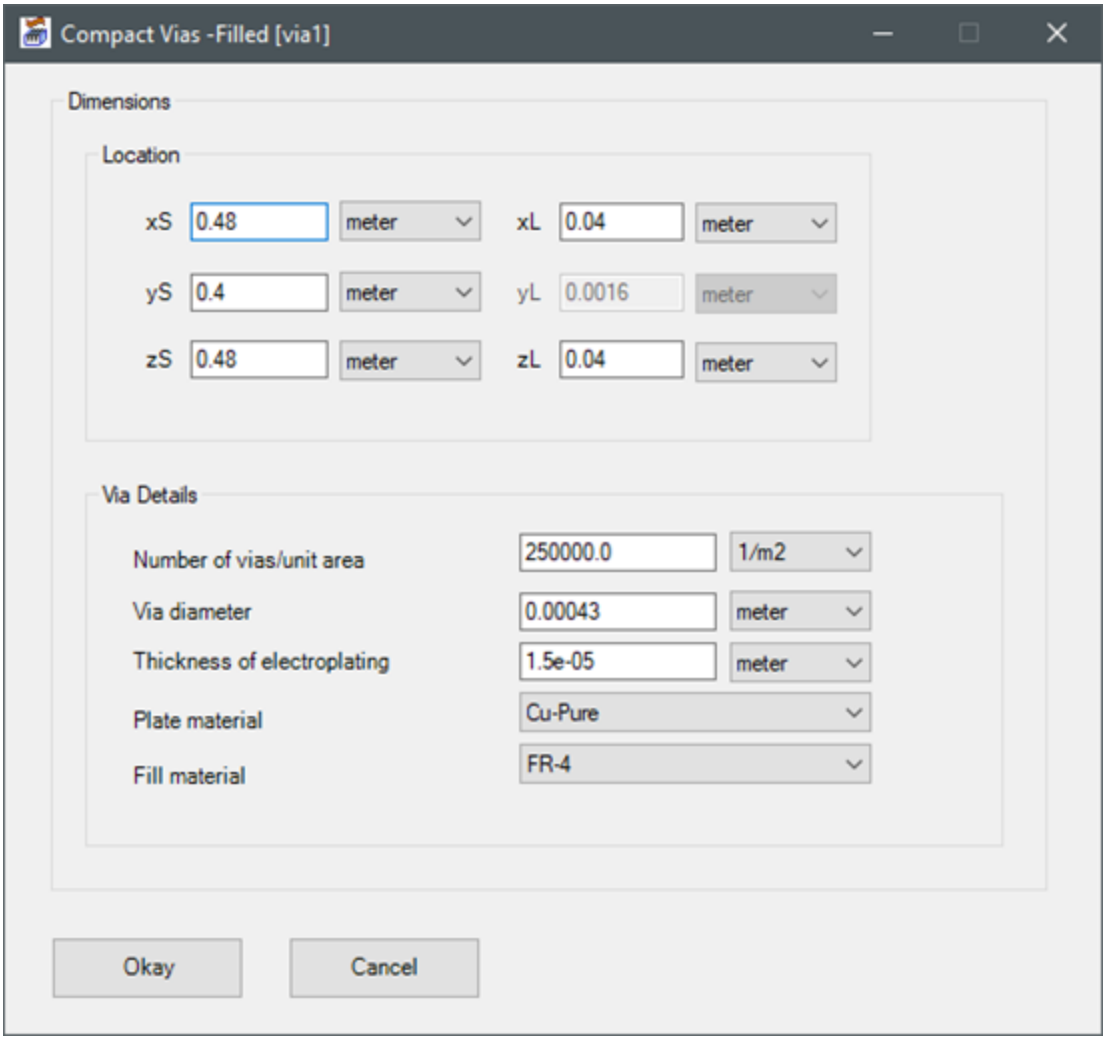


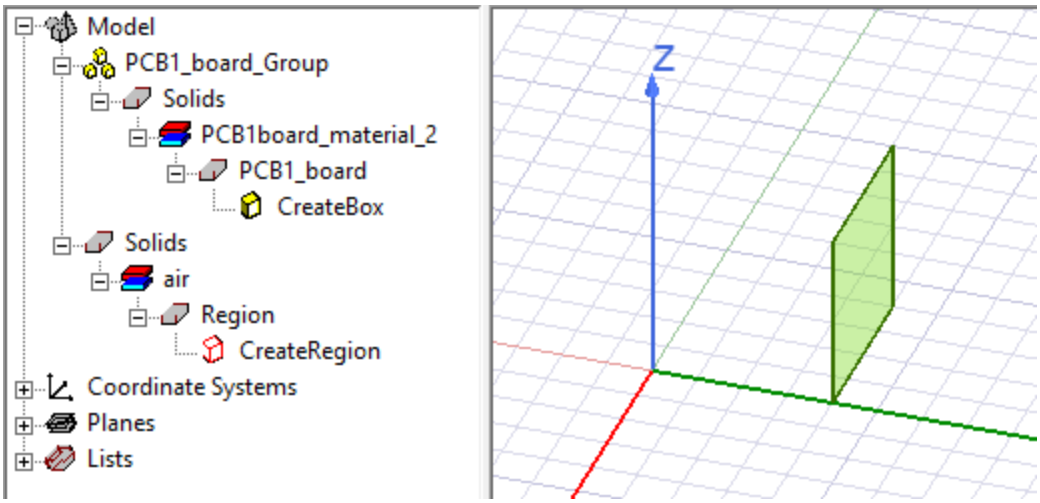
PCB type	Specify whether the PCB is compact or detailed.
Rack specification	Specify the number of boards in the rack of PCBs. If you specify more than one board, you must also specify the **Rack spacing** between the boards. If you specify a positive value for the **Rack spacing**, the boards will be generated in the positive direction of the axis normal to the plane of the PCB. If you specify a negative value for the **Rack**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	spacing, the boards will be generated in the negative direction of the axis normal to the plane of the PCB. By default the spacing is set to zero.
Thermal specification	Specify a **Total Power** for the PCB. Also, specify the **Substrate Thickness** and then specify the **Substrate Material** to be used for the PCB.
Vias parameters	Click **Configure Vias** to display the **Configure vias** panel. Click **Add** to add as many vias as needed. Click **Clear** to remove all vias. Click **Edit** to edit a row. Click the delete button to delete a row. If you delete a row, the rows below it shift up and are re-numbered sequentially.
Compact Vias-Filled dialog	Specify the **Location** of the vias. Enter the **Via Details** and click **Okay**. See the image below.
Trace layer Type = Simple	
Trace layer parameters	For simple trace layers, specify the **High surface thickness**, **Low surface thickness**, and **Internal layer thickness** and then specify the **% coverage** for each under **Trace layer parameters**. Finally, specify the **Number of internal layers** and select the **Trace Material**.
Trace layer Type = Detailed	
Trace layer parameters	For detailed trace layers, you can create additional layers by clicking the **Add layer** button. Specify the **Layer thickness**, **% coverage** and **Layer Material** for each layer under Trace layer parameters. Click **Delete layer** to remove the last created layer. Finally, specify the **Layer Material**.
Recalculate	Click **Recalculate** to compute the effective conductivities (plane and normal).
Create PCB	Click **Create PCB** to insert the PCB in the Icepak design.
Use Default	Click **Use Default** to revert any changes you made to the default values.
Cancel	Click **Cancel** to close the dialog without creating a PCB.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iron Python Script – *PCB.py*:

The following is the toolkit's scripting information.

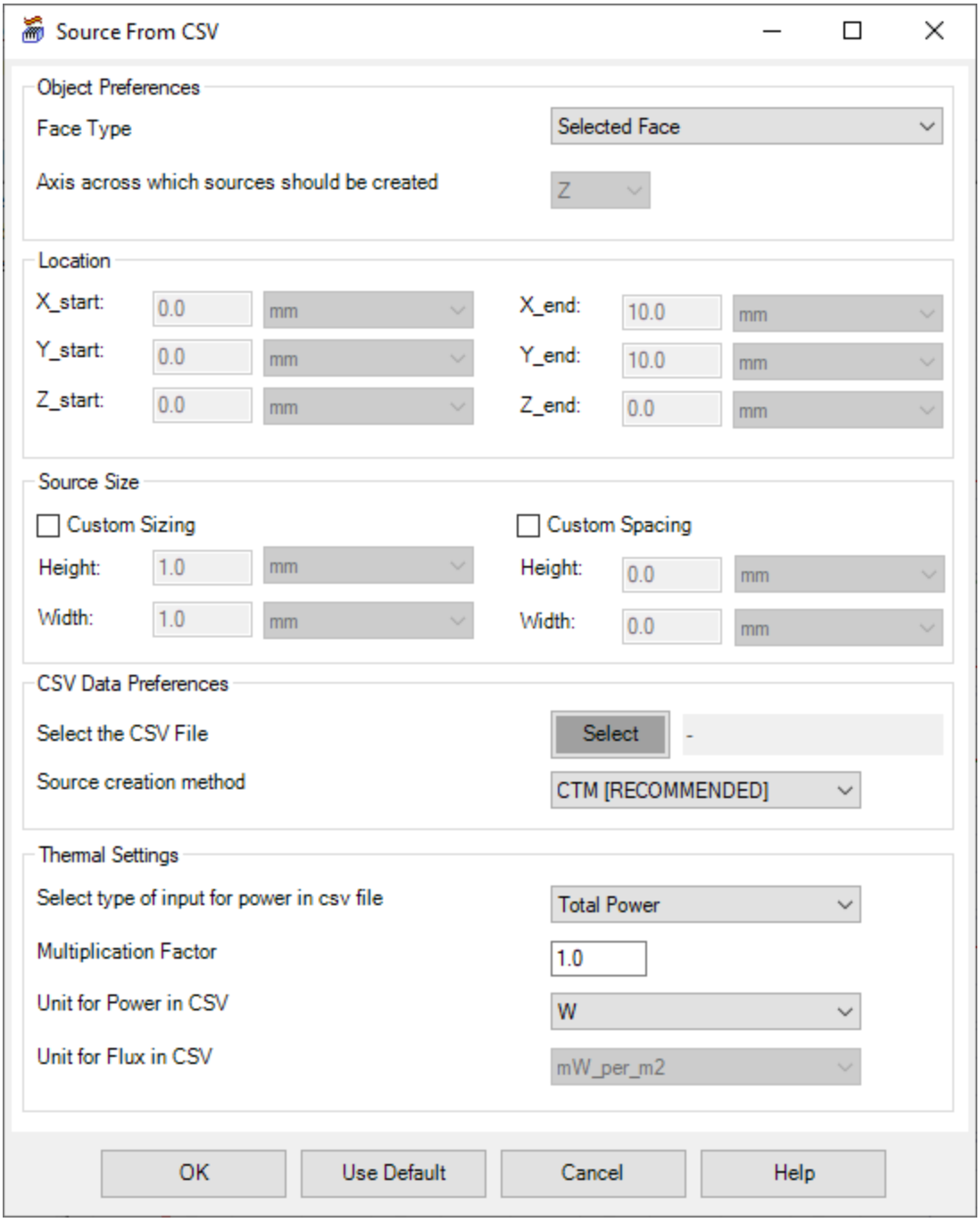
```
 data = ["ng",
         "Name:PCB",
         'Length_unit:meter',
         'Power_unit:W',
         'via_area_unit:1/m2',
         'Plane:ZX',
         'Shape:Polygon',
         'WCS:Global',
         'Rectangle_dimensions:[0.4,0.4,0.4,0.2,0.0016,0.2]',
#xs,ys,zs,xl,yl,zl
            'Polygon_vertices:[[0.4,0.4,0.4],[0.6,0.4,0.4],
[0.6,0.4,0.6],[0.4,0.4,0.6]]',
              #x1,y1,z1
         'Pcbtype:Compact',
         'Numberinracks:3',
         'Rackspacing:0.2',
         'TotalPower:1',
         'Substratethickness:0.0016',
         'SubstrateMaterial:FR-4',
         'TraceMaterial:Cu-Pure',
         'Tracelayertype:Detailed',
         'HighSurfacethickness:1e-6',
         'LowSurfacethickness:1e-6',
         'Internallayerthickness:1e-6',
         'Numberofinternallayers:2',
         'coverage_high:30',
         'coverage_low:30',
         'coverage_internal:30',
            'Detailed_params:[[1e-6,30,"Cu-Pure"],[1e-6,30,"Cu-
Pure"]]', #layer_thickness,
            layer_coverage,layer_material
            'via_params:[[0.48,0.4,0.48,0.04,0.0016,0.04,
250000,0.43e-3,15e-6,"Cu-Pure",
            "FR-4"],
[0.48,0.4,0.48,0.04,0.0016,0.04,250000,0.43e-3,15e-6,
            "Cu-Pure","FR-4"]]' #xs,ys,zs,xl,yl,zl,vias_
area,diameter,thickness,
            Plate_material,Fill_material
         ]
 oDesign.RunToolkit("SysLib", "Geometry/PCB_SCRIPT", [])
 oDesign.RunToolkit("SysLib", "Geometry/PCB_SCRIPT", ['ng'])
 oDesign.RunToolkit("SysLib", "Geometry/PCB_SCRIPT", data)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Source From CSV

Source From CSV allows you to create a source object from a comma-separated value file. The sources can be modeled as a CTM native component or source thermal boundary conditions. The CTM native component option is recommended. The Individual Sources option takes longer to process as individual source geometry are created and boundary conditions automatically assigned.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

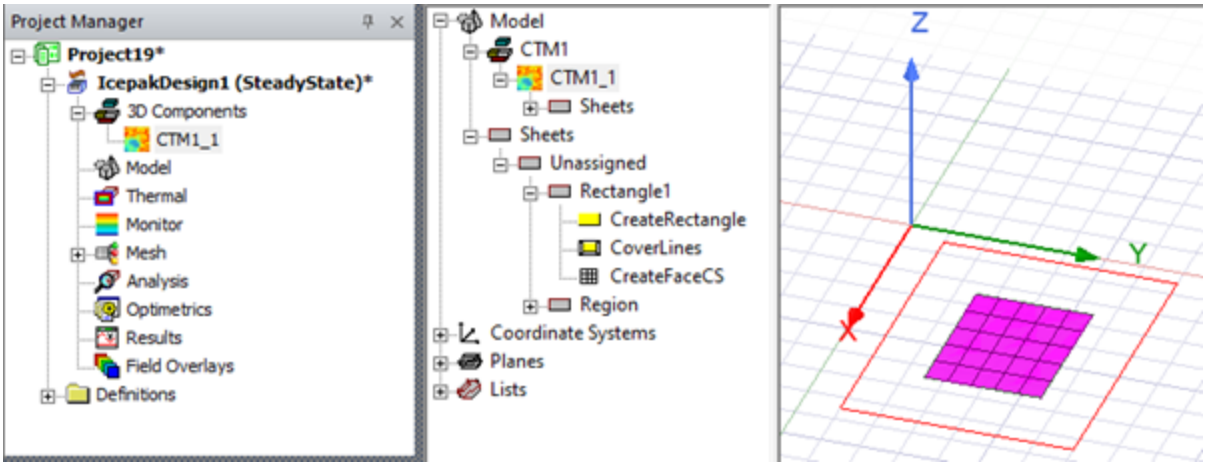


Object Preferences	
Face Type	Specify either **Selected Face** if you selected a piece of sheet geometry or **New sheet** if you want to specify **Location** for a new piece of sheet geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axis across which sources should be created	Specify the **Axis across which sources should be created**.
Location	
X, **Y**, and **Z start**	If you selected **New sheet**, specify the X, Y, and Z coordinate start points for the sheet geometry.
X, **Y**, and **Z end**	If you selected **New sheet**, specify the X, Y, and Z coordinate end points for the sheet geometry.
Source Size	
Custom Sizing	If needed, select **Custom Sizing** and specify **Height** and **Width** values.
Custom Spacing	If needed, select **Custom Spacing** and specify **Height** and **Width** values.
CSV Data Preferences	
Select the CSV File	Click **Select** to open the **Open** dialog and browse to the CSV file location.
Source creation method	From the drop-down list, select **CTM**, **Dataset**, or **Individual Sources**. CTM creates a CTM native component. Dataset creates a design dataset. Individual Sources creates source boundary conditions,
Thermal Settings	
Select type of input for power in csv file	Select **Total Power** or **Surface Flux** for the type of power specified in the CSV file.
Multiplication Factor	Enter the multiplication factor to be applied to the power.
Unit for Power in CSV	If you selected **Total Power**, select the unit for the total power.
Unit for Flux in CSV	If you selected **Surface Flux**, select the unit for the surface flux.
OK, **Use Default**, **Cancel**, **Help**	Click **OK** to create the sources as specified. Click **Use Default** to restore the default settings. Click **Cancel** to close the **Source From CSV** dialog box without creating the sources. Click **Help** to open a dialog box with additional information about the toolkit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



CSV File

The following images are examples of a comma-separated value (.csv) file as displayed in a spreadsheet application and a plain text editor.

	A	B	C	D	E
1	0.3	0.3	0.3	0.3	0.3
2	0.3	0.3	0.3	0.3	0.3
3	0.3	0.3	0.3	0.3	0.3
4	0.3	0.3	0.3	0.3	0.3
5	0.3	0.3	0.3	0.3	0.3

```
1  0.3,0.3,0.3,0.3,0.3
2  0.3,0.3,0.3,0.3,0.3
3  0.3,0.3,0.3,0.3,0.3
4  0.3,0.3,0.3,0.3,0.3
5  0.3,0.3,0.3,0.3,0.3
```

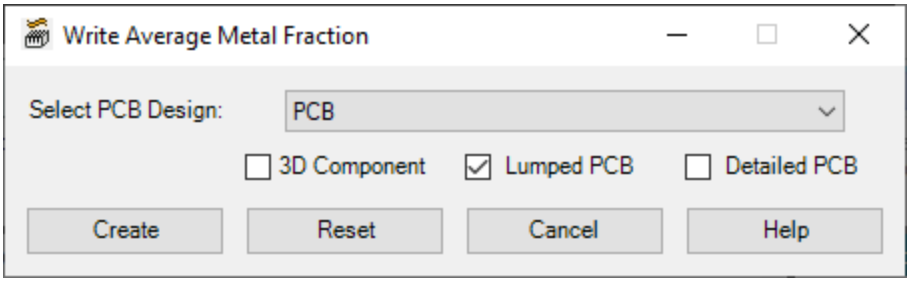
Write Average Metal Fraction

Write Average Metal Fraction allows you to calculate the average metal fraction for each layer of a PCB linked to an HFSS 3D Layout design. It also creates a PCB in a new project using one of three methods: 3D Component, Lumped PCB, or Detailed PCB.

The following are prerequisites to access and run the Write Average Metal Fraction toolkit:

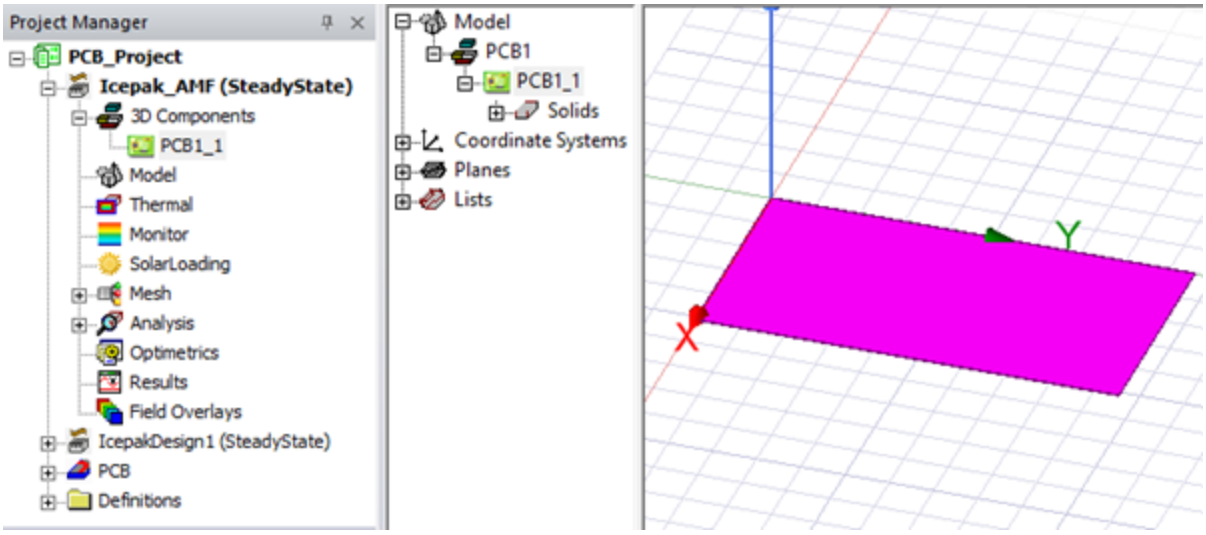
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The project must contain at least one Icepak design.
- An Icepak design must be the active design.
- HFSS 3D Layout design must contain valid ECAD.
- Thermal properties must be defined in the layer materials specifications in HFSS 3D Layout.



Select PCB Design	From the **Select PCB Design** drop-down list, select the PCB for which you want to calculate average metal fractions and generate the new project.
3D Component	Select **3D Component** to create an Icepak PCB component with a full, layer-based conductivity map.
Lumped PCB	Select **Lumped PCB** to create a single block PCB with normal-to-plane and in-plan orthotropic conductivities.
Detailed PCB	Select **Detailed PCB** to model each layer individually with averaged isotropic conductivity.
Create	Click **Create** to calculate the average metal fractions and generate the new project with the specified PCB.
Reset	Click **Reset** to clear the **Select PCB Design** selection.
Cancel	Click **Cancel** to close the toolkit.
Help	Click **Help** to open the **Help** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

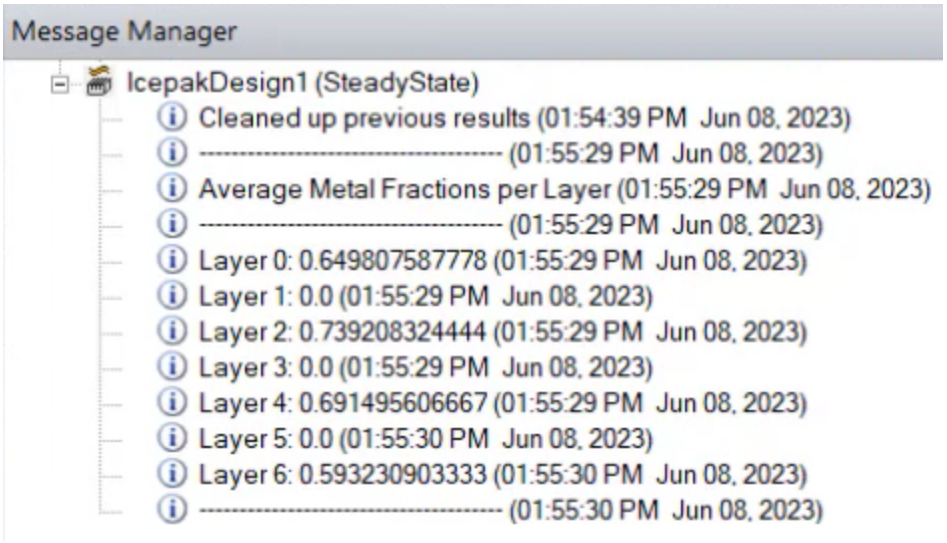


The average metal fractions are displayed in a dialog box and the message manager.

```
---------------------------------------
Lumped conductivity of the PCB:
 ---------------------------------------
In-plane Lumped PCB Conductivity : 56.5906   W/m-k
Normal Lumped PCB Conductivity   : 0.44318   W/m-k
---------------------------------------

Average Metal Fractions:
--------------------------
Layer 1: 0.64981
Layer 2: 0.0
Layer 3: 0.73921
Layer 4: 0.0
Layer 5: 0.6915
Layer 6: 0.0
Layer 7: 0.59323
--------------------------
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Icepak Modeling Toolkit

Icepak has the following toolkits that assist in creating a model.

- Blower
- DTM - Thermostat for Source Control
- IC Packages - Extract Delphi Network
- IC Packages - JEDEC
- IC Packages - Extract JB JC
- LPV ROM Parametric Setup
- LTI ROM Parametric Setup
- Rmatrix_AEDT
- Solar Flux Calculator
- Static ROM
- STM Network Extraction
- Thermoelectric Cooler - Create TEC

Blower

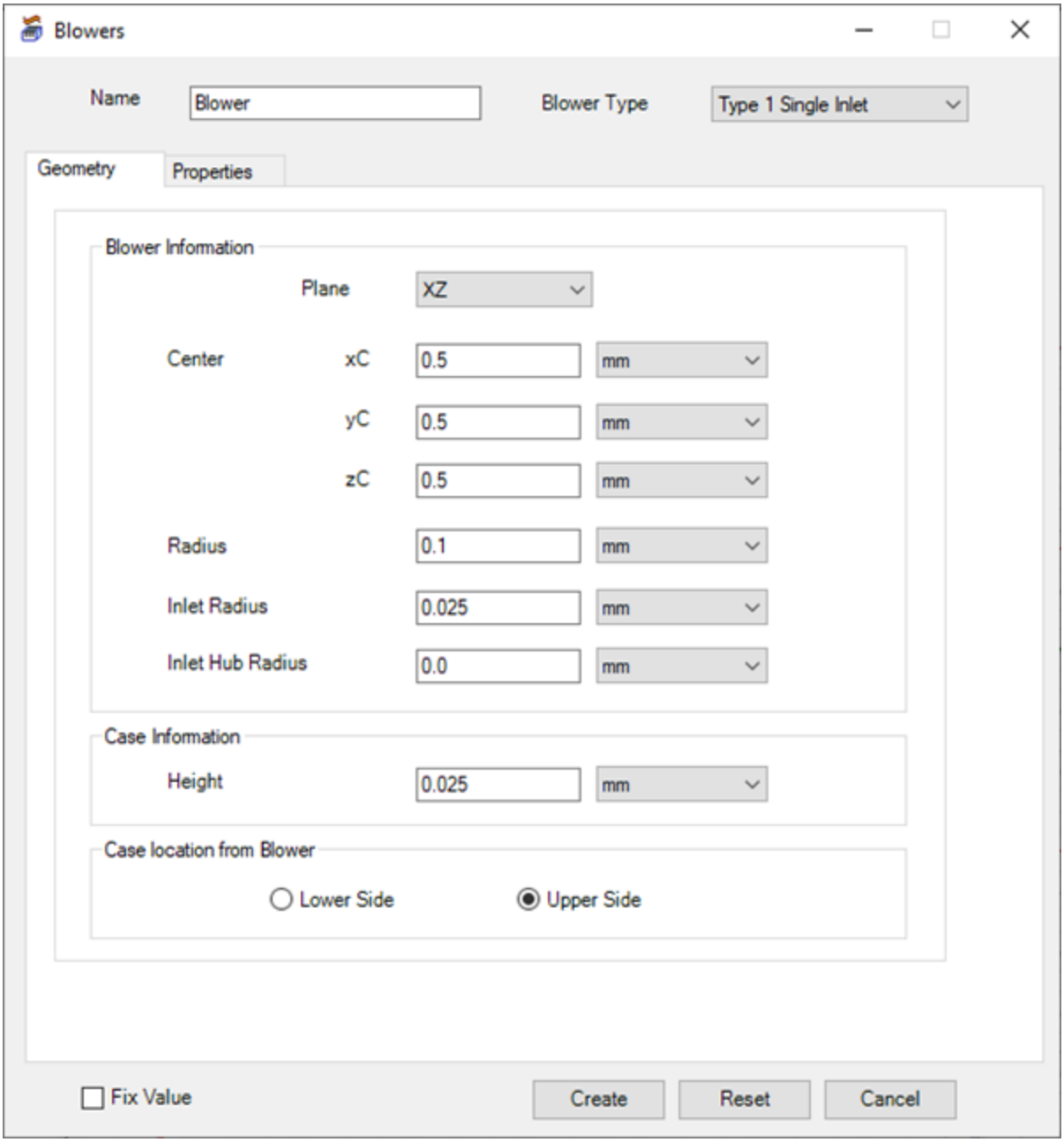
Blower enables you to create blower object geometry apply the blower boundary condition. Specify the blower **Name** and select from the following blower types:

- Type 1 Single Inlet
- Type 1 Dual Inlet

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Type 2 Single Inlet
- Type 2 Dual Inlet

Type 1 Blowers

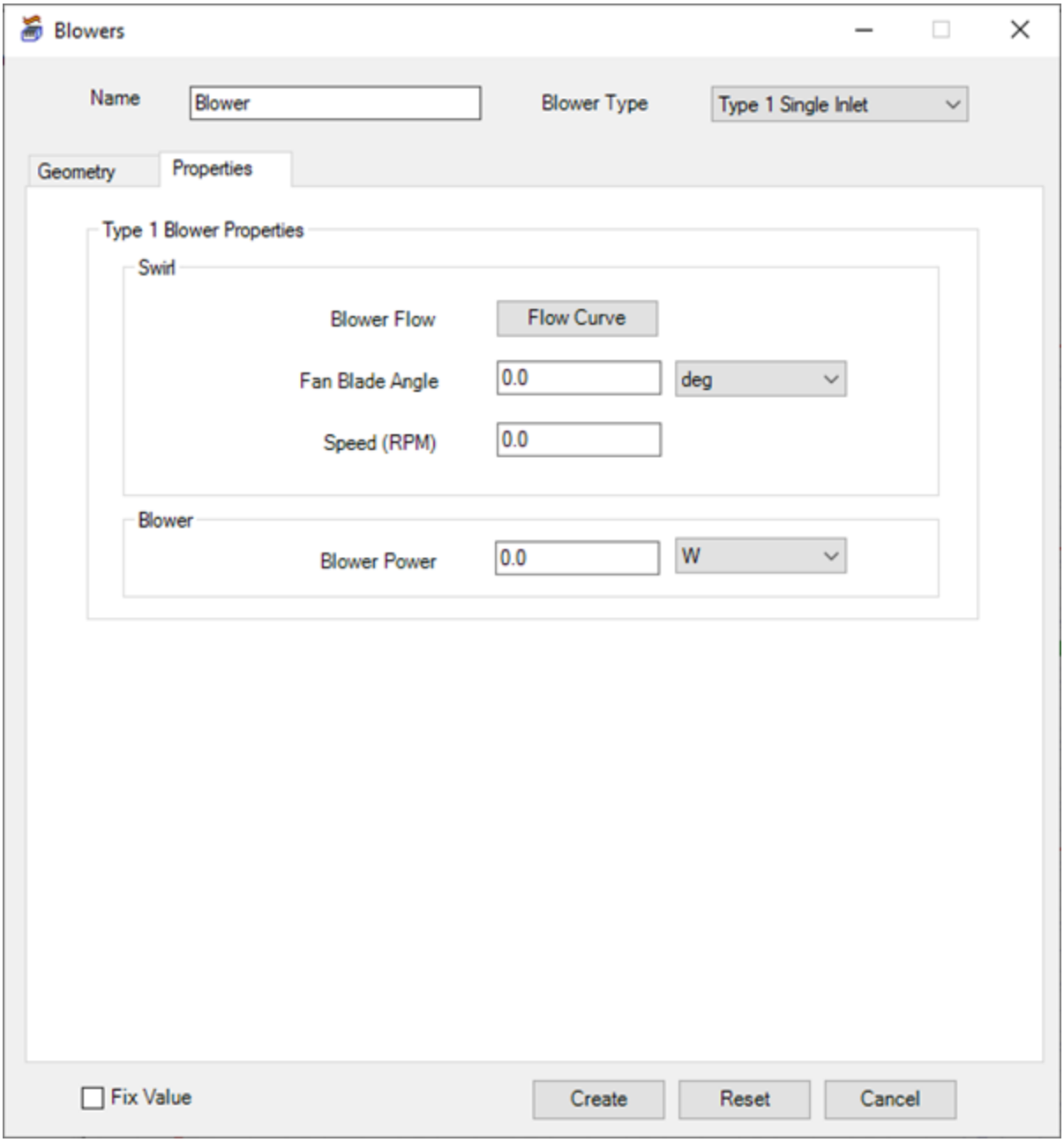


Blower Information

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane	**Plane** specifies which plane the blower will exist on.
Center	**Center** specifies the **xC**, **yC**, and **zC** coordinates to define the center location of the blower.
Radius	**Radius** specifies the length of the blower's radius.
Inlet Radius	**Inlet Radius** specifies the length of the blower inlet's radius.
Inlet Hub Radius	**Inlet Hub Radius** specifies the length of the blower inlet hub's radius.
Case Information	
Height	**Height** specifies the height of the blower's case.
Case Location From Blower	
Lower Side/Upper Side	**Lower Side** or **Upper Side** specifies whether the blower case exists on the lower or upper side of the object.
Create/Reset/Cancel	Click **Create** to generate the blower geometry and boundary condition. Click **Reset** to restore the default values. Click **Cancel** to close the **Blowers** dialog box without creating the blower.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



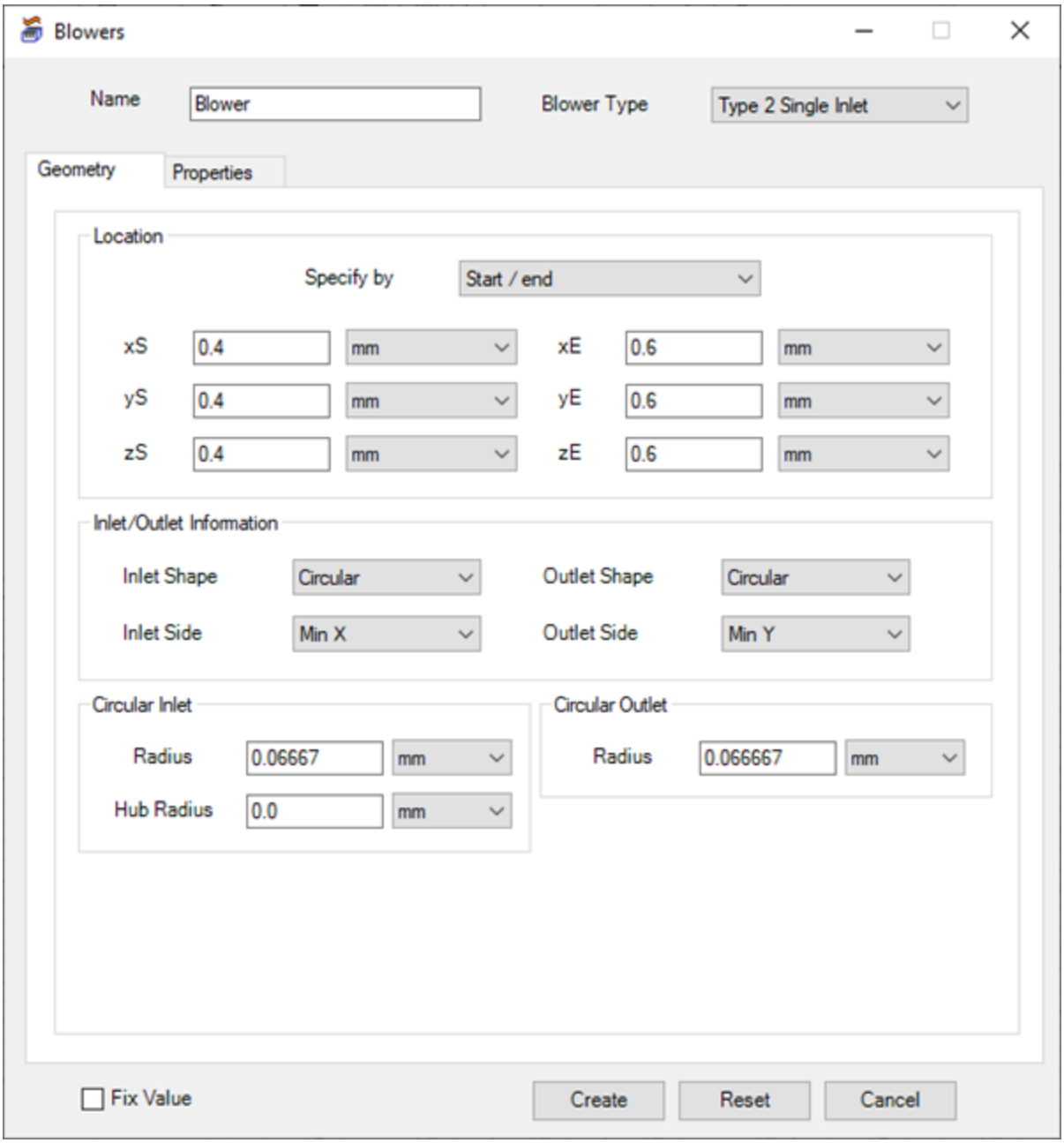
Swirl	
Blower Flow	**Blower Flow** specifies the flow based on a specified flow curve. Click **Flow Curve** to open the **Edit Dataset** dialog box and enter volume flow and pressure data to create the flow curve.
Fan Blade Angle	**Fan Blade Angle** specifies the angle of the blower's fan blade for a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	type 1 blower.
Speed (RPM)	**Speed (RPM)** specifies the revolutions per minute of the blower fan for a type 1 blower.
Blower	
Blower Power	**Blower Power** specifies the total power dissipated by the blower.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type 2 Blowers

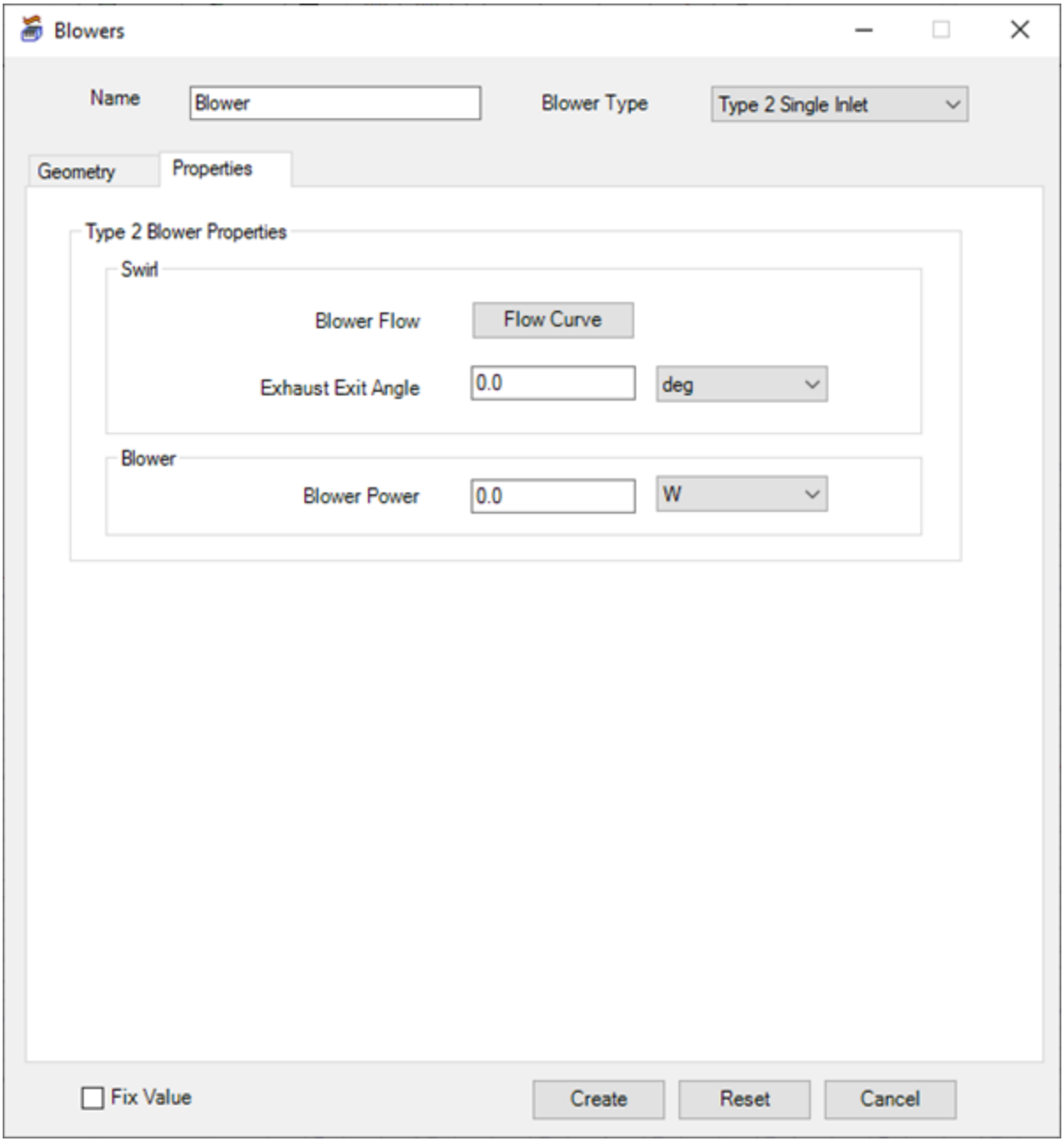


Location	
Specify by	**Specify by** specifies whether to define the blower geometry dimensions by start and end coordinates (**Start/end**) or by a starting coordinate and length values (**Start/length**)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inlet/Outlet Information	
Inlet Shape/Outlet Shape	Under **Inlet/Outlet Information**, specify the shape of the blower's inlet and outlet. Also, specify the side of the inlet and outlet.
Inlet Side/Outlet Side	Under **Inlet/Outlet Information**, specify the side of the blower's inlet and outlet.
Inlet (Circular)	
Radius	**Radius** specifies the length of the blower's inlet radius.
Hub Radius	**Hub Radius** specifies the length of the blower inlet hub's radius.
Inlet (Rectangular)	
Size	**Size** specifies values for the length and width of the opening for the blower's inlet.
Hub Radius	**Hub Radius** specifies the length of the blower inlet hub's radius.
Offset	**Offset** specifies the offset distances from the center of the face for the blower's inlet.
Outlet (Circular)	
Radius	**Radius** specifies the length of the blower's outlet radius.
Outlet (Rectangular)	
Size	**Size** specifies values for the length and width of the opening for the blower's inlet.
Offset	**Offset** specifies the offset distances from the center of the face for the blower's outlet.

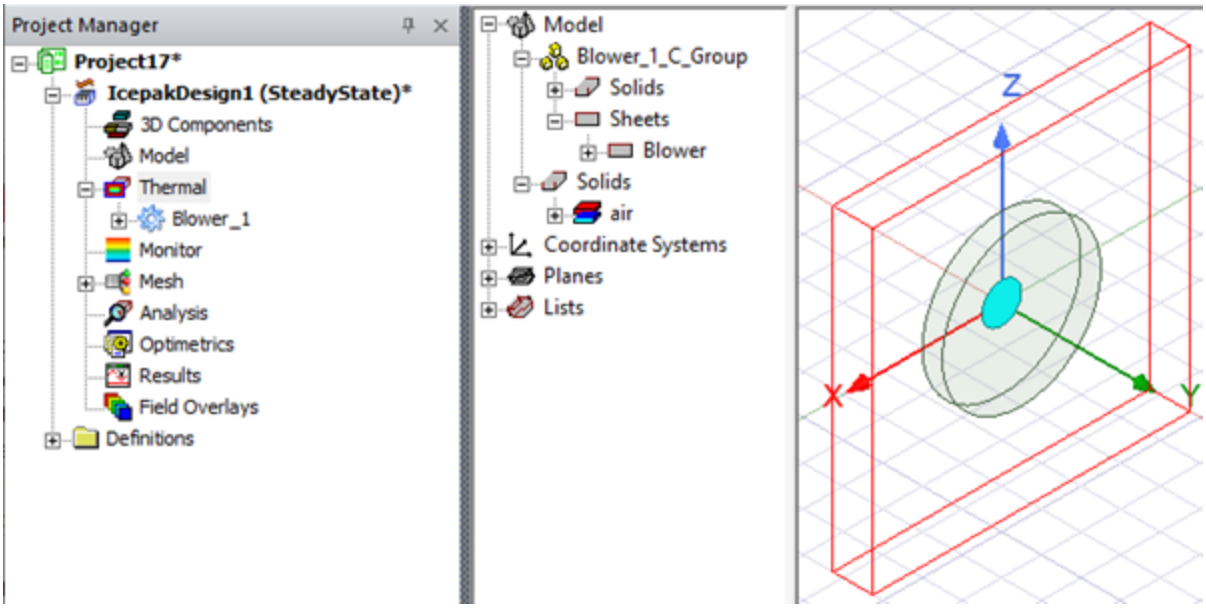
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Swirl	
Blower Flow	**Blower Flow** specifies the flow based on a specified flow curve. Click **Flow Curve** to open the **Edit Dataset** dialog box and enter volume flow and pressure data to create the flow curve.
Exhaust Exit Angle	**Exit Exhaust Angle** specifies the angle of the exhaust flow for a type

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

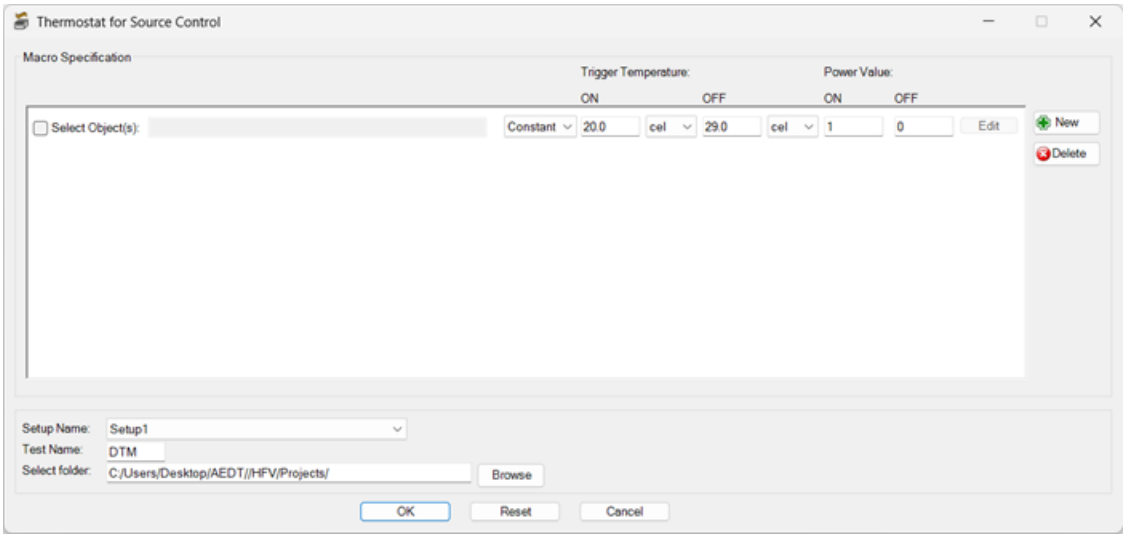
	2 blower.
Blower	
Blower Power	**Blower Power** specifies the total power dissipated by the blower.



DTM - Thermostat for Source Control

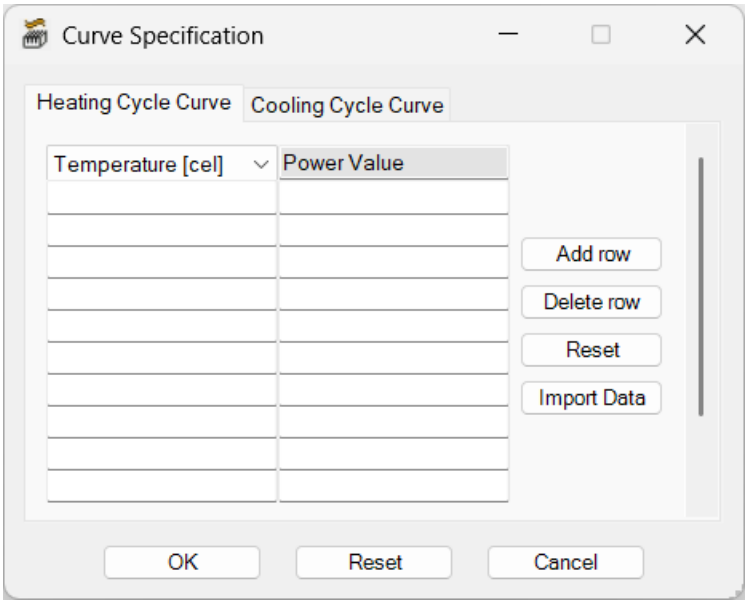
Thermostat for Source Control enables you to define temperature-dependent power in transient designs for selected 2D source geometry (sheets) and 3D block geometry by specifying trigger temperatures and power values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



New/Delete	Click **New** to add a row to define temperature-dependent source or block geometry. Click **Delete** to remove a row.
Selected Object(s)	Enable **Selected Object(s)** to open the **Select Object(s)** dialog box and select the source(s) and/or block(s) for which you want to define temperature-dependent power.
Constant/Varying	Select **Constant** or **Varying**. With **Constant** selected, enter **Trigger Temperature** and **Power Value** values for the selected object. With **Varying** selected, click **Edit** to define multiple temperature and power value values to define heating and cooling cycles.
Trigger Temperature	Under **Trigger Temperature**, enter the **ON** and **OFF** trigger temperatures. When the objects' temperature is between the specified temperatures, the associated power value is applied.
Power Value	Under **Power Value**, enter the **ON** and **OFF** power values to be applied to the selected objects' power when the objects' temperature is between the specified trigger temperatures.
Edit	With **Varying** selected, click **Edit** to specify heating and cooling cycle curves by entering **Temperature** and **Power Value** values in the **Curve Specification** dialog box.
Setup Name	In the **Setup Name** drop-down list, select the Icepak solve setup.
Test Name	In the **Test Name** field, enter a file name for the control program script.
Select folder	Click **Browse** to select a folder in which to save the control program script.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Curve Specification** dialog box, the **Temperature** and **Power Value** values control how much power is applied during different stages of the simulation. For example, if the ON temperature is 30 degrees Celsius and OFF temp is 40 degrees Celsius, you can create curves for finer control.

Heating Cycle Curve			**Cooling Cycle Curve**	
Temperature	**Power Value**		**Temperature**	**Power Value**
32	4		38	3
37	4.5		33	1
39	5		31	0

> **Note:**
>
> The heating cycle is applied after the first cooling cycle is executed.

In this case, until the temperature reaches 40 degrees, a constant ON power of 4 watts will be used. Once it reaches 40 degrees, the model applies values specified in cooling cycle. The values in cooling cycle are used until the temperature reaches 30 degrees. When the output temperature drops to 30 degrees, the heating cycle is used until the temperature reaches 40 degrees.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The power values are not interpolated as the temperature increases or decreases. For example, in the cooling cycle above, a power value of 1 watt will be used when the temperature is between 37 and 33 degrees.

IC Packages - Extract Delphi Network

> **Note:**
>
> The Extract Delphi Network toolkit uses PyAEDT functionality to run. If PyAEDT is not installed when first accessing the toolkit, a script runs to install the necessary components.

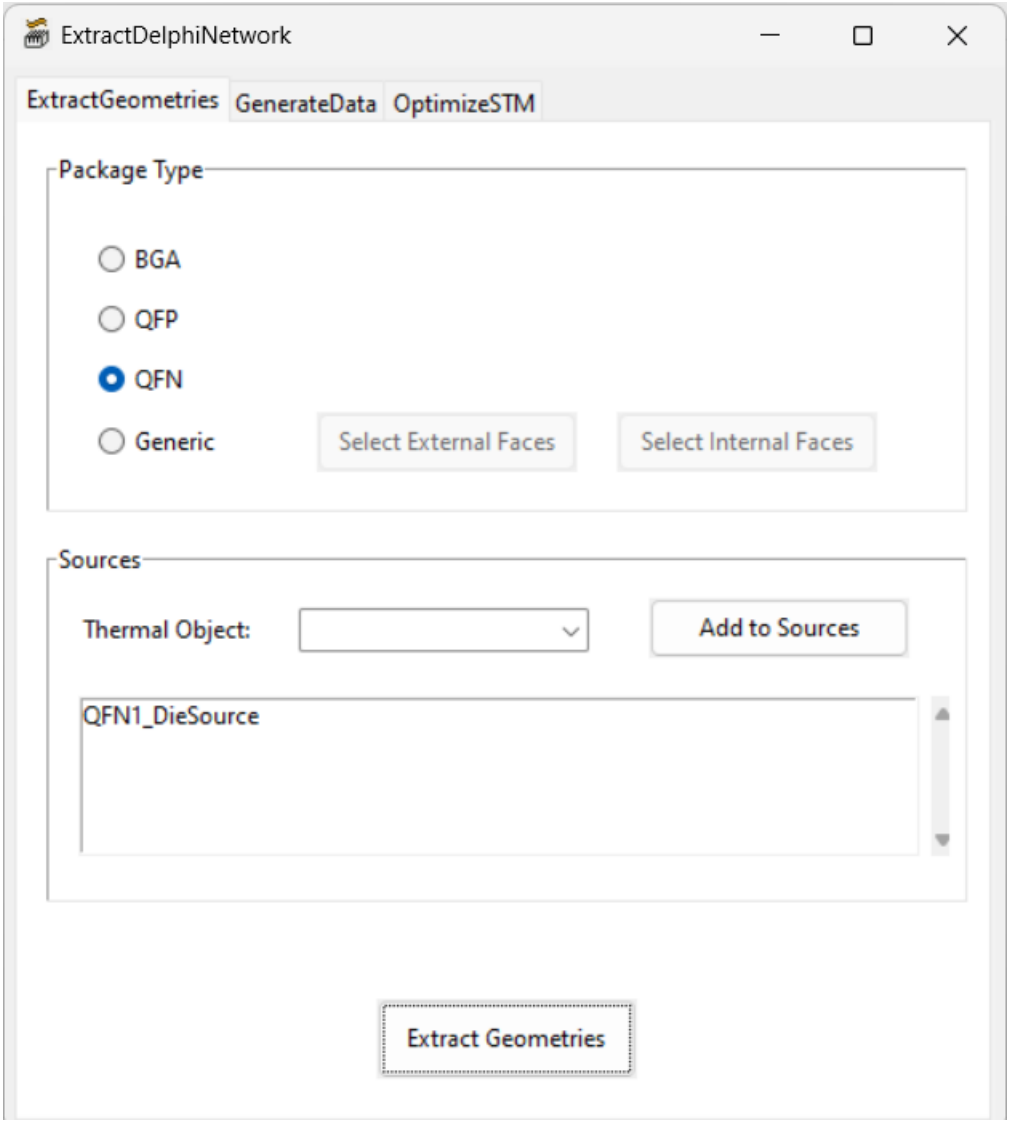
Extract Delphi Network creates, trains, and exports a Delphi-like Surrogate Thermal Model (STM) that is Boundary Condition Independent (BCI) and can be used to emulate the thermal behavior of the IC package.

> **Note:**
>
> See Getting Started with Icepak - Extract Delphi Network for a workflow related to this feature.

Extract Geometries Tab

The **Extract Geometries** tab controls the process of selecting specific geometric features from the Detailed Thermal Model (DTM). These features include external faces, internal faces, and heat sources, and they have two primary purposes: to inform the DTM simulations for data production and to fully define the STM's internal and external geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



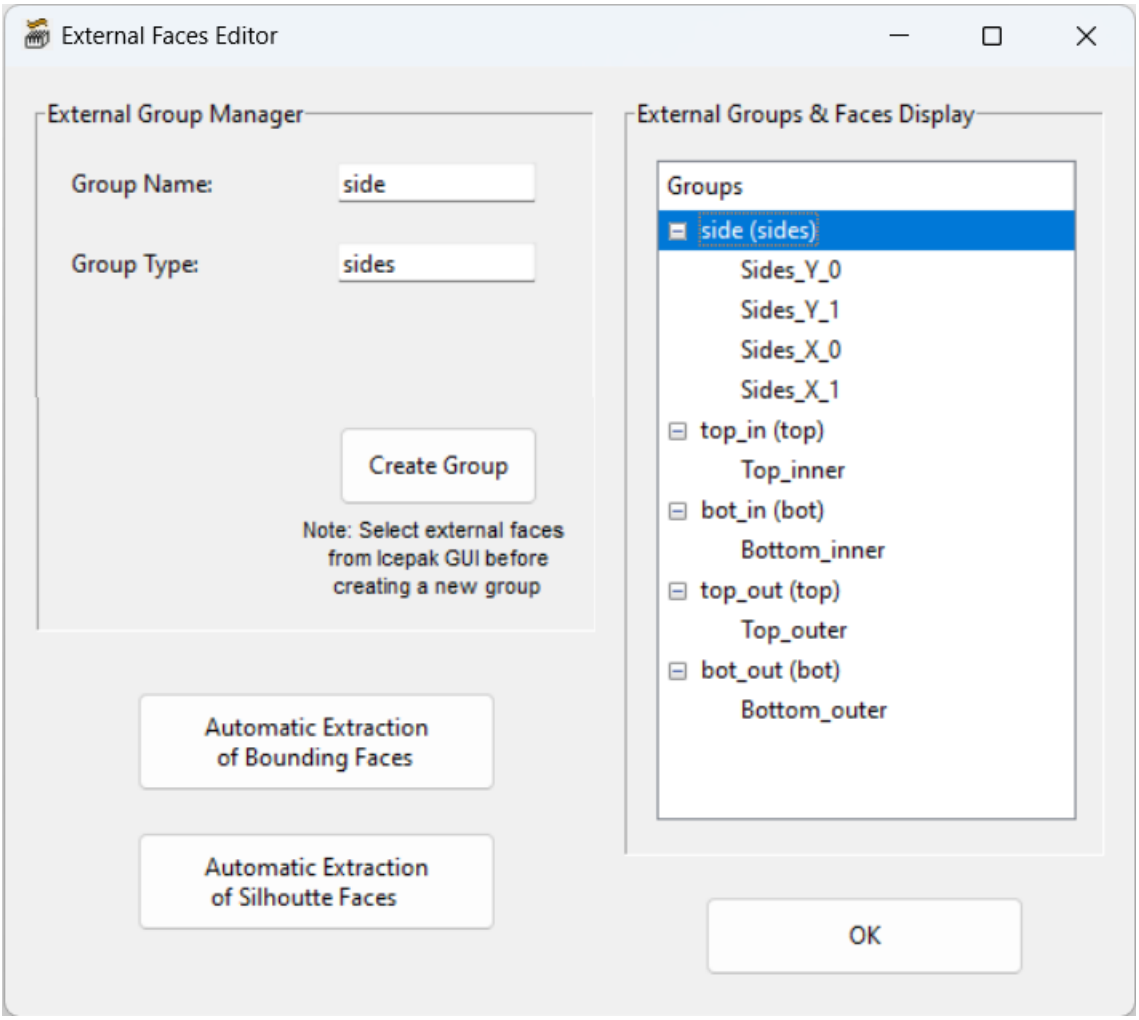
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Package Type	
Package Type	Under **Package Type**, select the package type for the delphi network extraction. Select **Generic** for packages that are not BGA, QFP, or QFN created using an **Icepak** > **Toolkits** > **Geometry** toolkit. For generic packages, click **Select External Faces** to export groups of external faces manually or automatically and **Select Internal Faces** to manually export groups of internal faces. See "External Faces Editor" below and "External Faces Editor" belowfor more information.
Sources	
Thermal Object **Add to Sources**	Sources are displayed in the list. If needed, select a thermal object and click **Add to Sources** to include additional active sources to the resulting network.
Extract Geometries	Click **Extract Geometries** to finalize the collection of external faces, internal faces, heat sources, and their related properties. When the process is complete, the **Extract Geometries** button appears green.

External Faces Editor

In the **External Faces Editor**, use the **External Group Manager** to manually create groups of external faces based on geometry selected from the **3D Modeler** window or the History tree. Alternatively, you can click **Automatic Extraction of Bounding Faces** to automatically identify the external faces touching the package's bounding box and categorize them in groups or **Automatic Extraction of Silhouette Faces** to automatically identify the faces that compose the DTM's silhouette.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

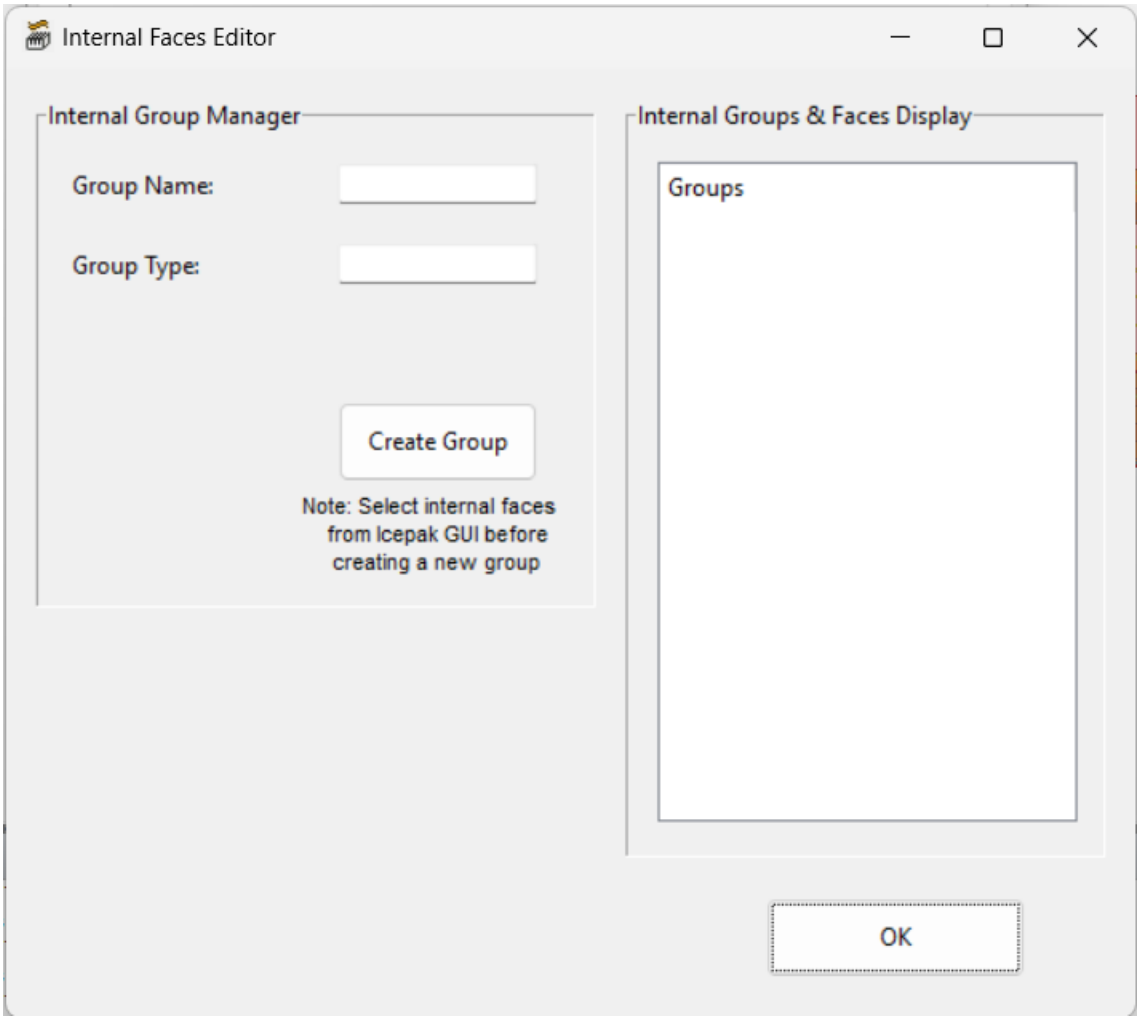
External Group Manager	
Group Name	Enter a **Group Name** for the group of external face geometry.
Group Type	Enter a **Group Type** for the group of external face geometry.
Create Group	Click **Create Group** to save the group name and/or type changes. The Create Group button appears when a group is selected in the Groups tree.
External Groups & Faces Display	
Group Tree	The **External Groups & Faces Display** tree lists the groups and associated faces you've created. If needed, select a group and modify its name or type. From a group's right-click menu, you can unite all the faces into one piece of geometry, delete the group, or delete all groups. From an external face's right-click menu, you can split the face, remove it, or move it to a different group. With multiple faces selected, you can split the faces, unite them, remove them, or move them to a different group.
Automatic Extraction of Bounding Faces	Click **Automatic Extraction of Bounding Faces** to automatically identify the external faces touching the package's bounding box and categorize them in groups. After confirming the bounding box selection, the **Get Boundaries** dialog box appears. Choose whether to create single sheet geometry for each of the six bounding surfaces or to split the sheets into more than one based on the intersecting geometry. Splitting the geometry will require more time to generate all of the geometry.
Automatic Extraction of Silhouette Faces	Click **Automatic Extraction of Silhouette Faces** to automatically identify the faces that compose the DTM's silhouette (the faces of the 3D object formed by the union of all solid geometries in the DTM).
OK	After creating all external groups, click **OK** to save the external groups and close the **External Faces Editor** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Internal Faces Editor

In the **Internal Faces Editor**, use the **Internal Group Manager** to manually create groups of internal faces based on geometry selected from the **3D Modeler** window or the History tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



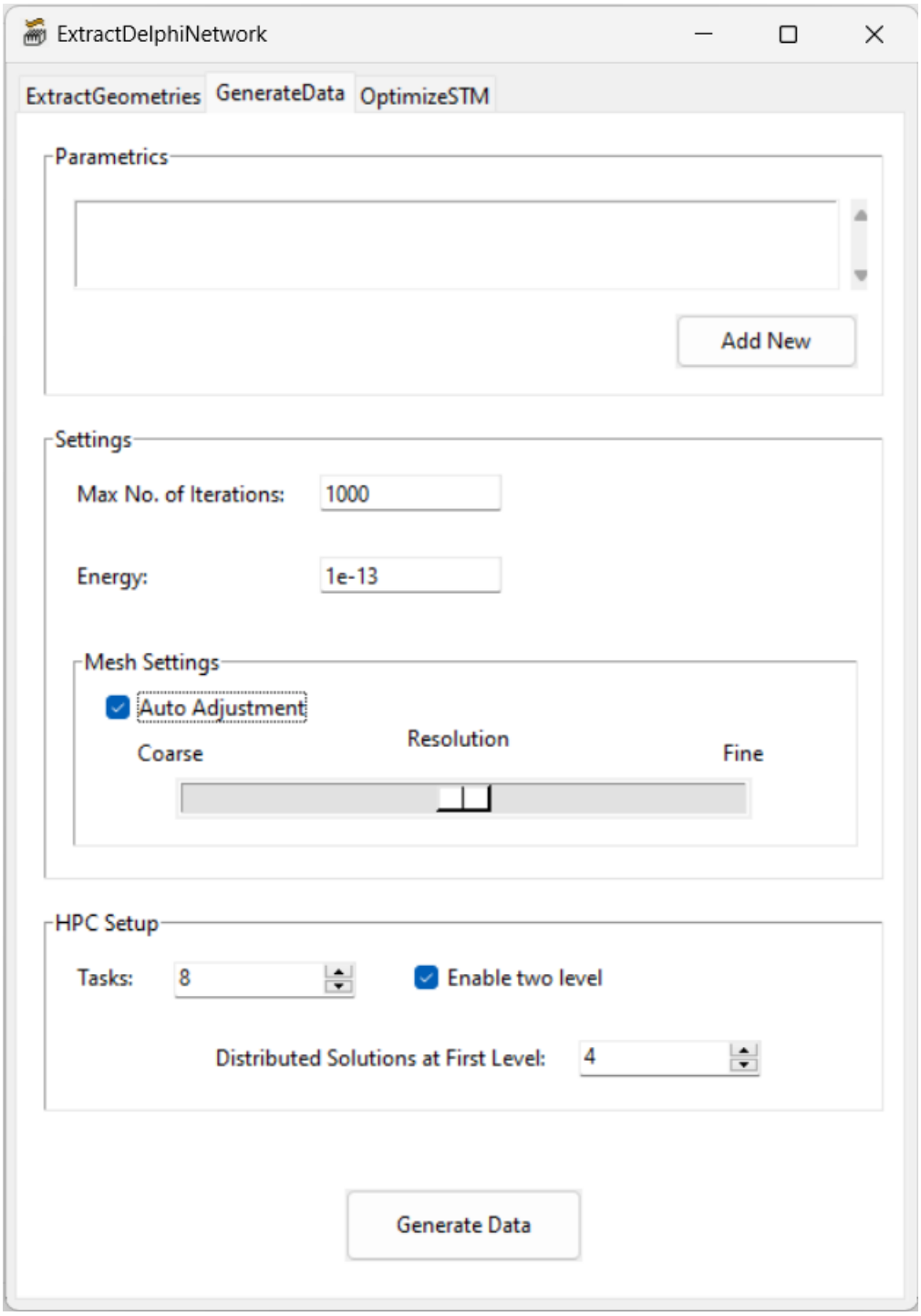
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Internal Group Manager	
Group Name	Enter a **Group Name** for the group of internal face geometry.
Group Type	Enter a **Group Type** for the group of internal face geometry.
Create Group	Click **Create Group** to save the group name and/or type changes. The Create Group button appears when a group is selected in the Groups tree.
External Groups & Faces Display	
Group Tree	The **Internal Groups & Faces Display** tree lists the groups and associated faces you've created. If needed, select a group and modify its name or type. From a group's right-click menu, you can unite all the faces into one piece of geometry, delete the group, or delete all groups. From an internal face's right-click menu, you can split the face, remove it, or move it to a different group. With multiple faces selected, you can split the faces, unite them, remove them, or move them to a different group.
OK	After creating all internal groups, click **OK** to save the internal groups and close the **Internal Faces Editor** dialog box.

Generate Data Tab

The **Generate Data** tab controls the creation, analysis, and post-processing of parametric variations. Boundary conditions, including source powers and ambient temperature, as well as monitor points are generated. It also contains settings for simulation iterations, energy convergence, mesh, and high performance computing. The solutions extracted from the solved parametrics can then be used as training and/or test data sets on the **Optimize STM** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametrics	
Add New	Under **Parametrics**, click **Add New** to open the **New Parametric Setup Editor** dialog box. See "New Parametric Setup Editor" on the facing page for information on specifying parametric setup settings. From a setup's right-click menu, you can change the solution type, delete it, create the parametric setup under **Optimetrics** in the **Project Manager**, or initiate post-processing of a setup that was analyzed individually.
	Note: If you add multiple setups, you can use the right-click options to create and post-process them individually instead of clicking **Generate Data**, which creates, analyzes, and post-processes all setups in the **Parametric** field. Analyzing the parametric setups individually from the **Project Manager** also allows you to optimize your High Performance Computing settings for complex models.
Settings	
Max No. of Iterations	**Max No. of Iterations** specifies the maximum number of solution iterations to be performed in a simulation.
Energy	**Energy** specifies the convergence criterion for the parametric trials. When the residual is less than or equal to the specified value, the solution will be considered converged.
Transient Settings	Click **Transient Settings** to open the **Transient Settings Editor** dialog box. See The Icepak Solve Setup Dialog (Transient) for more information on transient settings. The Transient Settings button is only available when one or more parametric setups in the list has a transient solution type.
Mesh Settings	Select **Auto Adjustment** to display the automatic mesh settings slider bar. The slider bar is a visual representation of the resolution you choose as ranging from coarse resolution with a small mesh size through a five-position scale to a fine resolution with a large mesh size. See Defining Global Mesh Settings for more information on automatic mesh settings.
HPC Setup	
Tasks **Enable two level** **Distributed Solutions at First Level**	Under **HPC Setup**, specify the high performance computing settings. For more information on tasks and two-level job distribution, see High Performance Computing in Icepak.
Generate Data	Click **Generate Data** to begin solving the parametric setups listed in the **Parametrics** field.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Parametric Setup Editor

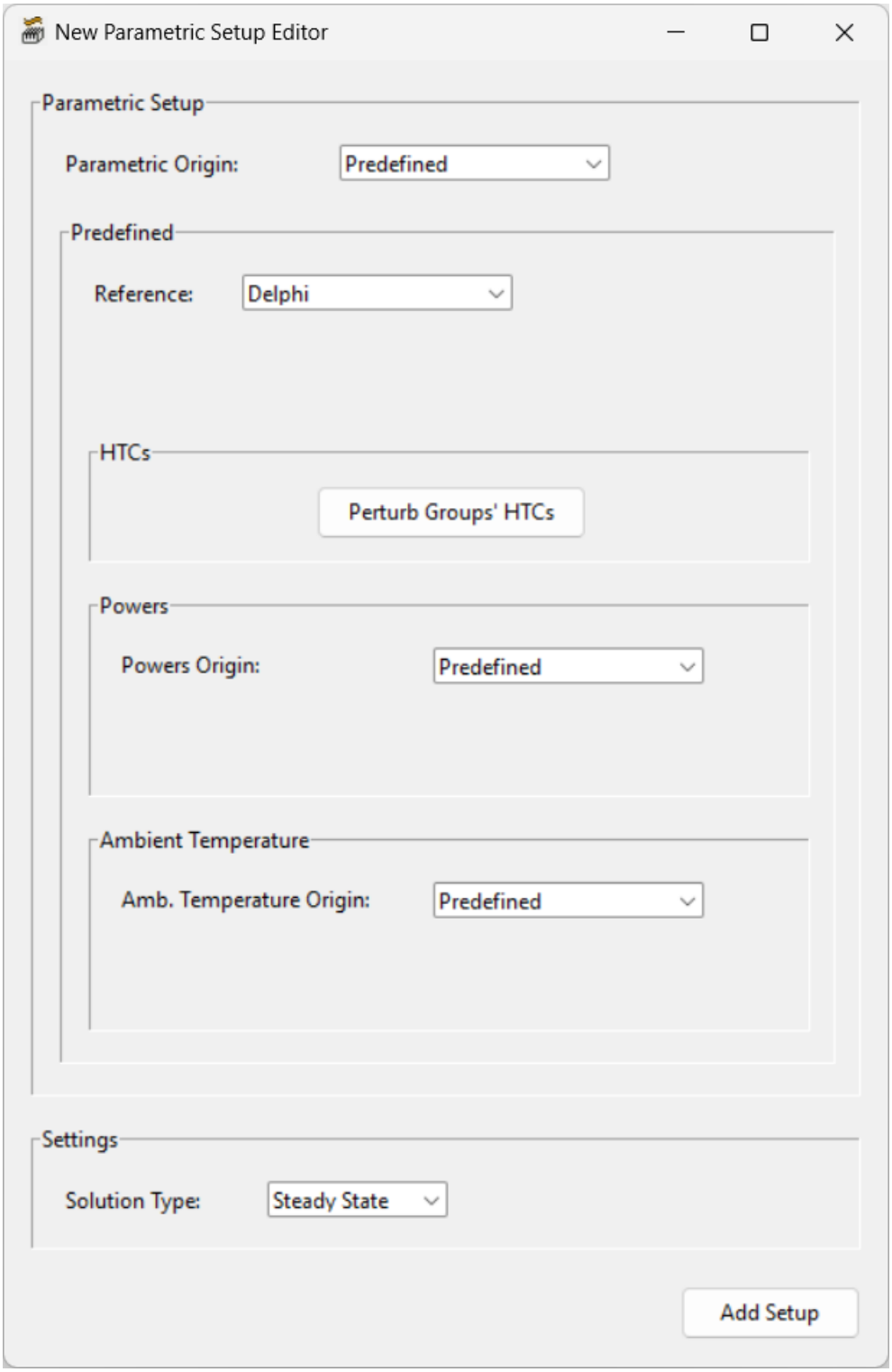
The **New Parametric Setup Editor** allows you to configure and add parametric setups to add to the Parametrics field on the **Generate Data** tab. Configure parametric setups based on three approaches:

- "Predefined" below
- "Sample" on page 3-477
- "Import from File" on page 3-481

Predefined

The Predefined approach loads a parametric setup containing default parametric variations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



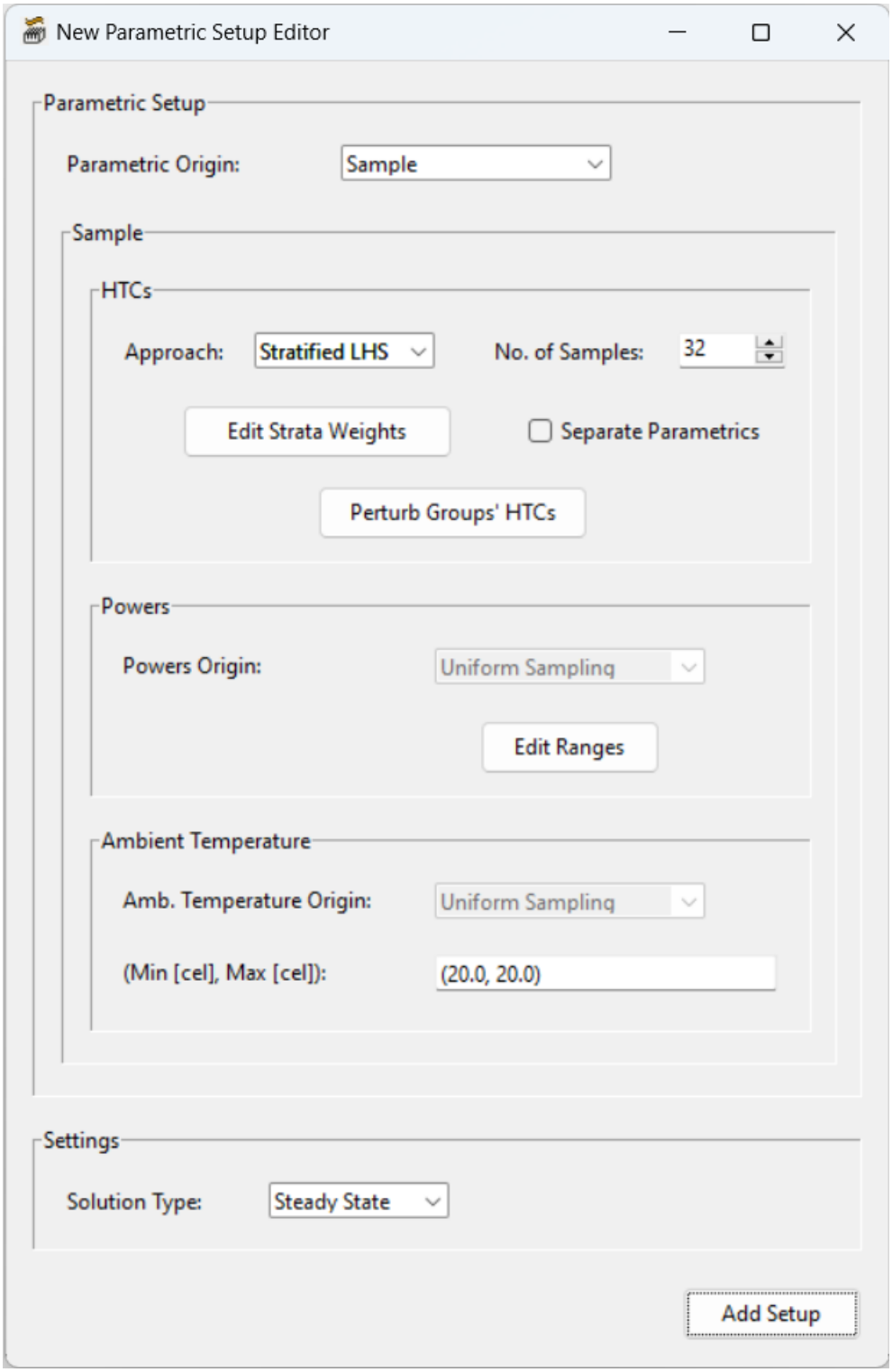
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Setup - Predefined Origin	
Predefined	
Reference	Select the variation set for the parametric trials. **Delphi** uses the set used for training by the Extract Delphi Network toolkit in Electronics Desktop version 2023 R2. **JEDEC** uses the set used for validation by the Extract Delphi Network toolkit in Electronics Desktop version 2023 R2 and defined by the JESD15-4 DELPHI model guidelines. Demo uses a set of seven variations for fast data generation.
HTCs	
Perturb Groups' HTCs	Click **Perturb of Groups' HTCs** to open the **Perturbation Editor for Groups' HTCs** dialog box. Enter a standard deviation for the perturbation of each external face group.
	Note: The heat transfer coefficient for each external face is obtained by multiplying the associated groups' heat transfer coefficient by a multiplicative perturbation obtained by sampling a normal distribution characterized by mean = 1 and standard deviation = the specified value. A standard deviation equal to 0.0 corresponds to no perturbation around the group's value.
Powers	
Power Origin	By default, **Predefined** is selected to use the default power values for the parametric trials. If needed, click **Edit Ranges** to specify a range of power in the **Power Ranges Editor** dialog box.
Ambient Temperature	
Amb. Temperature Origin	By default, **Predefined** is selected to use the default ambient temperature values for the parametric trials. Alternatively, select **Uniform Sampling** and enter minimum and maximum values for the range of ambient temperatures.
Settings	
Solution Type	Select **Steady State** or **Transient** to set the solution type.
Add Setup	Click **Add Setup** to create the parametric setup and add it to the **Parametric** field on the **Generate Data** tab.

Sample

The Sample approach generates new parametric variations based on a specified sampling method.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Setup - Sample Origin	
Sample	
Approach	**Approach** specifies the method for sampling in the hyper-space of heat transfer coefficients for the parametric trials. **Stratified LHS** specifies Stratified Latin Hypercube Sampling. A hyper-space of heat transfer coefficients is defined for the i-th physical environment, and N_i variations are sampled from it based on Latin Hypercube Sampling (**Note**: $\sum_{i=1}^{5} N_i = N_{Tot}$). See "Stratified LHS" on the next page for more information. **LHS** specifies Latin Hypercube Sampling. **Uniform** specifies a sample set that is divided equally within the range.
No. of Samples	Specify the number of samples to include in the variation set.
Edit Strata Weights	For Stratified LHS, click **Edit Strata Weights** to open the **Stratified LHS** dialog box. Specify the percentage of variations to be sampled in each environment. See "Stratified LHS" on the next page for more information on strata weights.
Separate Parametrics	For Stratified LHS, select **Separate Parametrics** if a separate parametric study is needed for each of the environments.
Perturb Groups' HTCs	Click **Perturb of Groups' HTCs** to open the **Perturbation Editor for Groups' HTCs** dialog box. Enter a standard deviation for the perturbation of each external face group. **Note**: The heat transfer coefficient for each external face is obtained by multiplying the associated groups' heat transfer coefficient by a multiplicative perturbation obtained by sampling a normal distribution characterized by mean = 1 and standard deviation = the specified value. A standard deviation equal to 0.0 corresponds to no perturbation around the group's value.
Power	
Power Origin	By default, **Uniform Sampling** is selected to specify random variations of power values for the parametric trials. If needed, click **Edit Ranges** to specify a range of power in the **Power Ranges Editor** dialog box.
Ambient Temperature	
Amb. Temperature Origin	By default, **Uniform Sampling** is selected to specify random variations of ambient temperature values for the parametric trials. Alternatively, select **Uniform Sampling** and enter minimum and maximum values for the range of ambient temperatures.
Settings	
Solution Type	Select **Steady State** or **Transient** to set the solution type.
Add Setup	Click **Add Setup** to create the parametric setup and add it to the **Parametric** field on the **Generate Data** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

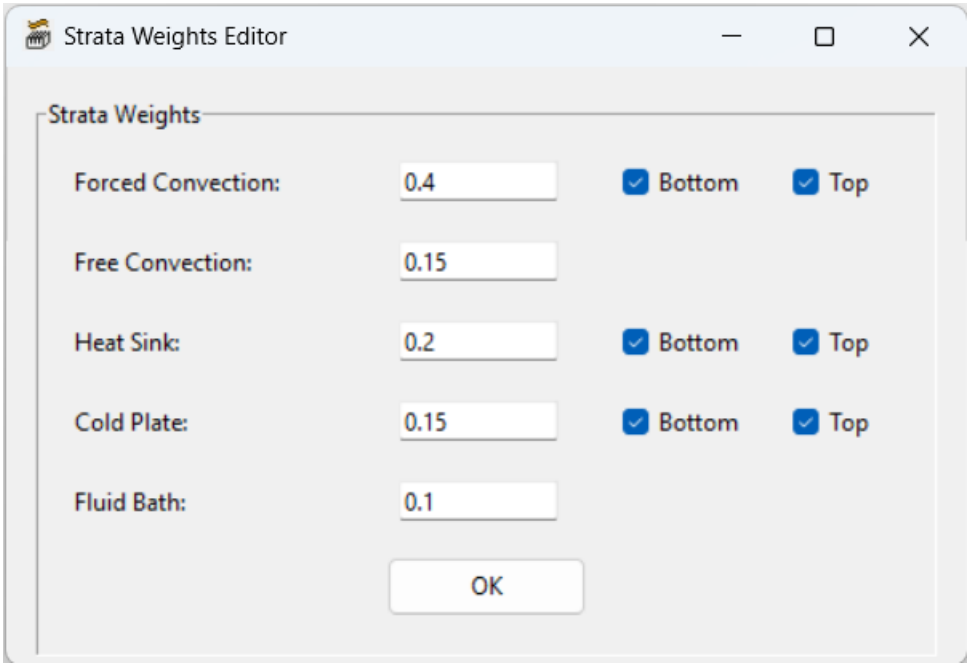
Stratified LHS

Stratified LHS uses stratified sampling to divide a population into distinct groups, or “strata,” so that each group is homogeneous in some respect. In this case, each stratum represents a different physical environment. Once the population is divided into strata, a random sample is then drawn from each stratum based on Latin Hypercube Sampling.

You can specify strata weight in the Stratified LHS dialog box. Enter weight values for each physical environment, including forced convection, free convection, heat sink, cold plate, and fluid bath.

You can select Bottom and/or Top for Forced Convection, Heat Sink, and Cold Plate to specify that the conditions be applied to the bottom and/or top faces, respectively.

> **Note:** The relative percentages do not have to add up to 1. For example, if (0.4,0.4,0.2,0.5,0.5) are specified as strata weights, (20%,20%,10%,25%,25%) of variations are sampled from each environment, respectively.



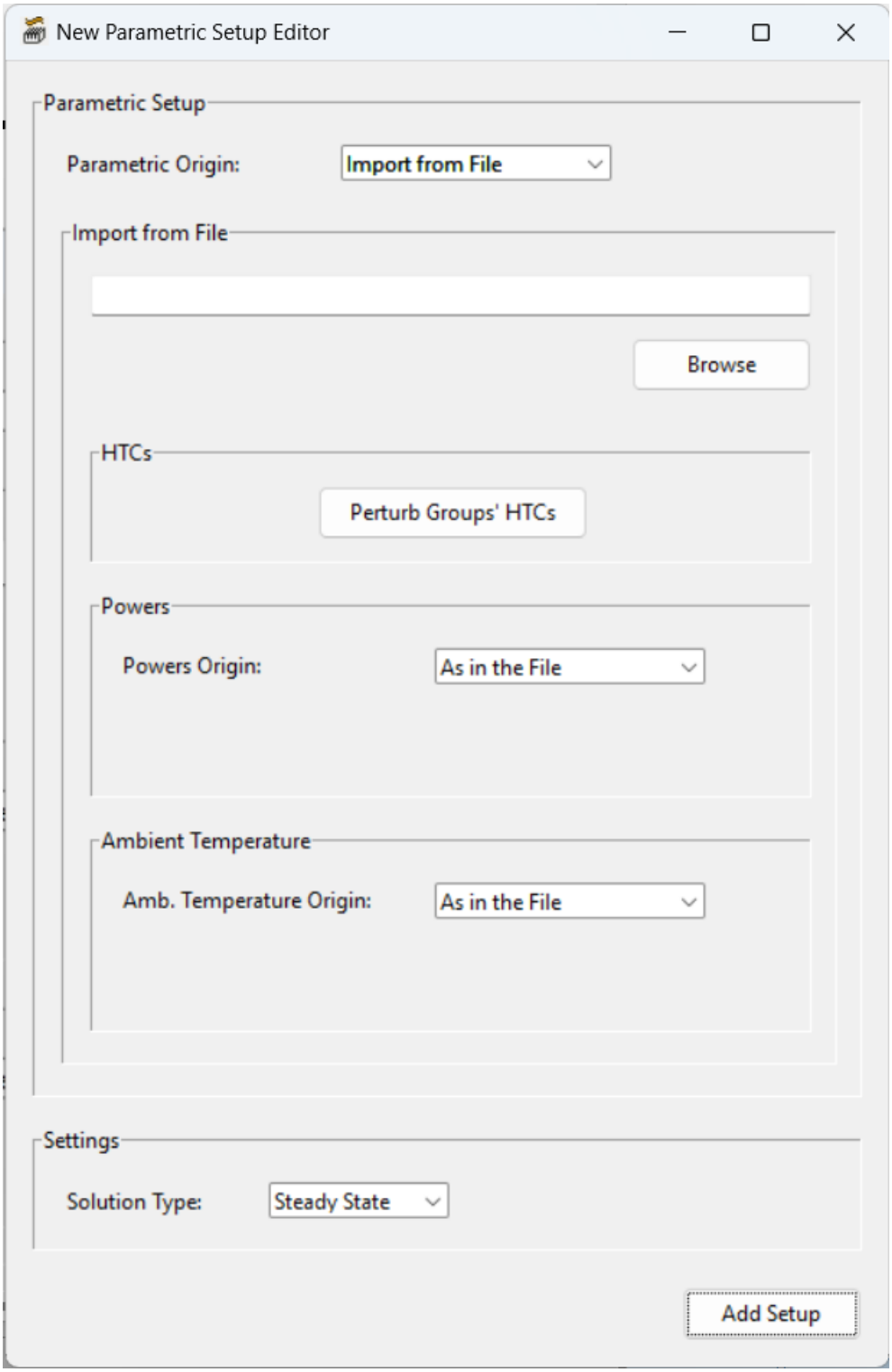
This approach ensures that ensures no variations are sampled in unphysical heat transfer coefficient regions and each stratum is properly represented in the parametric setup based on the importance designated by the strata weights. This potentially increases the representation and accuracy of the overall sample compared to simple random sampling.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import from File

Import from File allows you to generate parametric trials specified in a comma-separated value (.csv) file. Each column in the file must have a header column that includes the variable name with the cells beneath it listing the values. Each value cell must include a unit of measure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Setup - Import from File Origin	
Import from File	
Browse	Click **Browse** to navigate to search for and select the .csv file.
HTCs	
Perturb Groups' HTCs	Click **Perturb of Groups' HTCs** to open the **Perturbation Editor for Groups' HTCs** dialog box. Enter a standard deviation for the perturbation of each external face group. **Note**: The heat transfer coefficient for each external face is obtained by multiplying the associated groups' heat transfer coefficient by a multiplicative perturbation obtained by sampling a normal distribution characterized by mean = 1 and standard deviation = the specified value. A standard deviation equal to 0.0 corresponds to no perturbation around the group's value.
Powers	
Power Origin	By default, **As in the File** is selected to specify the power values defined in the .csv file. Alternatively, select **Uniform Sampling** and Click **Edit Ranges** to replace all the power values with samples from uniform distributions in the range specified in the **Power Ranges Editor** dialog box.
Ambient Temperature	
Amb. Temperature Origin	By default, **As in the File** is selected to specify the ambient temperature values defined in the .csv file. Alternatively, select **Uniform Sampling** to replace all the ambient temperature values with samples from a uniform distribution in a specified range.
Settings	
Solution Type	Select **Steady State** or **Transient** to set the solution type.
Add Setup	Click **Add Setup** to create the parametric setup and add it to the **Parametric** field on the **Generate Data** tab.

The following image displays an example of the acceptable format of a .csv file.

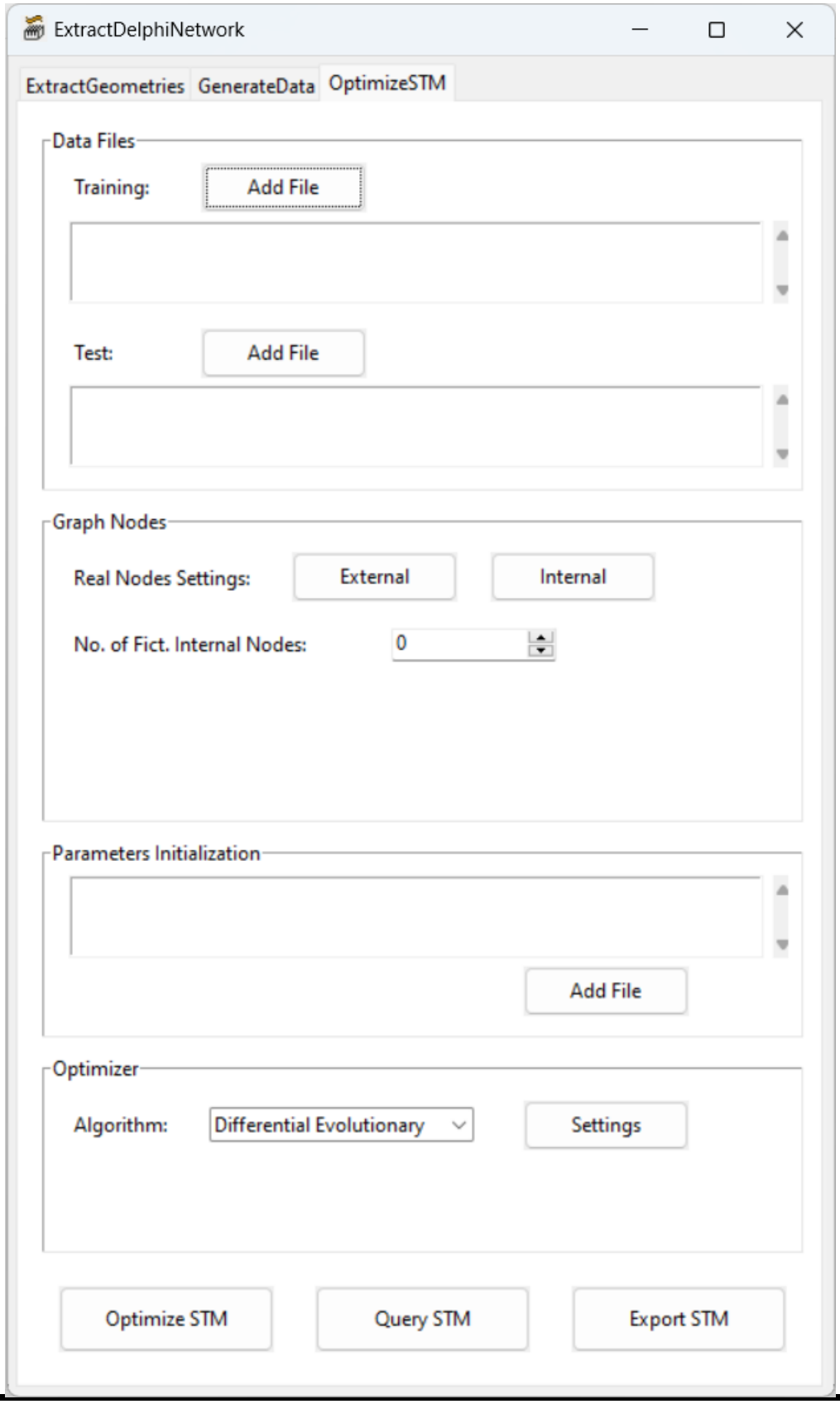
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B	C	D	E
1	$Q_QFN1_DieSource	$hsides	$htop	$hbot	$T_amb
2	1.0W	15.0w_per_m2kel	15.0w_per_m2kel	50.0w_per_m2kel	20.0cel
3	1.0W	15.0w_per_m2kel	15.0w_per_m2kel	100.0w_per_m2kel	20.0cel
4	1.0W	10.0w_per_m2kel	20.0w_per_m2kel	500.0w_per_m2kel	20.0cel
5	1.0W	20.0w_per_m2kel	30.0w_per_m2kel	200.0w_per_m2kel	20.0cel
6	1.0W	10.0w_per_m2kel	50.0w_per_m2kel	250.0w_per_m2kel	20.0cel
7	1.0W	20.0w_per_m2kel	40.0w_per_m2kel	1000.0w_per_m2kel	20.0cel
8	1.0W	100000.0w_per_m2kel	100000.0w_per_m2kel	100000.0w_per_m2kel	20.0cel
9					

Optimize STM Tab

The **Optimize STM** tab controls the construction, training, and export of the Delphi-like Surrogate Thermal Model (STM), starting from DTM (Detailed Thermal Model) data produced in the previous tab or coming from external sources.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Files	
Training	Click **Add File** and select the parametric trial data for the optimizer to use for training.
Testing	Click **Add File** and select the parametric trial data to use for testing against the results of the optimizer.
Note: The training and testing data is saved comma-separated value (.csv) files appended with "_monitors" (steady-state) or "_aedtexport" (transient) at the end of the file name.	
Graph Nodes	
Real Nodes Settings	Click **External** and/or **Internal** to open the **Node Selection** dialog box. See "Node Selection" on the facing page for more information about selecting an approach to modeling network nodes.
No. of Fict. Internal Nodes	Specify the number of fictitious internal nodes to include in the network. Fictitious nodes do not correspond to any sources, external faces, or other real parts of the package. They are added to the graph only to increase its predictive capacity, and they are connected to all other nodes in the network.
No. of Intermediate Nodes	For transient simulations, specify the number of intermediate nodes. Intermediate nodes are fictitious internal nodes that are placed on a network schematic edge to split this into multiple sub-edges. They are added to the network schematic only to increase its predictive capacity, and they are only connected to two other nodes.
Equally Dist. Intermediate Nodes	For transient simulations, adjust the slider bar to control the distribution of the intermediate nodes on the network schematic. The right extreme of the bar distributes intermediate nodes on the schematic as equally as possible (i.e., all the network schematic edges will be split into the same number of sub-edges). The left extreme of the bar assigns intermediate nodes to the edges characterized by smaller thermal resistances (i.e., by preferential heat-pathways).
Note: Capacitances are assigned to dinternal nodes, including sources and fictitious and intermediate nodes (if any).	
Parameters Initialization	
Add File	For transient simulations, click **Add File** and select the *yinf.csv* file, which contains thermal conductance data from a previous steady-state training to improve the steady-state behavior of the CTM. If needed, right-click the file and select **Fix During Optimization** to ensure the thermal resistance values are fixed to the ones learned from steady-state simulations and not retrained and updated.
Note: In order to use the previously generated thermal conductance data, the steady-state component of the DELPHI network schematic must be identical to the previous schematic (i.e., the settings for external nodes and fictitious internal nodes must be identical to the ones previously used for steady-state training).	

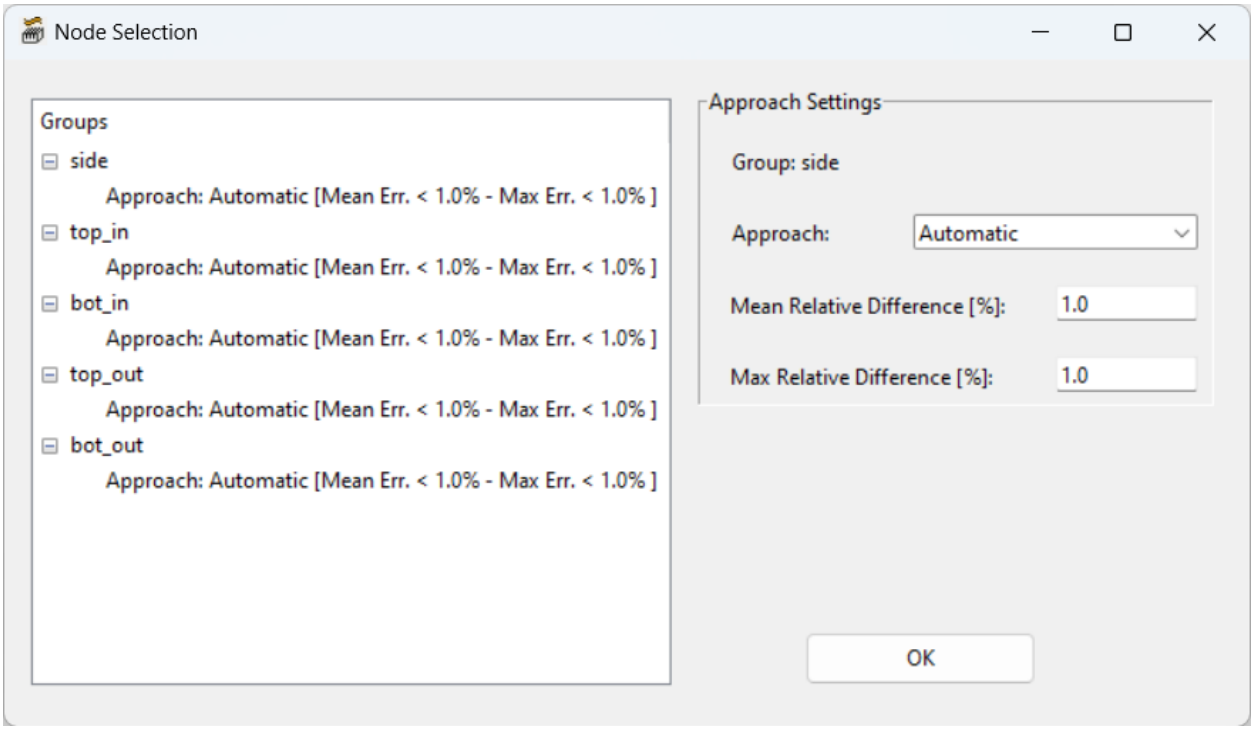
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Files	
Optimizer	
Algorithm	Select the **Gradient-Based** or **Differential Evolutionary** to specify the algorithm used to train the data. See **Gradient-Based Settings** and **Differential Evolutionary Settings** for a list of their respective settings.
Settings	Click **Settings** to open the **Gradient-Based Algorithm** or **Differential Evolutionary Algorithm** dialog box and define the settings for the training.
Optimize STM	Click **Optimize STM** to begin training the STM.
Query STM	Click **Query STM** to open the **STM Querying Settings** dialog box. Select the STM file , a **Querying Mode**, and a data file generated by the optimetrics trials. In the **Delphi-like S.T.M. Predictions - From Data File** dialog box, compare the DTM to the STM. Select **Int. Features** to view the temperature data on the real internal nodes.
Export STM	Click **Export STM** to generate an Electronics Desktop 3D component (.a3Dcomp) file, which is saved in a new folder in the project directory (prepended with "Train_"). Results images are also located in this directory.

Node Selection

In the **Node Selection** dialog box, define the approach for creating network nodes for each group of external faces.

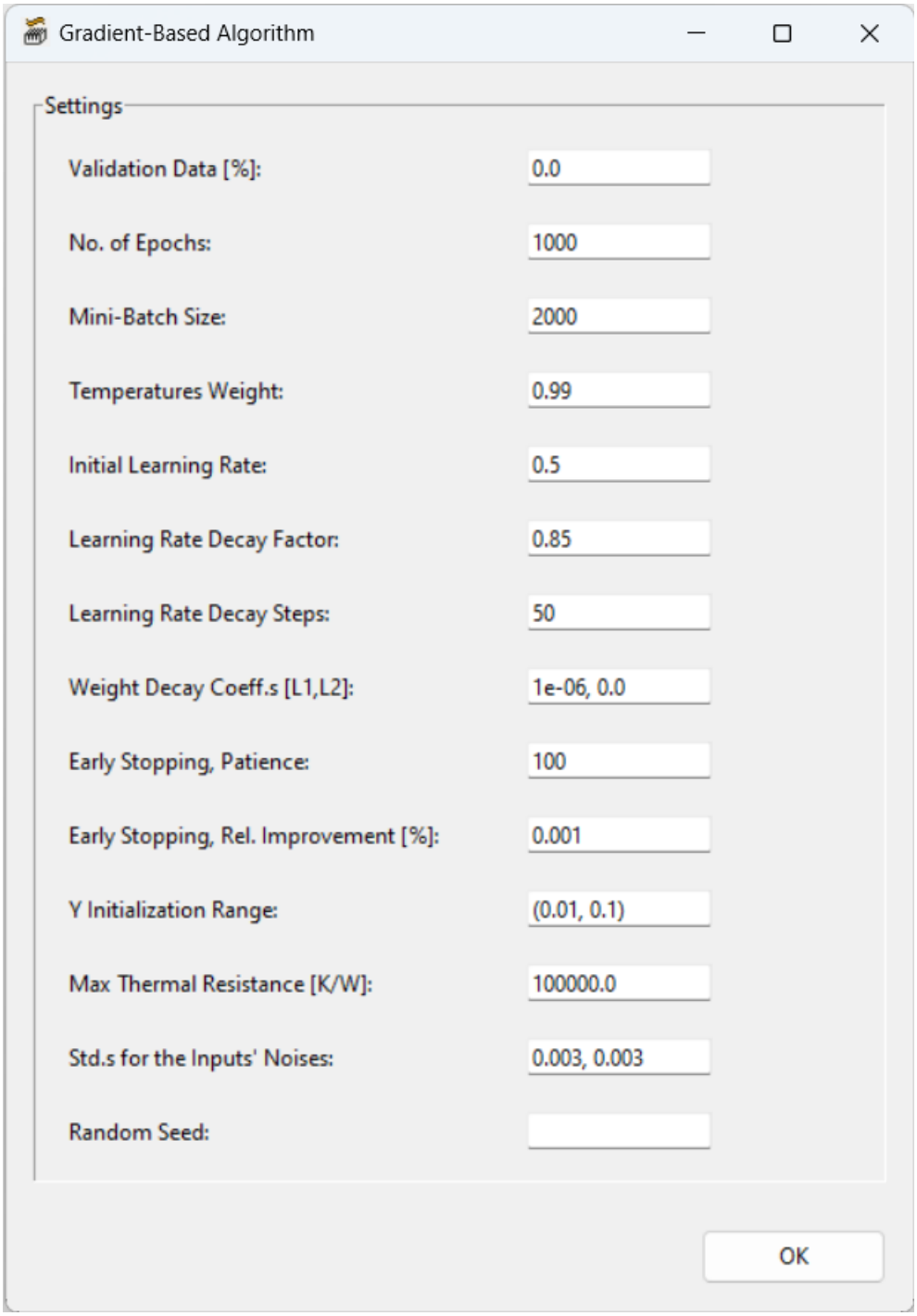
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Groups	
Groups	**Groups** lists the groups of external faces. Select a group from the groups list to define its **Approach Settings**.
Approach Settings	
Group	**Group** displays the selected group.
Approach	From the **Approach** drop-down list, select **Automatic**, **Group-Specific**, or **Face-Specific**. **Automatic** maps two or more faces in the group to a node if their temperature difference is below specified thresholds. **Group-Specific** creates one node for the entire group of faces. Face-specific creates a node for each face in the group.
Note: The i-th face contributes to the corresponding i-th group's node with weight $w_i = A_i / \sum A_i$.	
Mean Relative Difference [%]	For the **Group-Specific**, define the mean and max relative temperature difference for the creation of network nodes.
Max Relative Difference [%]	

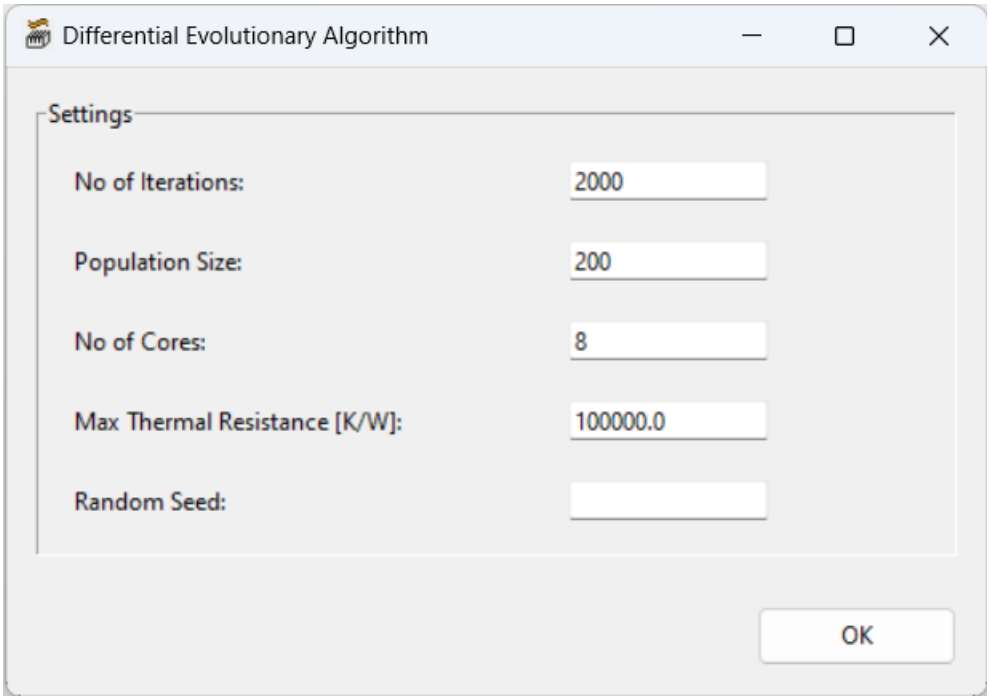
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gradient-Based Algorithm Settings



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Differential Evolutionary Algorithm Settings



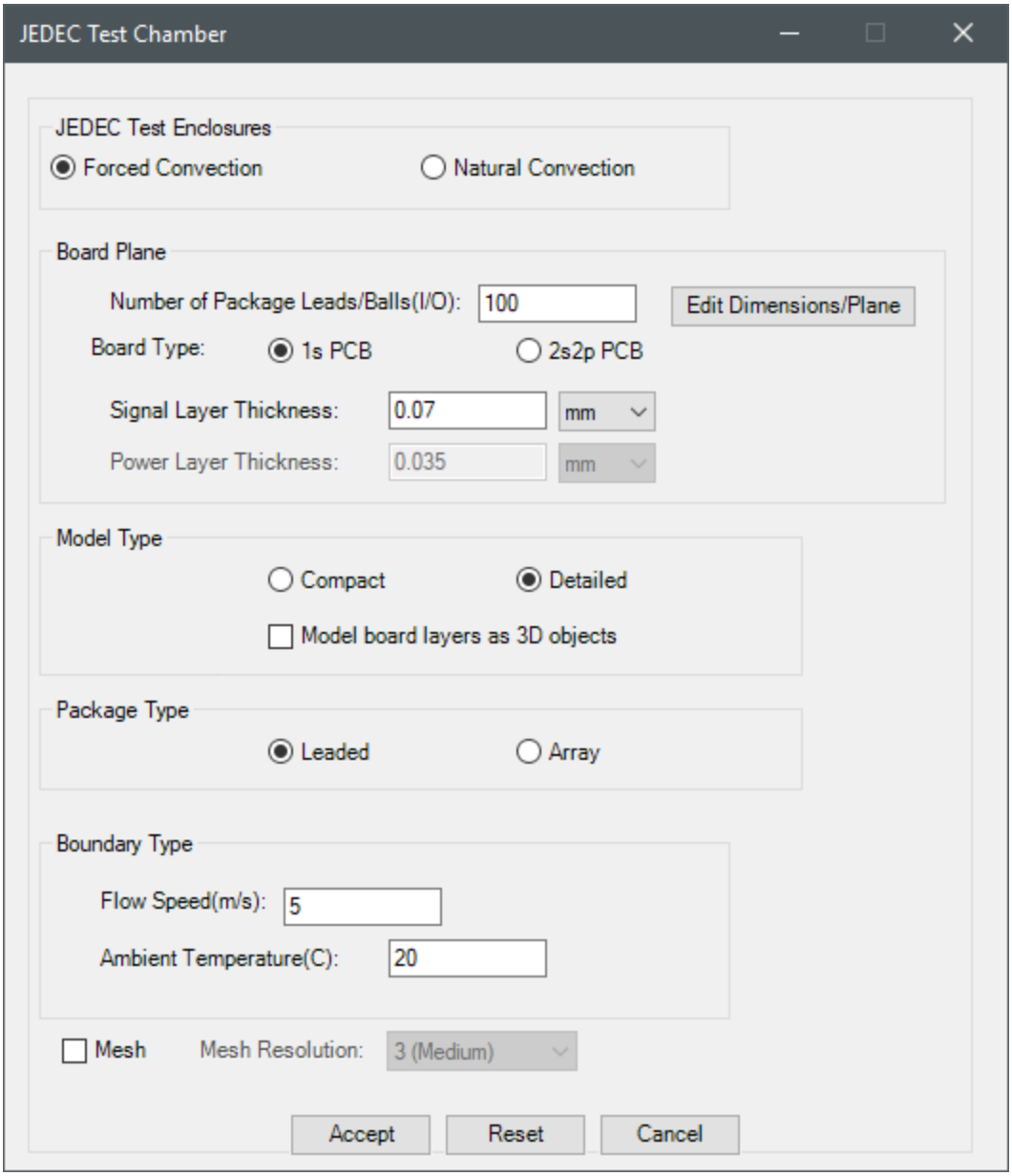
Training Results

In the training files directory, open the .png image file appended with "_error_bars" to view the temperature mean and max temperature error percentage.

IC Packages - JEDEC

JEDEC enables you to quickly set up test chambers for IC package thermal characterization. You can analyze the thermal resistance under conditions of forced convection or natural convection.

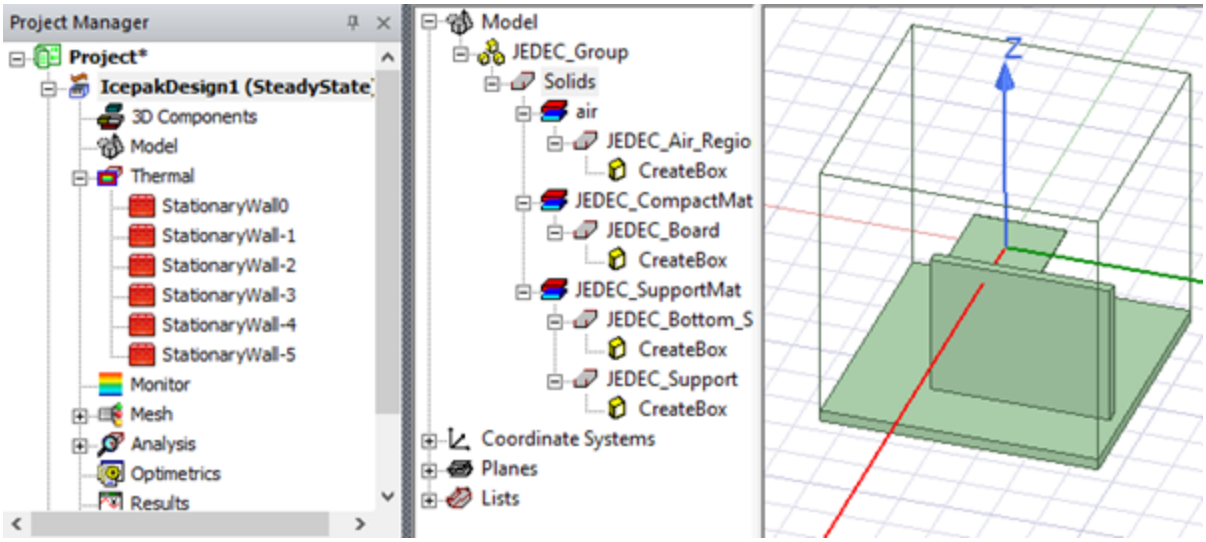
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



JEDEC Test Enclosures	Under **JEDEC Test Enclosures**, specify the type of chamber, forced or natural convection.
Board Plane	Under **Board Plane**, specify the number of package leads/balls in the device that you are characterizing, board type, and thickness of the signal layer and power layer (2s2p PCB only). Also, to specify the package dimensions and plane, click **Edit Dimensions/Plane** to open the **Edit Plane/Board Dimensions** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Type	Under **Model Type**, specify whether the model is compact or detailed. A compact board will be represented by a single thick plate regardless of how many layers it contains. The effect of copper layers is accounted for in the form of directional conductivities of the solid material assigned to the thick plate. A two-layer detailed board will be modeled by two thick plates representing the top (metal) trace layer and the plate with isotropic conductivity (dielectric) underneath the trace layer. A four-layer detailed board includes two additional thin conducting plates that are embedded inside the dielectric to represent two intermediate trace layers. Enable **Model board layers as 3D objects** to create box geometry for the board layers.
Package Type	Under **Package Type**, specify whether the package is leaded or array.
Boundary Type	Under **Boundary Type**, specify the speed of the flow (forced convection only) and temperature.
Mesh	Select **Mesh** to specify the **Mesh Resolution** and mesh the model when you click **Accept**.
Solve	Select **Solve** to analyze the model when you click **Accept**.
Accept, **Reset**, **Cancel**	Click **Accept** to create the test chamber. If you selected **Mesh** and **Solve**, the Electronics Desktop will generate a mesh and analyze the design, respectively. Click **Reset** to restore the default values. Click **Cancel** to close the dialog box without creating the test chamber.



Iron Python Script – *JEDEC.py*:

The following is the toolkit's scripting information.

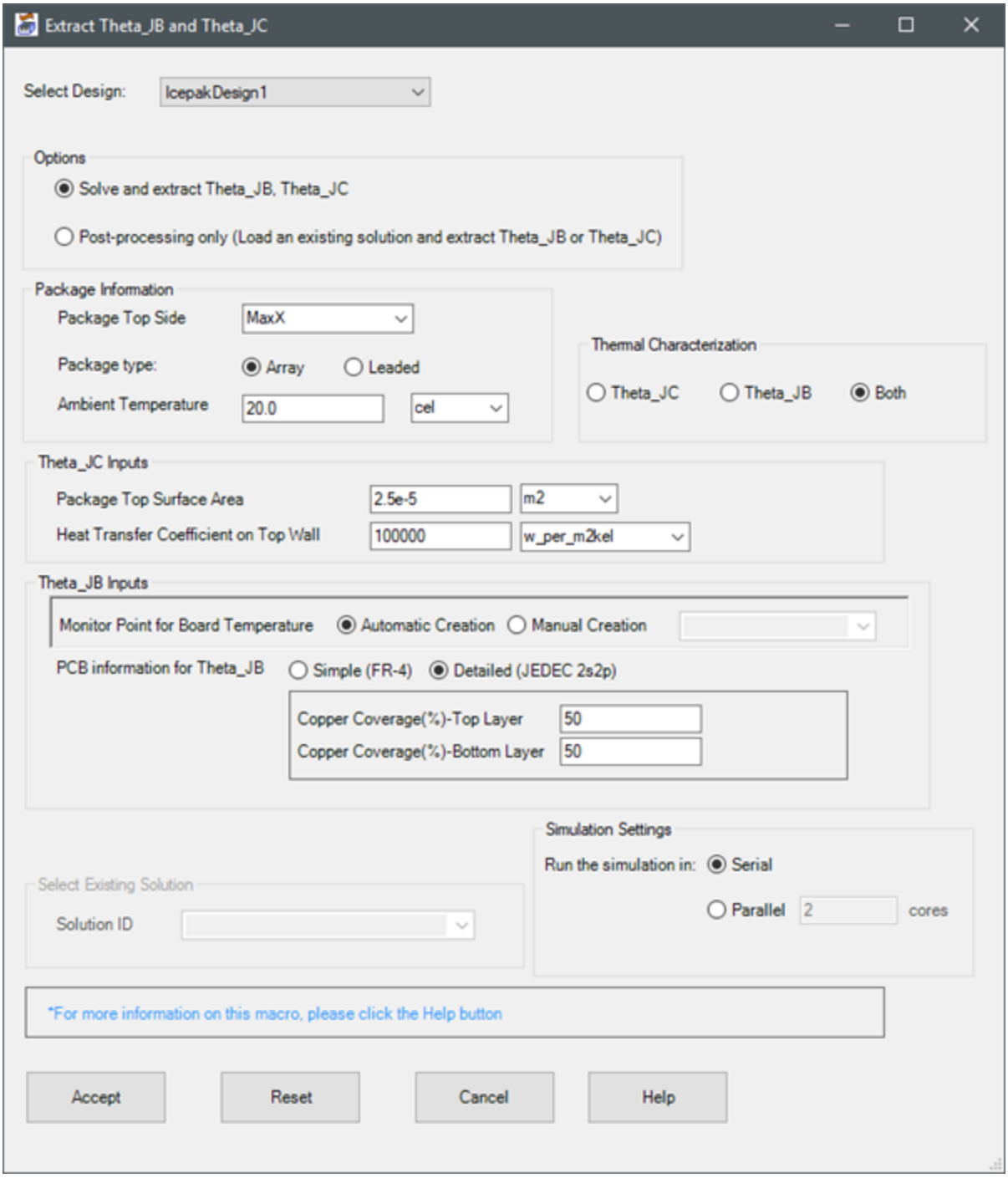
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 data = ["ng",
         'length_unit:meter',
         'xS:-0.00075',
         'yS:-0.00075',
         'zS:-0.00075',
         'xL:0.015',
         'yL:0.015',
         'zL:0.015',
         'plane:ZX'
         'Test Enclosure Type:Natural Convection',
         'Number of Package Leads/Balls:100',
         'Board Type:1s PCB',
         'Signal Layer Thickness:0.00007',
         'Power Layer Thickness:0.000035',
            'Model Plane:Detailed',
                'Model board layers as 3D objects:False',
         'Package Plane:Leaded',
         'Flow Speed(m/s):5',
         'Ambient Temperature(C):20',
         'Mesh:No',
         'Mesh Resolution:3',
         'Solve:No'
         ]
 oDesign.RunToolkit("SysLib", "Modeling/IC Packages/JEDEC", [])
 oDesign.RunToolkit("SysLib", "Modeling/IC Packages/JEDEC",
["ng"])
 oDesign.RunToolkit("SysLib", "Modeling/IC Packages/JEDEC",
data)
```

IC Packages - Extract JB JC

Extract JB JC enables you to extract Theta_JB and Theta_JC for a package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Options	You can choose to run the simulation and perform the extraction or select an existing solution.
Package Information	Define the package top side, type, and ambient temperature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Theta_JC Inputs	Define the package top surface area and heat transfer coefficient on the top wall.
Theta_JB Inputs	Define the monitor point creation method and PCB information (simple or detailed). For Detailed PCBs, define the copper coverage on the top and bottom layers.
Select Existing Solution	If you selected **Post-processing only** under **Options**, select the solution from the **Solution ID** drop-down list.
Simulation Settings	If you selected, **Solve and extract Theta_JB and Theta_JC**, choose where to run the simulation in serial or parallel mode. For parallel simulations, specify the number of cores to use.
Accept	Click **Accept** to run the simulation and/or perform the extraction.
Reset	Click **Reset** to restore the default values.
Cancel	Click **Cancel** to close the dialog box.
Help	Click **Help** to display a text file with additional information and limitations regarding using the Extract_JB_JC toolkit.

Iron Python Script – *Extract_JB_JC.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
           'designname:IcepakDesign1',
           'Options:Solve', #Post for post processing
           'topSide:MinX',
           'packagetype:Array', #Other option is Leaded
           'ambienttemp:20.0',
           'ambienttempunit:cel',
           'Type:ThetaJC', #Other options are ThetaJB,Both
           'packagetsa:2.5e-5',
           'packagetsaunit:m2'
           'htcwall:10000',
           'htcwallunit:w_per_m2kel',
           'Monitorpoint:Automatic', #Other option is Manual
           'pcb_info:Detailed', #Other option is Simple
           'toppercent:50',
           'bottompercent:50',
           'Solvesetting:Serial', #Other options is Parallel
           'cores:2',
           'setupname:Setup1',
           'Monitorpoint:Point1'
           ]
```

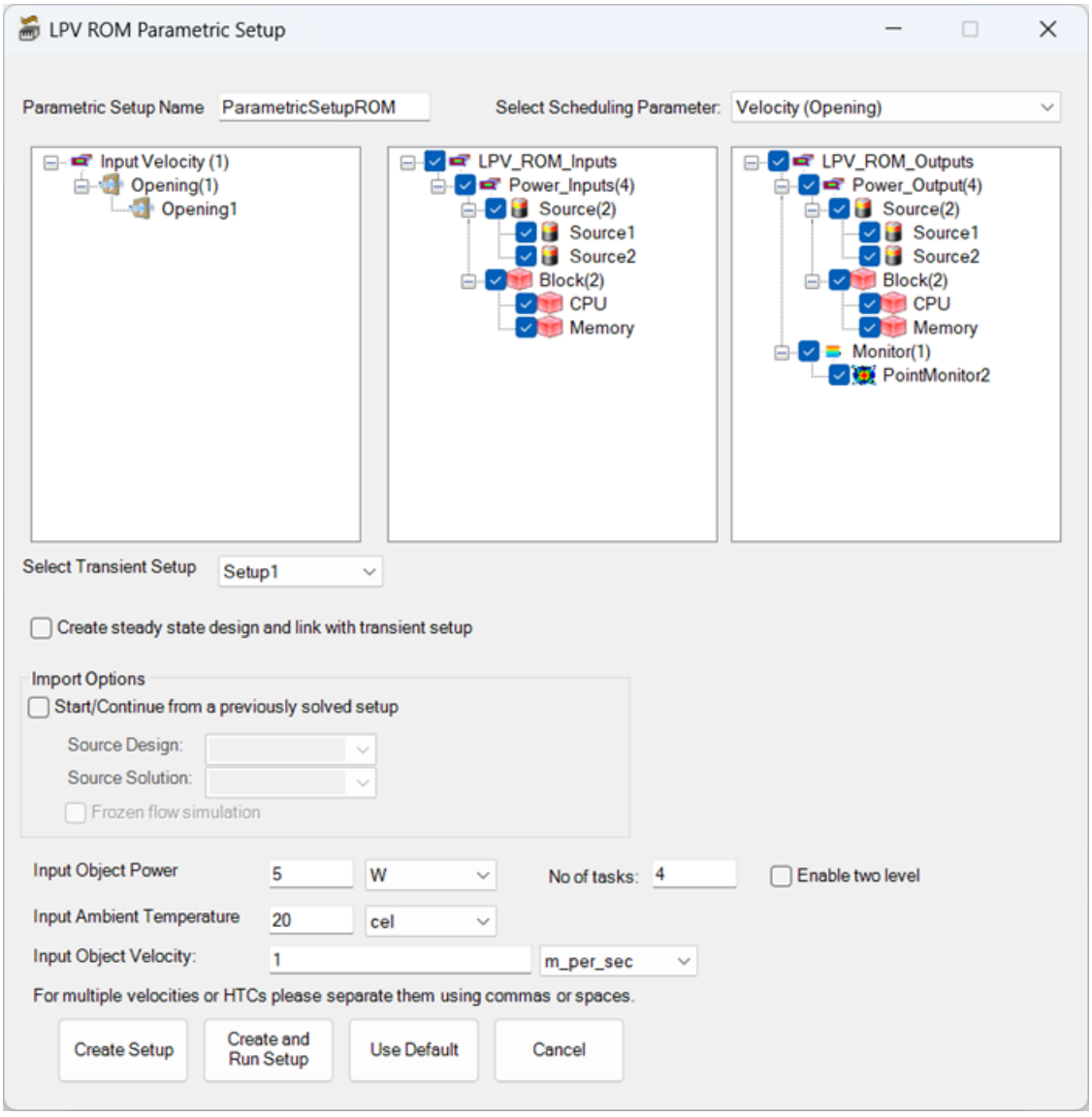

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LPV ROM Parametric Setup

LPV ROM Parametric Setup enables you to quickly create a parametric setup for state-space characterization. The toolkit provides options to model velocity (opening boundary) or heat transfer coefficient (wall boundary).

> **Note:**
>
> See Getting Started with Icepak - LPV ROM for a workflow related to this feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parametric Setup Name	Enter a name for the parametric setup.
Select Scheduling Parameter	From the drop-down list, select **Velocity (Opening)** or **Heat Transfer Coefficient (Wall)**.
Inputs and **Outputs**	Select the flow and thermal inputs and the outputs.
Select Transient Setup	Select the transient solution setup. If needed, enable **Create steady state design and link with transient setup**.
Import Options	Select **Start/Continue from a previously solved setup** to link to the steady-state design. Select the design and solution from the drop-

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

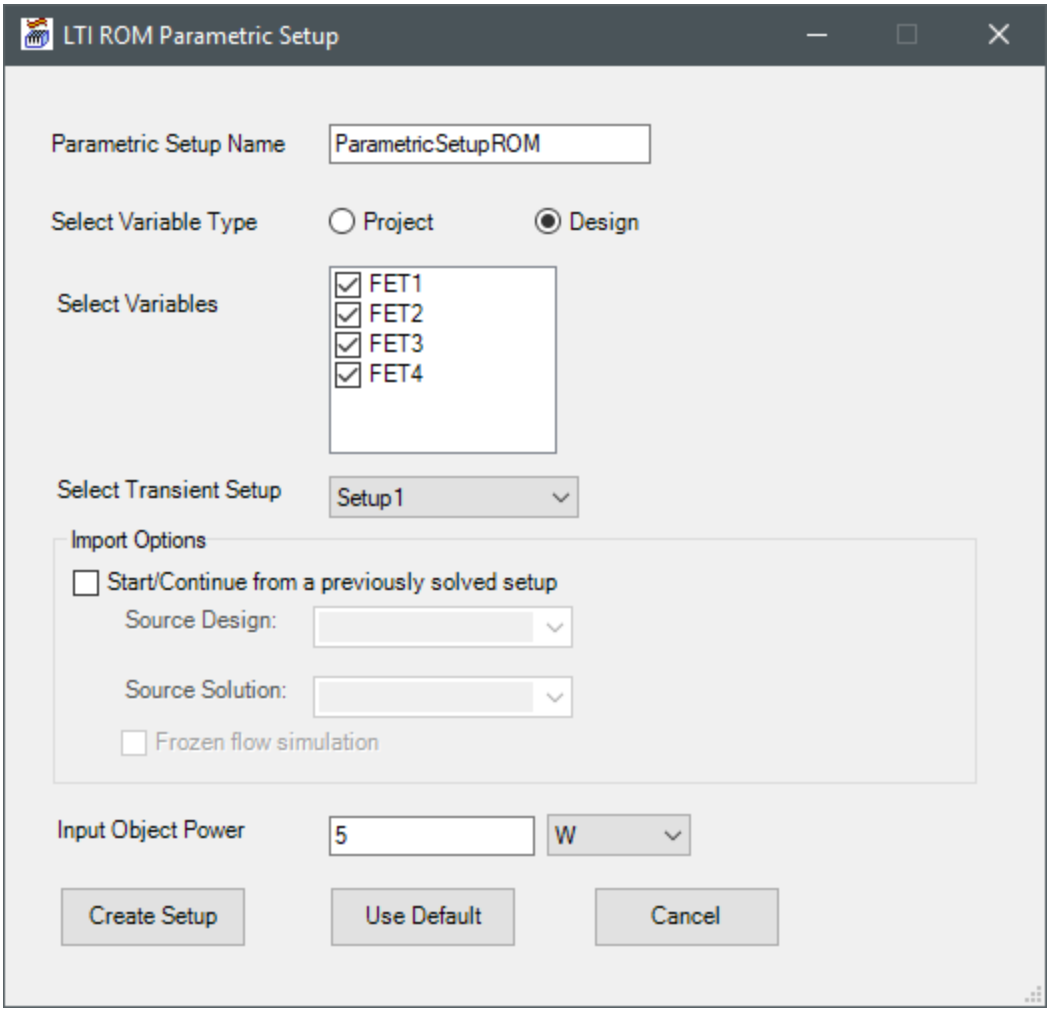
	down lists. If the steady-state design solves for flow and temperature, select **Frozen flow simulation**.
Input Object Power	Specify the power for the input objects for the parametric trials.
Input Object Velocity	With **Velocity (Opening)** specified as the scheduling parameter, specify the flow for the opening for the parametric trials.
Input Object HTC	With **Heat Transfer Coefficient (Wall)** specified as the scheduling parameter, specify the heat transfer coefficient for the wall for the parametric trials.
No. of tasks/Enable two level	If needed, edit the number of tasks for the solver to run and enable two-level high performance computing.
Create Setup	Click **Create Setup** to create the parametric setup, which is listed under **Optimetrics** in the **Project Manager**.
Create and Run Setup	Click **Create and Run Setup** to create and run the parametric setup.
Use Default/Cancel	Click **Use Default** to restore the default settings. Click **Cancel** to close the LPV ROM Parametric Setup toolkit without creating and running a parametric setup.

LTI ROM Parametric Setup

LTI ROM Parametric Setup enables you to quickly create a parametric setup for state-space characterization. See State-Space Characterization for more information.

> **Note:**
>
> See Getting Started with Icepak - LTI ROM for a workflow related to this feature.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parametric Setup Name	Enter a name for the parametric setup.
Select Variable Type	Select **Project** or **Design** to display project or design variables in the **Select Variables** list.
Select Variables	Select the variables to include in the parametric setup.
Select Transient Setup	Select the transient solution setup.
Import Options	Select **Start/Continue from a previously solved setup** to link to the steady-state design. Select the design and solution from the drop-down lists. If the steady-state design solves for flow and temperature, select **Frozen flow simulation**.
Input Object Power	Specify the power for the input objects for the parametric trials.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

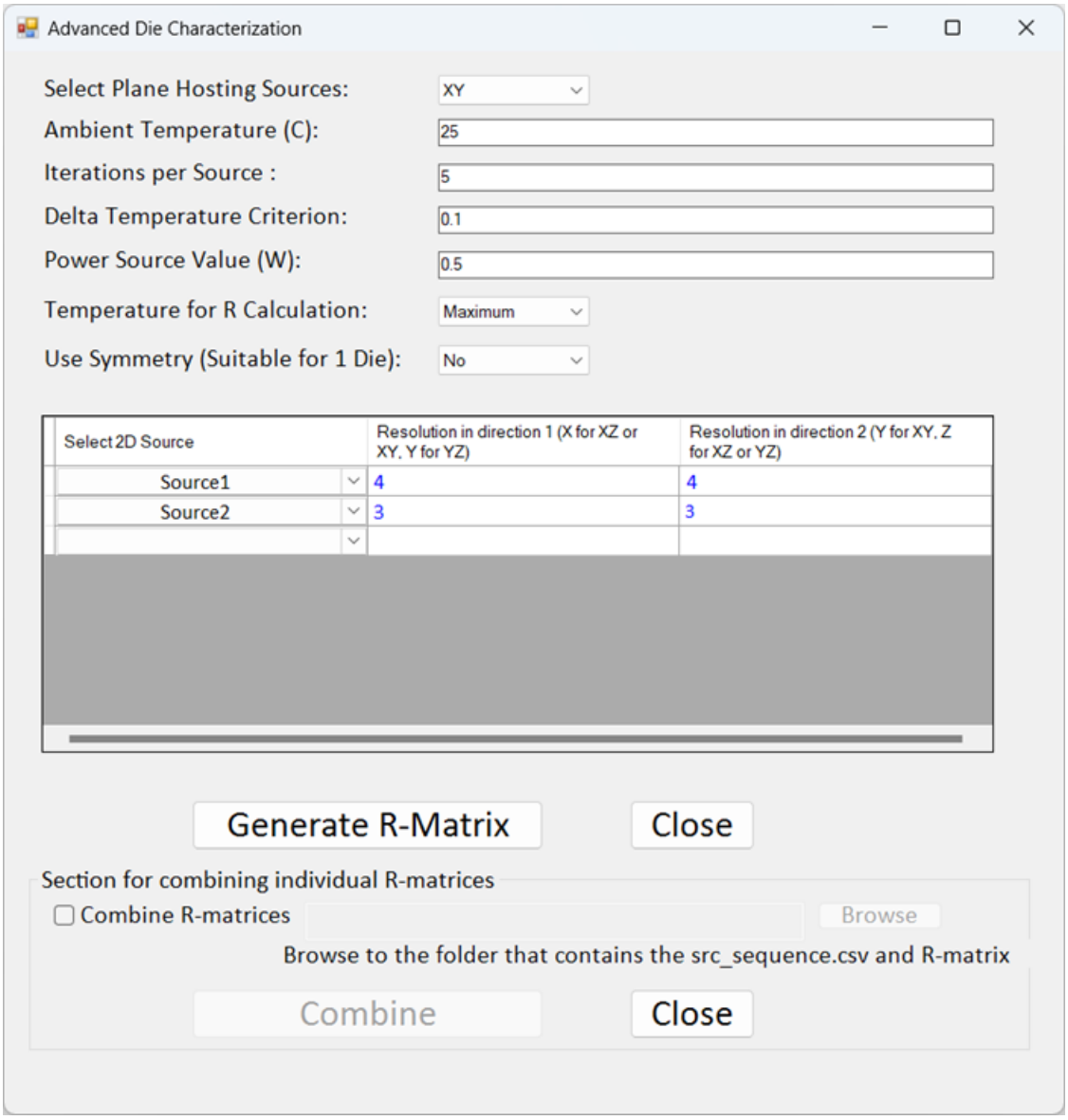
Create Setup	Click **Create Setup** to create the parametric setup, which is listed under **Optimetrics** in the **Project Manager**.

R-Matrix

> **Note:**
>
> Before running the R-Matrix toolkit, ensure that the mesh count on the source(s) is at least four times the resolution specified in both coordinate directions.

R-Matrix is a type of field ROM for use with a die/chip only. It is used to predict the temperatures on a die/chip with various combinations of tile level input powers. The toolkit produces results instantaneously without running a CFD simulation. It generates an R-matrix for a single or multiple dies. Traditionally, parametric trials are used to turn on individual tile level sources (similar to the LTI and LPV ROM approaches) and turn off the rest of the tile level sources to obtain temperature and resistance data for every tile. The R-Matrix toolkit uses a novel approach to turn on and off the tile level sources in a single analysis, saving computational resources and time. Temperatures of the sources are closely monitored every iteration, and a decision is made to turn off the current tile level source and move to the next source after the convergence criterion is met. All the tiles on a given source in the analysis are virtual and surface flux is applied to the meshed faces of the source. This reduces the modeling complexity of handling large number of tiles because all the tiles are represented by a single sources for one die/chip.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



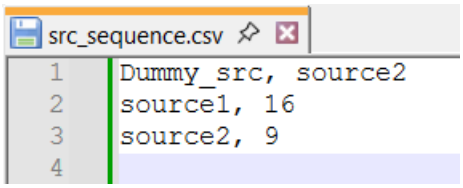
Select Plane Hosting Sources	For **Select Plane Hosting Sources**, select the plane on which the source geometry exists.
Ambient Temperature (C)	For **Ambient Temperature (C)**, specify the ambient temperature for the analysis.
Iterations per Source	For **Iterations per Source**, specify the number of iterations for the analysis to run for each tile on the selected source.
Delta Temperature Criterion	For **Delta Temperature Criterion**, specify the change in temperature for the analysis to converge and end for each tile on the selected source.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Power Source Value (W)	For **Power Source Value (W)**, specify (in Watts) the power for each tile on the selected source.
Temperature for R Calculation	For **Temperature for R Calculation**, select **Maximum** or **Average**. This temperature is used in the R value equation, which is as follows: (FaceTemp - AmbientTemp)/Power, where FaceTemp is the maximum or average temperature based on the selected source.
Use Symmetry (Suitable for 1 Die)	When analyzing one die, select one option for the solver to reduce the area of the die that is analyzed, assuming symmetrical properties in the remaining area(s) of the die. **Half - Horizontal** solves half of the die area divided vertically, assuming two symmetrical halves mirrored horizontally. **Half - Vertical** solves half of the die area divided horizontally, assuming two symmetrical halves mirrored vertically. **Quarter** solves one quarter of the die area divided horizontally and vertically, assuming symmetry both vertically and horizontally. Select **No** to solve the entire die area.
Select 2D Source	From the drop-down, select the source boundary condition(s) for the die characterization analysis.
Note: For a multi-die analysis, specify the powered source on the first line of the source table.	
Resolution	For each source, specify the number of tiles (resolution) to be analyzed in both coordinate directions.
Generate R-Matrix	Click **Generate R-Matrix** to run the analysis and create the R-matrix files.
Close	Click **Close** to close the toolkit dialog box.

Combining R-Matrix Files

After performing the analysis to create the final R-matrix files, you can use the toolkit again to combine them to create one R-matrix file per die/chip. First you must create a comma-separated value (.csv) file (named *src_sequence.csv*) and place it in the *Rmatrix_files* directory generated when you first created the R-matrix files. The src_sequence.csv file specifies the names of the sources in the R-matrices and the number of tiles in each source. See the following example:

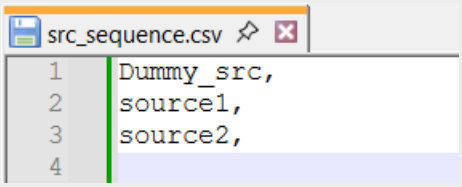


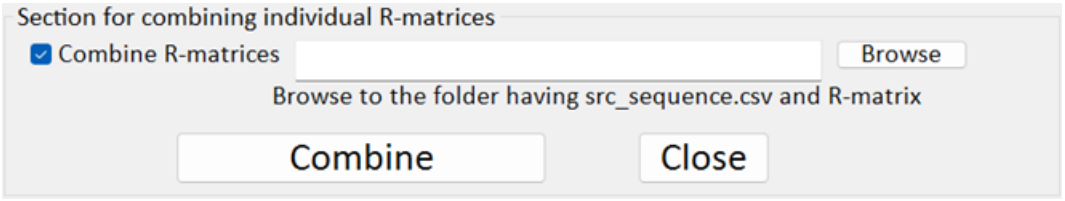
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this source sequence file, a dummy source is specified in the first line. The dummy source is used only to record the temperature; it is not powered. On the next lines, specify the sources to be combined in the desired order followed a comma and the number of tiles on each source.

Note:

A dummy source is optional. The following is an example of a source sequence file without a dummy source specified. No source is specified after *Dummy_src*.



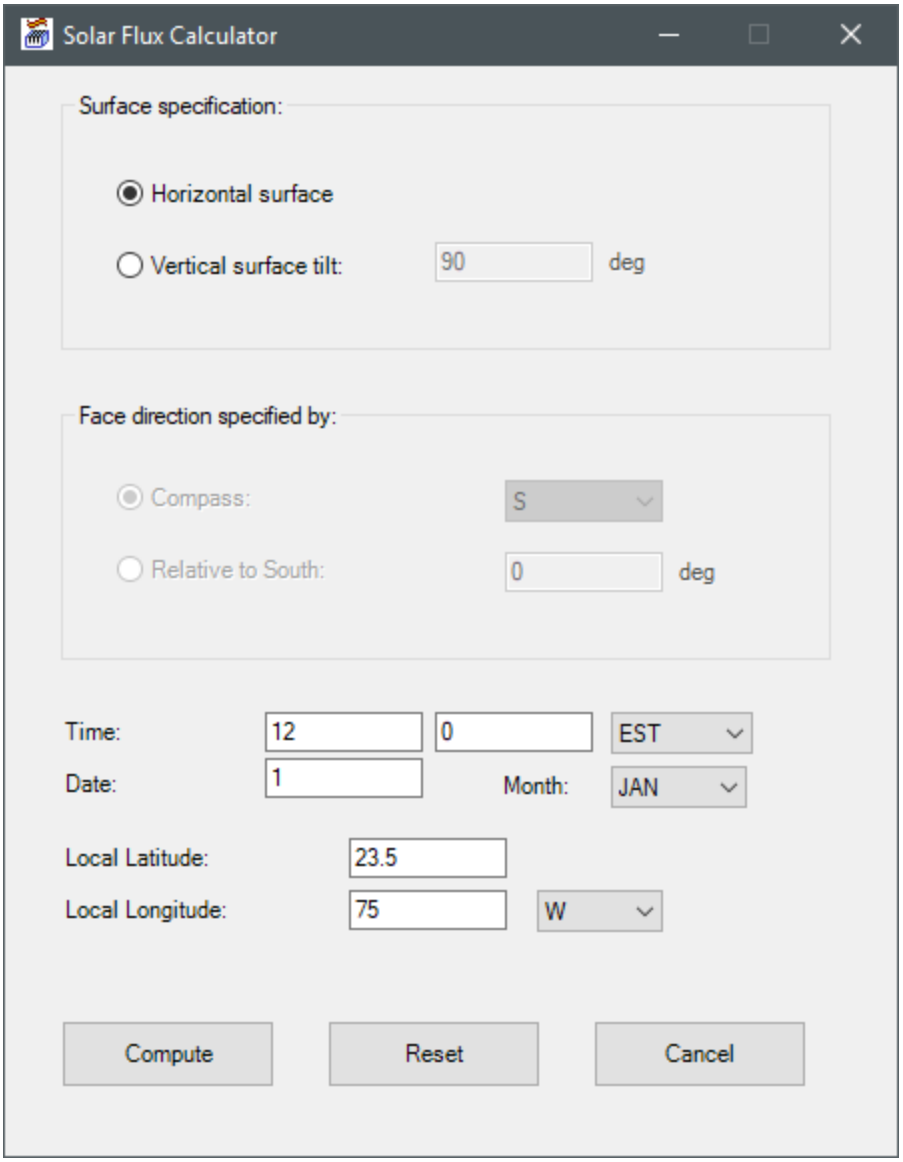


Section for combining individual R-matrices	
Combine R-matrices	Select **Combine R-matrices** to enable the **Browse** and **Combine**.
Browse	Click **Browse** and select to the *Rmatrix_files* folder where the R-matrix and src_sequence.csv files are located. With the *Rmatrix_files* folder selected, click **OK** to close the **Browse For Folder** dialog box.
Combine	Click **Combine** to generate the combined R-matrix files.
Close	Click **Close** to close the toolkit dialog box.

Solar Flux Calculator

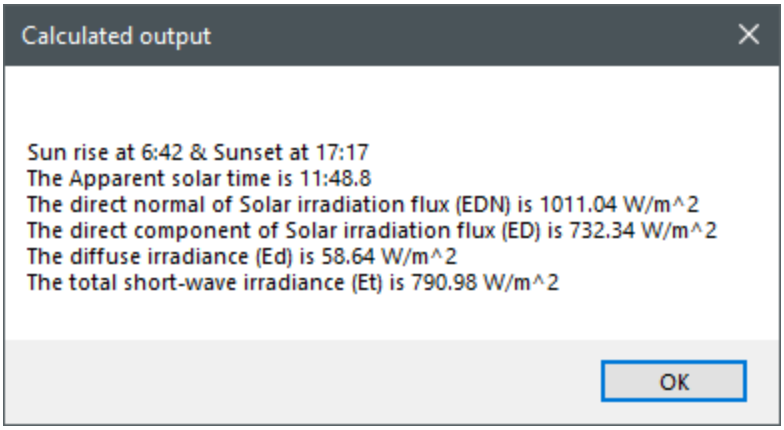
Solar Flux Calculator enables you to compute the solar flux along a surface at a particular latitude, longitude, date, and time of day.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Surface Specification	Under **Surface Specification**, specify whether the surface is horizontal or has a vertical surface tilt. If it has a vertical tilt, specify the degree of the tilt.
Face direction specified by	For surfaces with a vertical tilt, under **Face direction specified**, specify the direction of the surface by compass direction or degrees relative to south.
Time, Date, Month	Specify the time of day, date of the month, and month of the year for the calculation.
Latitude, Longitude	Specify the location latitude and longitude for the calculation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *solar_flux_calculator.py*:

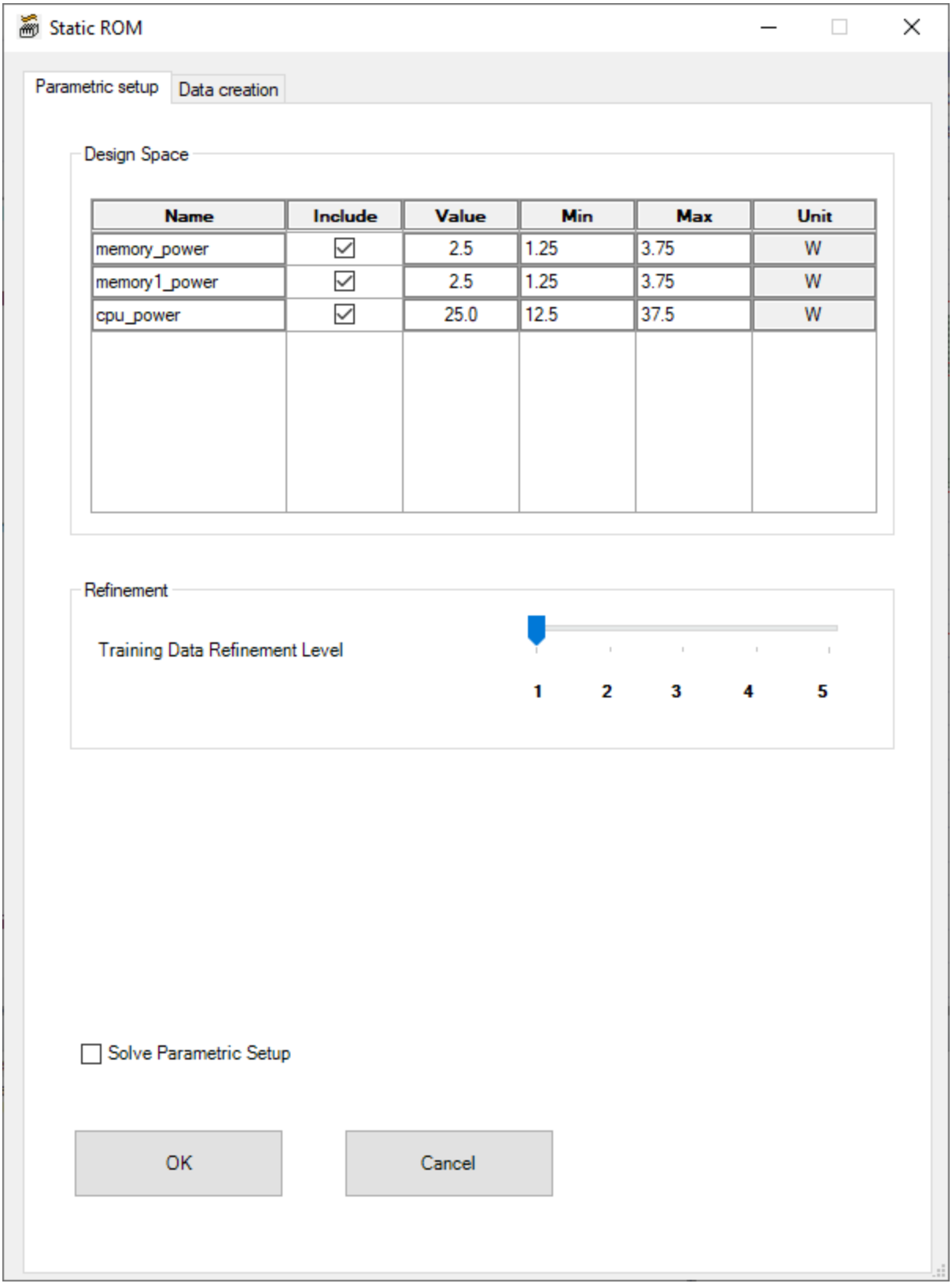
The following is the toolkit's scripting information.

```
 data = ["ng",
         'Horizontal_checked:True',
         'Verticaltilt:90',
         'compass_checked:True',
         'compass_direction:S',
         'angle_relative_to_south:0',
         'Hour:12',
         'Minute:0',
         'Zone:EST',
         'Month:JAN',
         'Date:1',
         'Latitude:23.5',
         'Longitude:75',
         'Longitudedirection:W'
         ]
 oDesign.RunToolkit("SysLib", "Modeling/solar_flux_calculator",
[])
 oDesign.RunToolkit("SysLib", "Modeling/solar_flux_calculator",
["ng"]
 oDesign.RunToolkit("SysLib", "Modeling/solar_flux_calculator",
data)
```

Static ROM

The Static ROM toolkit allows you to extract training data from Icepak for use in Twin Builder for ROM creation and validation. On the Parametric Setup tab, configure the parametric trials with the desired design variables. After running the parametric trials, use the Data creation tab to export the training data. In Twin Builder, use the data to build and validate a static ROM.

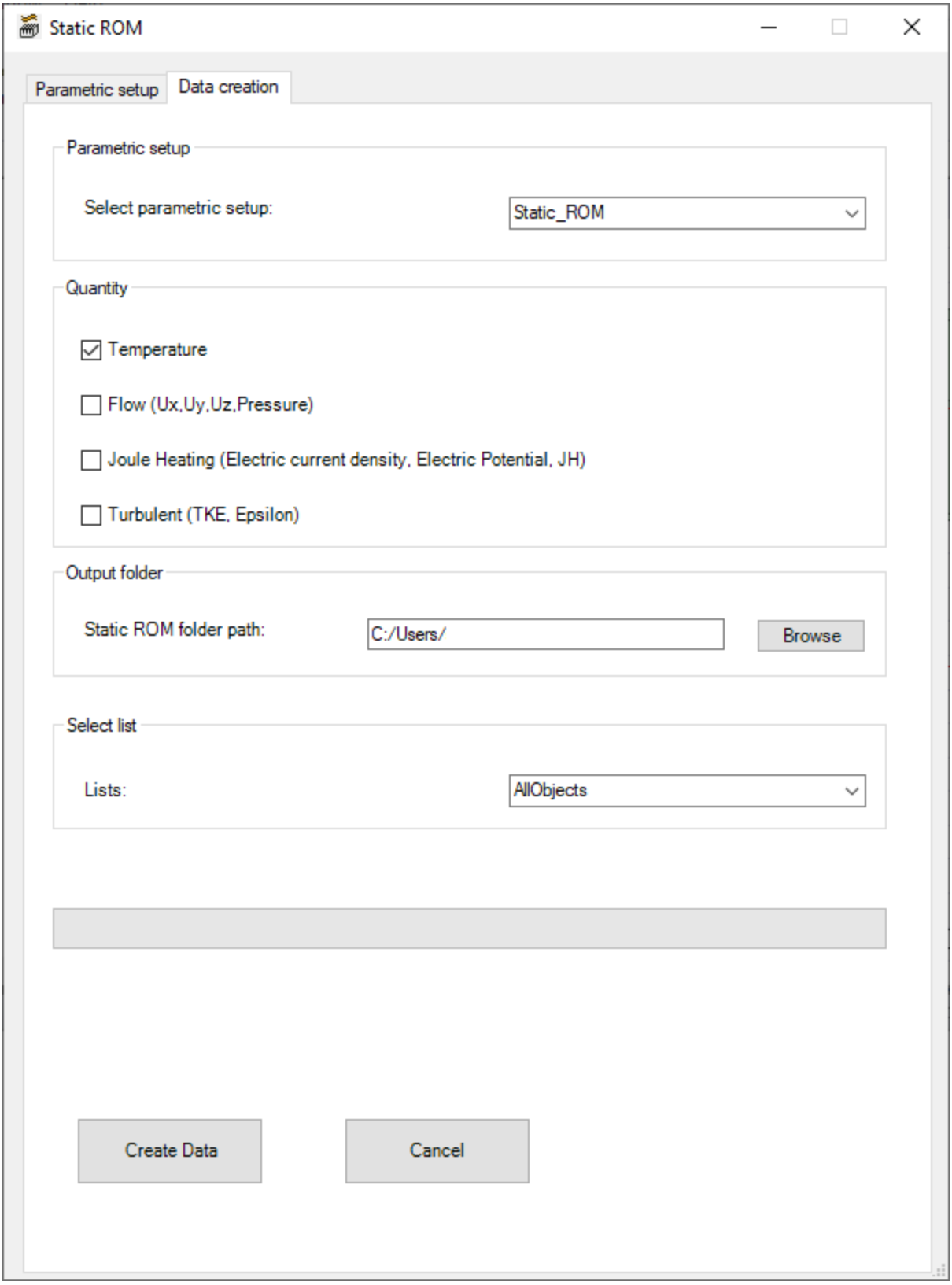
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Setup Tab	
Design Space	**Design Space** displays the design variables. Select **Include** for each variable to include in the parametric setup. Also define the **Value**, **Min**, **Max**, and **Unit** for each variable.
Refinement	Under **Refinement**, position the **Training Data Refinement Level** slider bar to the desired level. By default, Icepak creates n+2 trials, where n equals the number of variables included. That number is multiplied by the refinement level to determine the total number of trials in the design of experiments.
Solve Parametric Setup	Enable **Solve Parametric Setup** to analyze the design of experiments automatically.
OK/Cancel	Click **OK** to create and analyze the design of experiments (if **Solve Parametric Setup** is enabled). Click **Cancel** to close the toolkit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



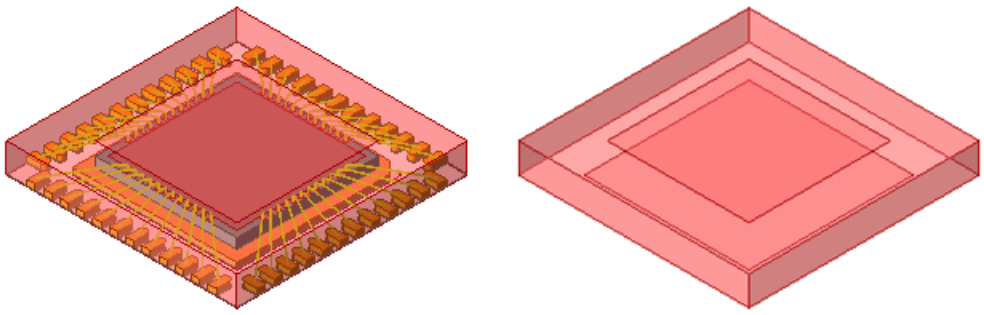
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric setup	Under **Parametric Setup**, select the parametric setup to be used for data creation.
Quantity	Under **Quantity**, select the quantities for which data will created.
Output folder	Under **Output folder**, specify the folder name and click **Browse** to navigate to the desired location.
Select list	Under **Select list**, select the list of objects to be included in the ROM.
Create Data/Cancel	Click **Create Data** to export the ROM data for use in Twin Builder. Click **Cancel** to close the toolkit.

STM Network Extraction

Surrogate Thermal Network Builder creates, trains, and exports a Delphi-like Surrogate Thermal Model (STM) that is boundary condition-independent and can be used to emulate the thermal behavior of the integrated circuit (IC) package. The toolkit allows you to select specific geometric features from the Detailed Thermal Model (DTM), including external faces and heat sources. These features have two primary purposes: to inform the DTM simulations for data production and to fully define the STM's external geometry.

The Surrogate Thermal Network Builder process consists of creating parametric trials for steady-state and transient designs to generate designs containing an STM that feature a network based on the training data. The simplified STM design (geometry and boundary conditions) enables faster, shorter meshing and analysis times. The following image displays a QFN package as a detailed thermal model (left) and surrogate thermal model (right).



The Surrogate Thermal Network Builder workflow consists of three main steps:

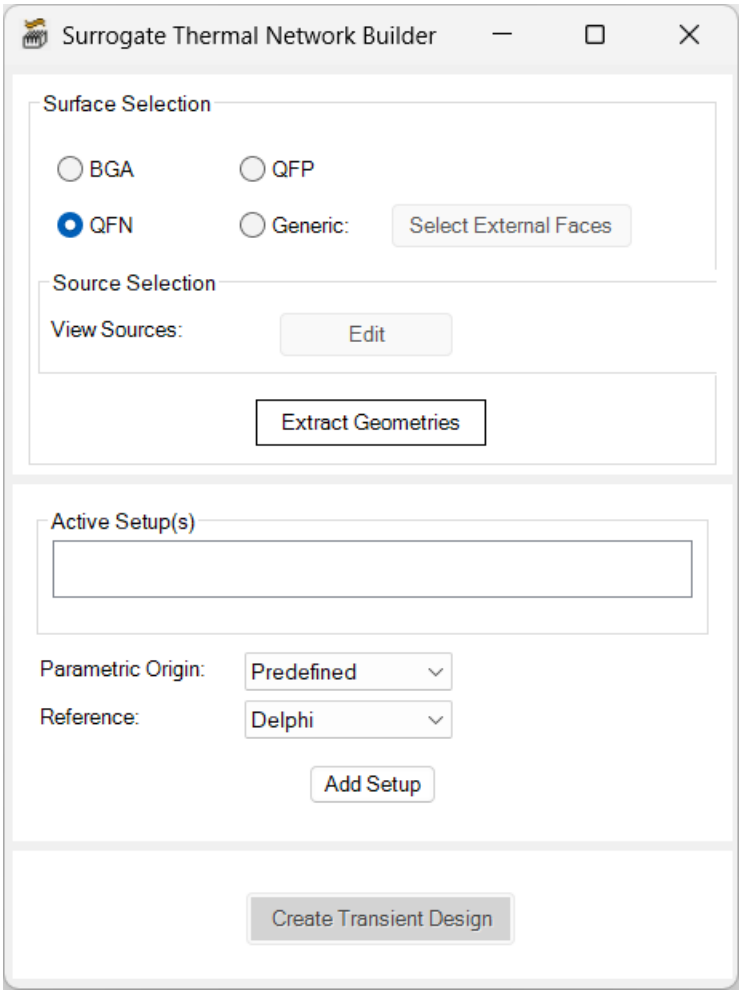
1. Geometry extraction: after specifying the package type under **Surface Selection**, click **Extract Geometries** to capture geometries and group them.
2. Setup creation: after specifying the parametric origin and associated settings, click **Add Setup** to add the setup to the **Active Setup(s)** list. Then, right-click the setup and select

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Setup to initiate a three-step process. Icepak specifies thermal boundary conditions, assigns monitors, and creates a parametric setup. After the process is complete, all of these items appear in the **Project Manager**.

3. Transient design creation: after running the steady-state to create the training data, click **Create Transient Design** to generate a transient design based on the selected steady-state design. This includes a consistent set of thermal boundary conditions and monitor assignments, and parametric setups. In addition, the solver settings are automatically updated to enable data generation for a dynamic surrogate model.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

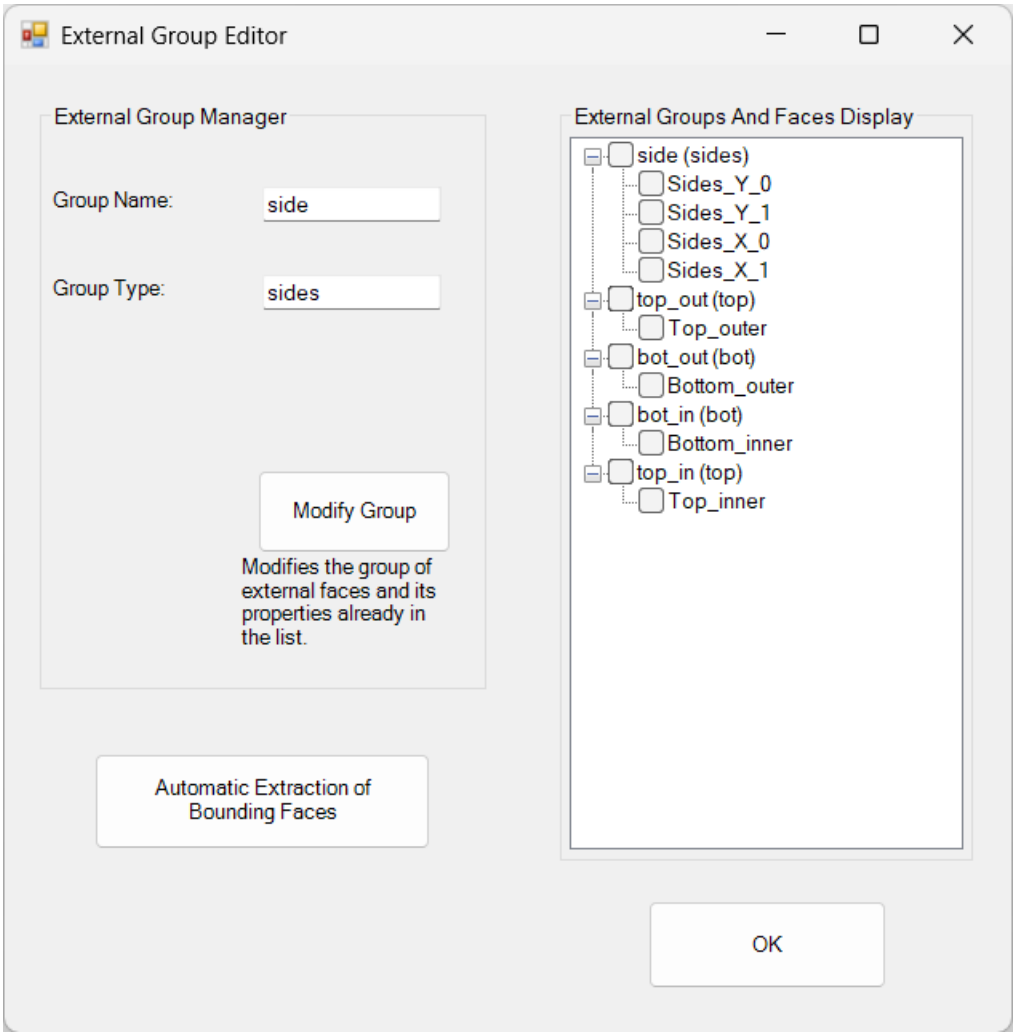
Surface Selection	Under **Surface Selection**, select the package type for the network extraction. Select **Generic** for packages that are not BGA, QFP, or QFN created using an **Icepak** > **Toolkits** > **Geometry** toolkit. For generic packages, click **Select External Faces** to export groups of external faces manually or automatically and **Select Internal Faces** to manually export groups of internal faces. See "External Faces Editor" on the facing pagefor more information.
Source Selection	Click **View Sources** to open the **Edit Source** dialog box. Sources present in the design are displayed in the list. If needed, select a source and click **Delete Selected Source** to remove it from the analysis. Click **Refresh Sources** to update the list of sources. Click **OK** to close the **Edit Source** dialog box.
Extract Geometries	Click **Extract Geometries** to finalize the collection of external faces, internal faces, heat sources, and their related properties. When the process is complete, the **Extract Geometries** button appears green.
Active Setups	**Active Setups** lists the parametric setups added to the design. See "Active Setup Options" on page 3-514 for more information.
Parametric Origin	For **Parametric Origin**, select **Predefined**, **Import from File**, or **Sampling**.
Predefined	For **Predefined**, select **Delphi**, **Jedec**, or **Demo**. **Predefined** is only available for packages with one heat source. If your package has multiple heat sources, this option is not available.
Import from File	For **Import from File**, click **Browse** to open the **Select CSV file** dialog box. Navigate to your working directory, select the .csv file, and click **Open**.
Sampling	For **Sampling**, enter a number of **Samples** and the **Standard Deviation**. If needed, enable **Include Ambient Temperature**. Also, enabled **Uniform Sampling** to open the **Configure Uniform Sampling Ranges** to specify a range of values for source power and ambient temperature (if enabled). See "Configure Uniform Sampling Ranges" on page 3-516 for more information.
Add Setup	Click **Add Setup** to add the specified parametric setup to the design.
Create	**Create Transient Design** is enabled after you create at least one active

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient Design	setup. Click **Create Transient Design** to generate a transient design based on the steady-state design. A transient parametric setup is created and is ready to analyze in the **Project Manager**.

External Faces Editor

In the **External Faces Editor**, use the **External Group Manager** to manually create groups of external faces based on geometry selected from the **3D Modeler** window or the History tree. Alternatively, you can click **Automatic Extraction of Bounding Faces** to automatically identify the external faces touching the package's bounding box and categorize them in groups.



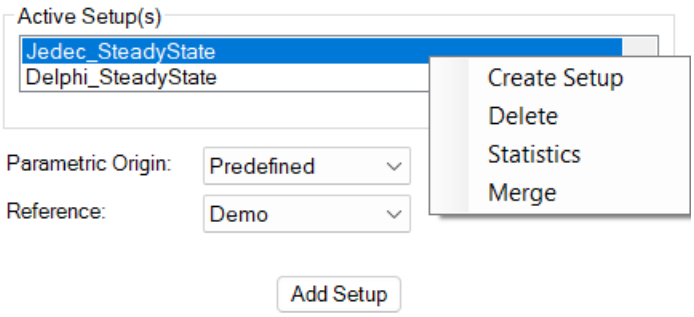
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Group Manager	
Group Name	Enter a **Group Name** for the group of external face geometry.
Group Type	Enter a **Group Type** for the group of external face geometry.
Create Group	Click **Create Group** to save the group name and/or type changes. The Create Group button appears when a group is selected in the Groups tree.
External Groups and Faces Display	
Group Tree	The **External Groups and Faces Display** tree lists the groups and associated faces you've created. If needed, select a group and modify its name or type. From a group's right-click menu, you can unite all the faces into one piece of geometry, delete the group, or delete all groups. From an external face's right-click menu, you can split the face, remove it, or move it to a different group. With multiple faces selected, you can split the faces, unite them, remove them, or move them to a different group.
Automatic Extraction of Bounding Faces	Click **Automatic Extraction of Bounding Faces** to automatically identify the external faces touching the package's bounding box and categorize them in groups. After confirming the bounding box selection, the **Get Boundaries** dialog box appears. Choose whether to create single sheet geometry for each of the six bounding surfaces or to split the sheets into more than one based on the intersecting geometry. Splitting the geometry will require more time to generate all of the geometry.
OK	After creating all external groups, click **OK** to save the external groups and close the **External Faces Editor** dialog box.

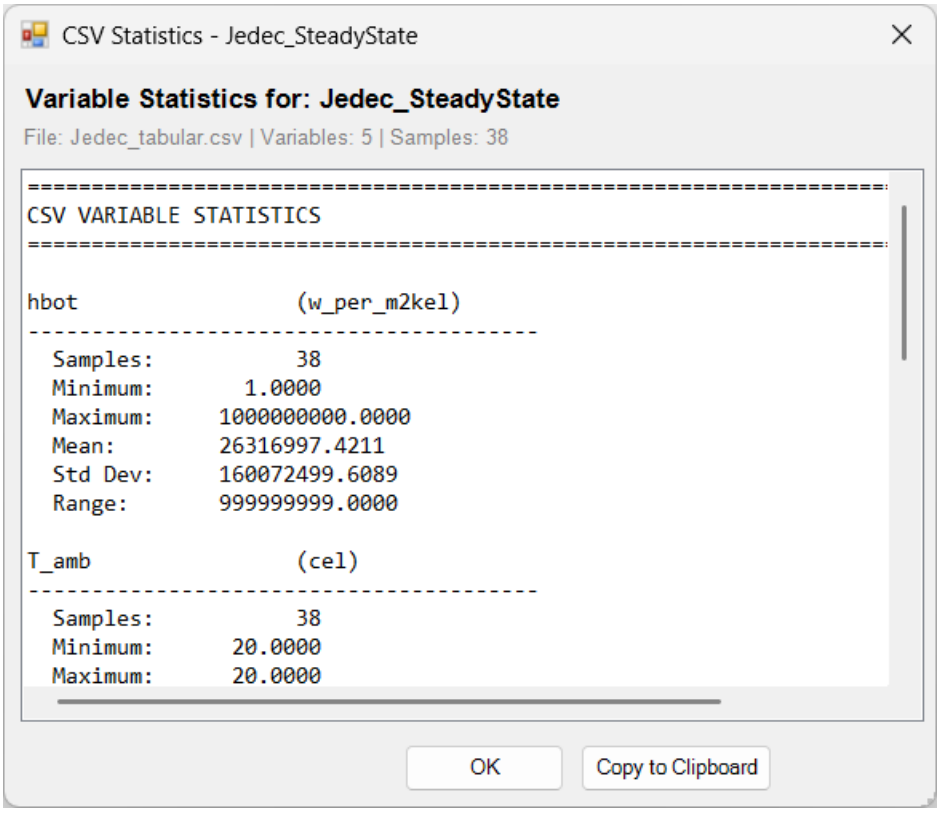
Active Setup Options

In the **Active Setup(s)** list, right-click a setup to display the following options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



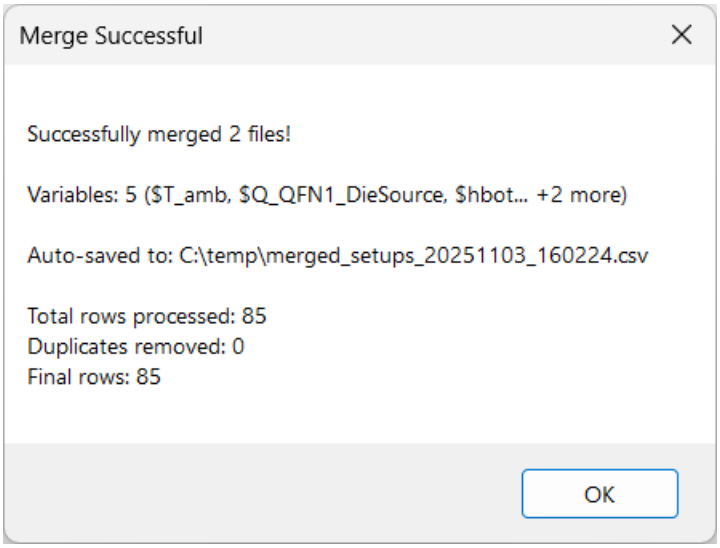
- **Create Setup**: Click **Create Setup** to generate boundary conditions, monitors, a solution setup, and parametric setup.
- **Delete**: Click **Delete** to remove the selected setup.
- **Statistics**: Click **Statistics** to display the project variable statistics for the selected setup. The dialog includes data for each variable included in the setup.



- **Merge**: If more than one setup is in the list, click **Merge** to combine the two setups into one setup.

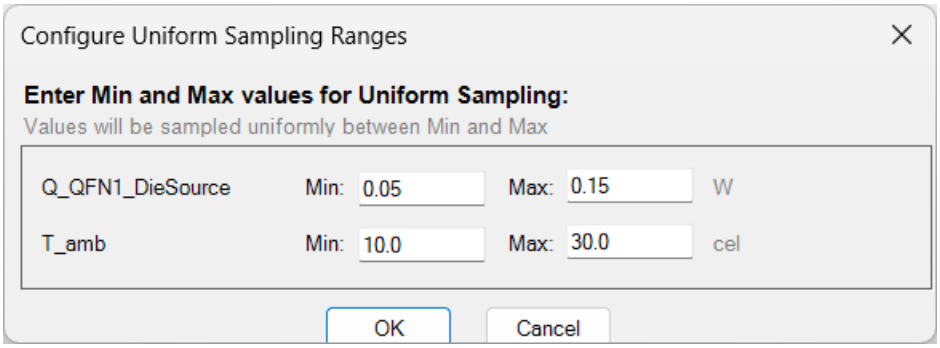
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> To merge setups, the setups' variable sets must be identical, though their order can differ. Merging accounts for any duplicates across the different setups in the list.



Configure Uniform Sampling Ranges

In the **Configure Uniform Sampling Ranges** dialog box and specify the **Min** and **Max** power and ambient temperature values (if enabled) to be sampled for the analysis. The values used are uniformly sampled between the specified minimum and maximum.



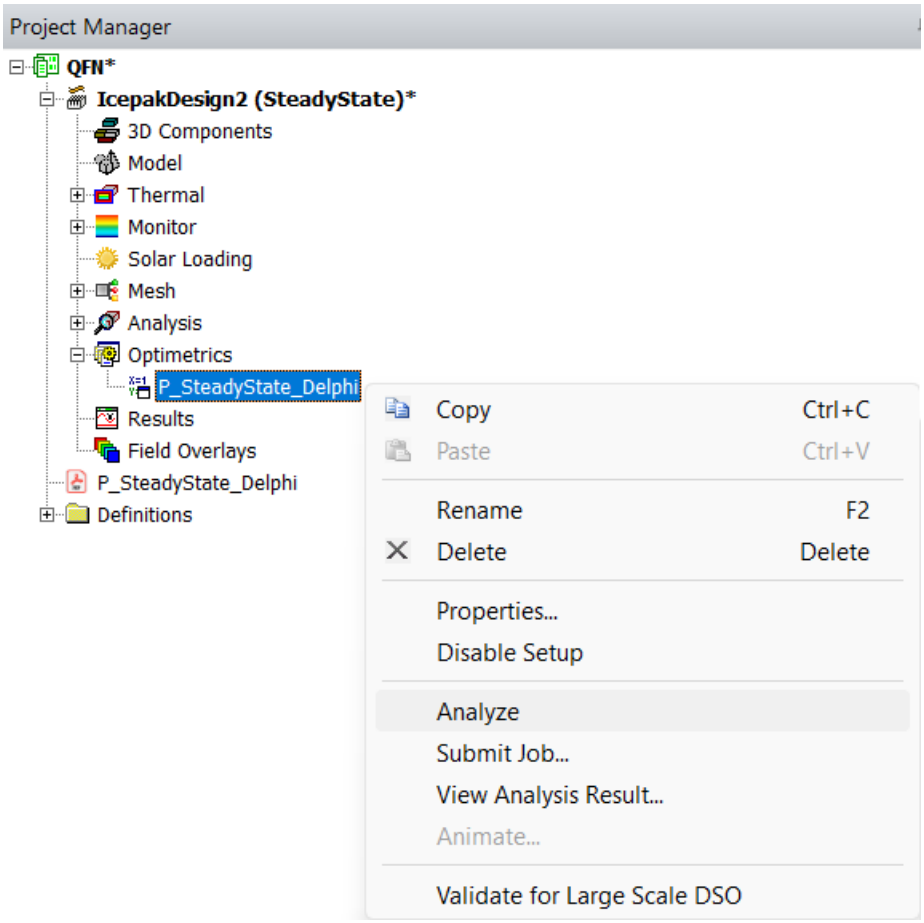
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solve the Steady-State Design for Training Data

After you click **Create Setup** from the **Active Setup(s)** options, a new design is created with boundary conditions, monitors, a solution setup, and a parametric setup. To solve for training data, right-click the parametric setup in the **Project Manager** under **Optimetrics** and select **Analyze**.

> **Note:**
>
> The naming convention for the newly created analyses is as follows. For an analysis *named P_Sampling_25Samples_SteadyState*:
>
> *P* = Parametric
>
> *Sampling* = Parametric Origin selection
>
> *25Samples* = Number of Samples (Sampling only)
>
> *SteadyState* = Solution type

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

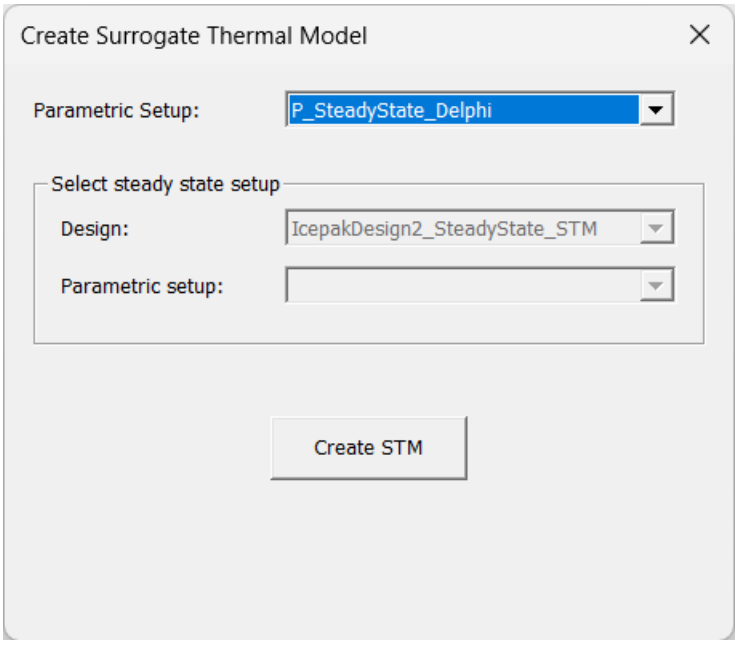


> **Note:**
>
> Monitor the status of the training process in the Progress Window.

Create the Steady-State Surrogate Thermal Model

After generating the training data by running the parametric trials, create the STM. In the **Project Manager**, right-click the design and select **Export** > **Create STM**.

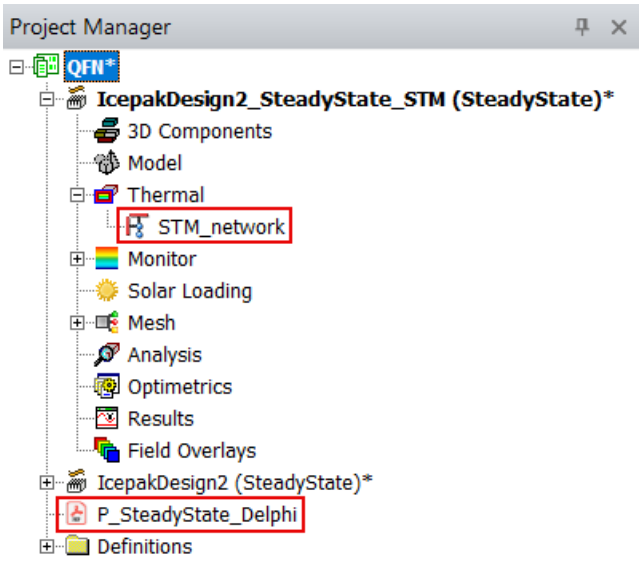
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Parametric Setup** drop-down list, select a setup. If you created more than one steady-state design for training data, under **Select steady state setup**, select a **Design** and **Parametric Setup**. Click **Create STM** to create a new design that includes a network boundary condition that represents the surrogate thermal model based on the package.

> **Note:**
>
> A PDF document that contains the STM analysis data is created. Double-click the PDF in the **Project Manager** to open it.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

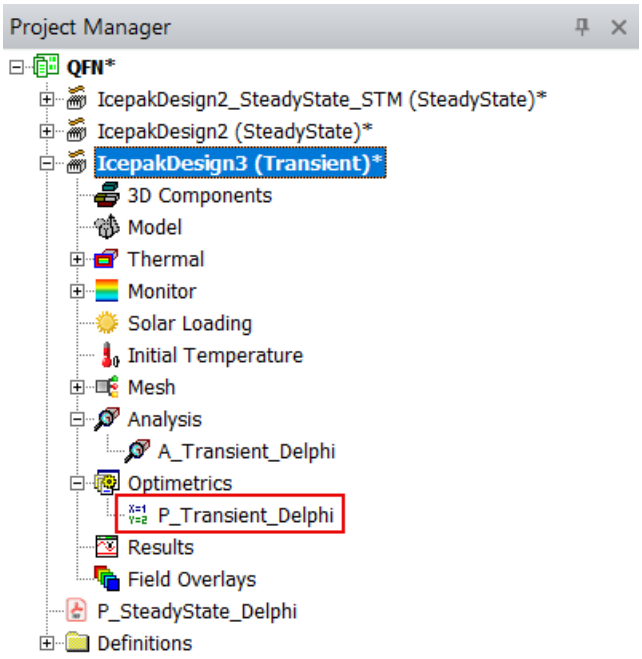


Create a Transient Design

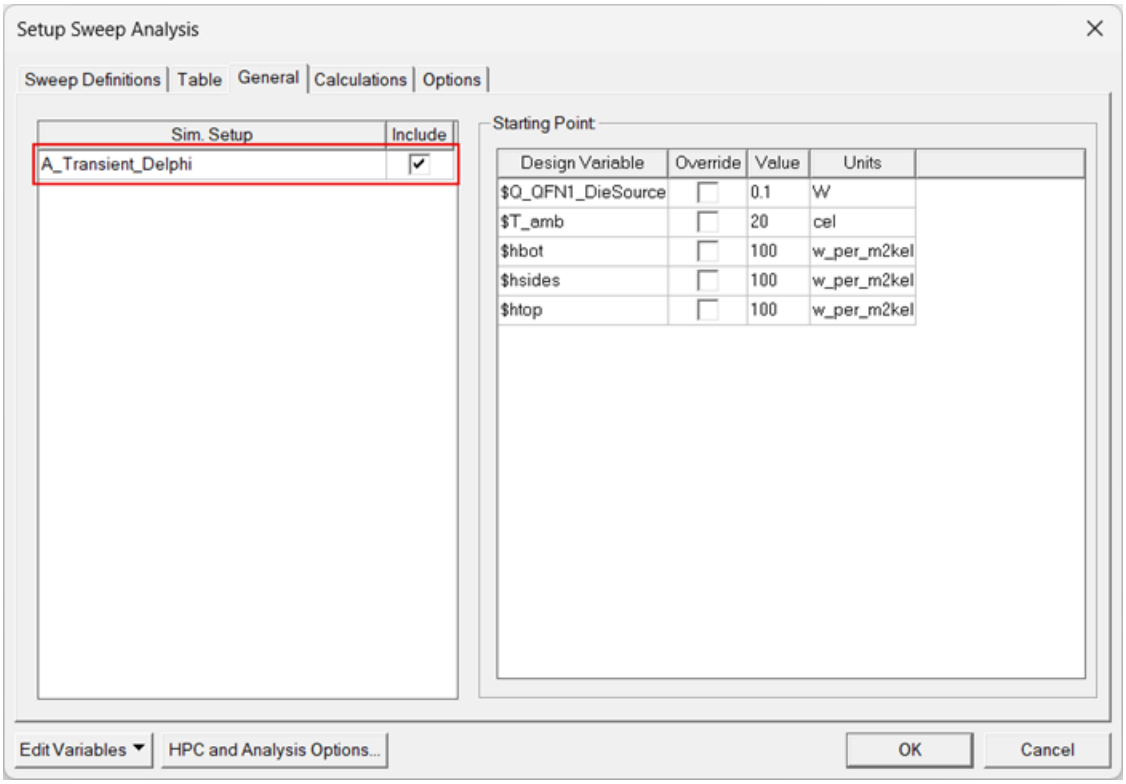
After you create an STM based on the steady-state training data, create a transient design to create transient training data. Double-click the newly created steady-state design with the STM to make it the active project. In the **Surrogate Thermal Network Builder** dialog box, click **Create Transient Design**.

A new transient design is generated with the same boundary conditions, monitors, solution setup, and parametric trials.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under **Optimetrics**, double-click the parametric setup to open the **Setup Sweep Analysis** dialog box. On the **General** tab, enable **Include** next to the parametric analysis.

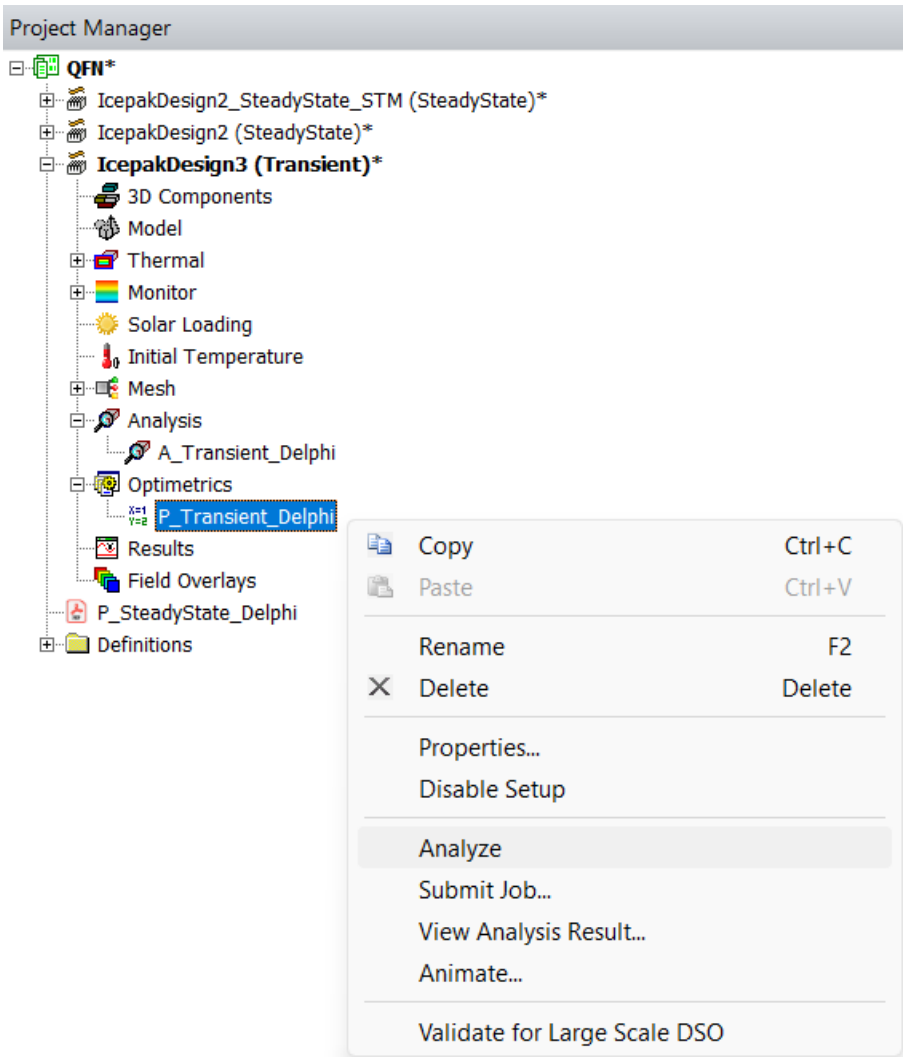


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Solve the Transient Design for Training Data

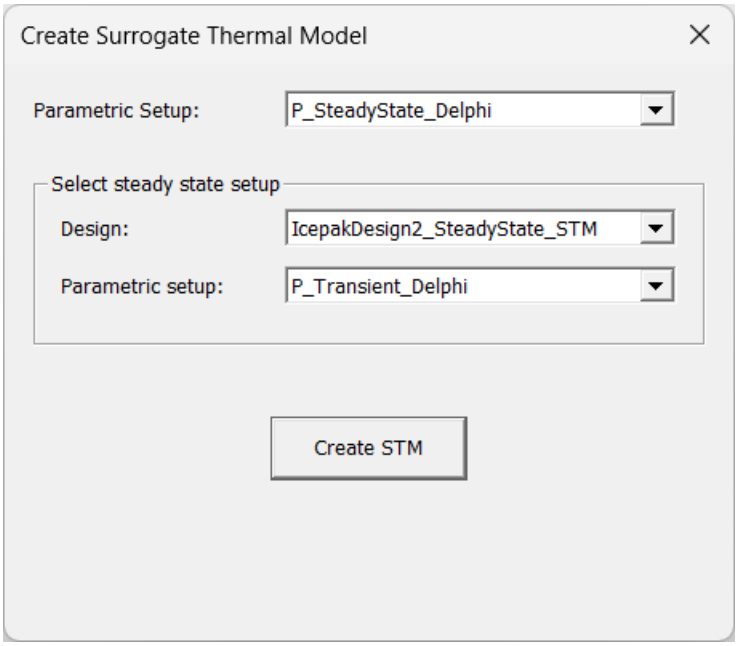
To solve for training data for the transient design, right-click the parametric setup in the **Project Manager** under **Optimetrics** and select **Analyze**.



Create the Transient Surrogate Thermal Model

After generating the training data by running the parametric trials, create the STM. In the **Project Manager**, right-click the design and select **Export** > **Create STM**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



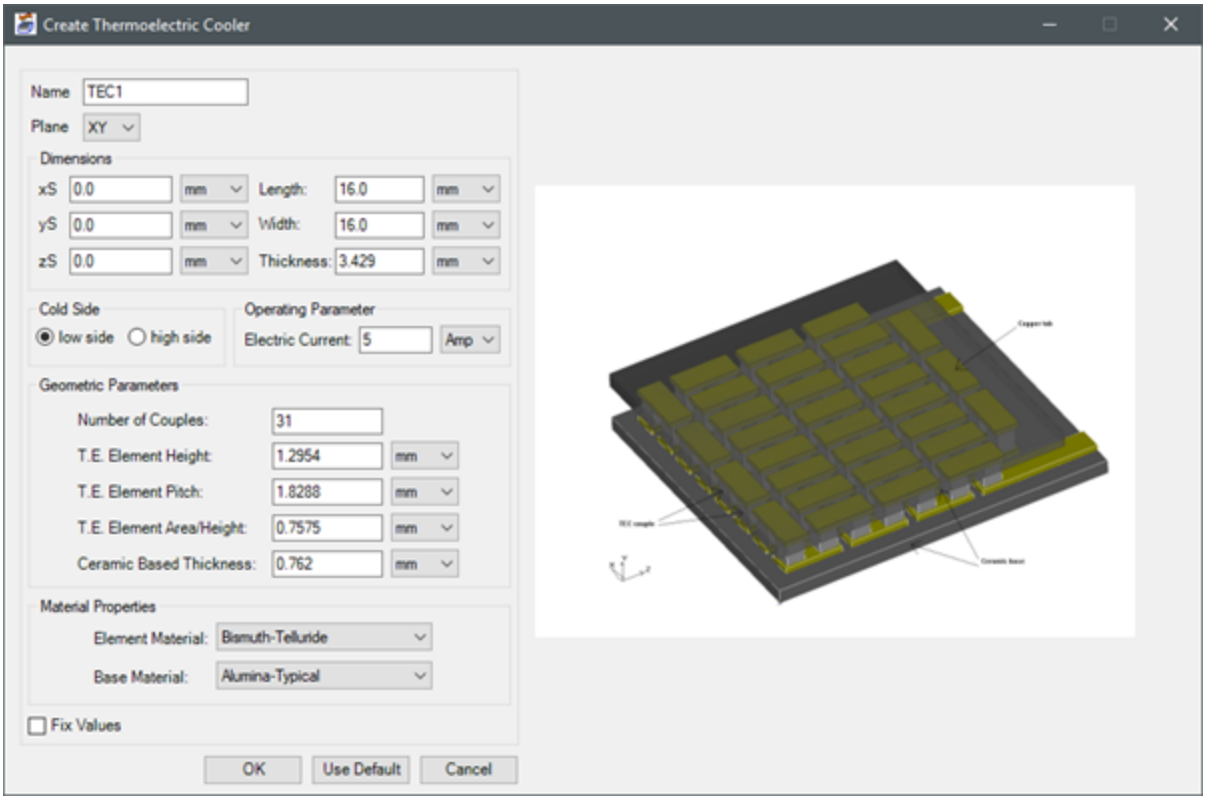
From the **Parametric Setup** drop-down list, select the steady-state setup (if there are multiple). Under **Select steady state setup**, select the steady-state STM **Design** and transient **Parametric setup**. Click **Create STM** to create a new design that includes a network boundary condition that represents the surrogate thermal model based on the package.

> **Note:**
>
> A PDF document that contains the transient STM analysis data is created. Double-click the PDF in the **Project Manager** to open it.

Thermoelectric Cooler - Create TEC

The Create TEC toolkit enables you to create custom thermoelectric coolers (TECs) that are not included in the standard library.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Enter a name for the TEC.
Plane	Specify the plane on which to place the TEC.
Dimensions	Specify the size of the TEC. Change the unit from the drop-down list if necessary.
Cold Side	Select the cold side of the TEC.
Operating Parameter	Specify the current that flows through the TEC.
Geometric Parameters	Specify the number of couples; the height, pitch, and area/height of the thermoelectric elements; and the thickness of the ceramic base. Change the unit from the drop-down list if necessary.
Material Properties	Specify the material for the elements and base.

Iron Python Script – *TEC.py*:

The following is the toolkit's scripting information.

```
data = ["ng",
        'length_unit:meter',
        'power_unit:W',
        'Plane:XY',
        'compass_direction:S',
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

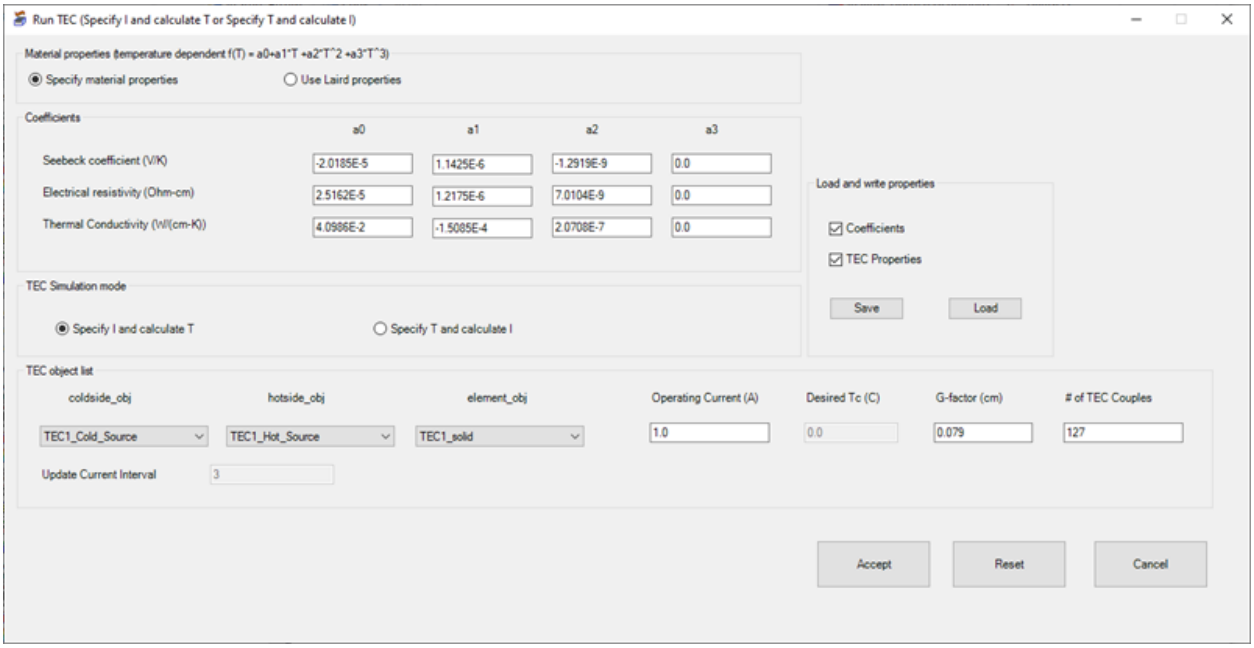
```
        'xS:0.0',
        'yS:0.0',
        'zS:0.0',
        'Length:0.016',
        'Width:0.016',
        'Thickness:0.003429',
        'Cold Side:Low',
        'Current:5',
        'num Couples:31',
        'Element Height:0.0012954',
        'Element Pitch:0.0018288',
        'Element Area:0.0007575',
        'Ceramic Thickness:0.000762',
        'Element Material:Bismuth-Telluride',
        'Base Material:Alumina-Typical'
        ]
 oDesign.RunToolkit("SysLib", "Modeling/Thermoelectric
Cooler/Create_TEC", [])
 oDesign.RunToolkit("SysLib", "Modeling/Thermoelectric
Cooler/Create_TEC", ["ng"]
 oDesign.RunToolkit("SysLib", "Modeling/Thermoelectric
Cooler/Create_TEC", data)
```

Thermoelectric Cooler - Run TEC

> **Note:**
>
> The Run TEC toolkit is not supported on Linux systems.

The Run TEC toolkit enables you to run an analysis of a design that includes a thermoelectric cooler.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Material properties	Select **Specify material properties** to define the **Coefficients** values or select **Use Laird properties** to use predefined Laird values.
Coefficients	If you selected **Specify material properties**, enter values for **Seebeck coefficient (V/K)**, **Electrical resistivity (Ohm-cm)**, and **Thermal conductivity (W/(cm-K))** under **a0**, **a1**, **a2**, and **a3**. The default coefficients shown are compatible with most Marlow TECs.
TEC Simulation mode	Select **Specify I and calculate T** to enter an **Operating Current (A)** value and calculate a **Desired Tc (C)** during the solution. Or select **Specify T and calculate I** to enter a **Desired Tc (C)** and calculate **Operating Current (A)** during the solution.
TEC object list	Define the following parameters: • Select objects for **coldside_obj**, **hotside_obj**, and **element_obj**. • **Operating Current (A)**: Only available for the **Specify I and calculate T** simulation mode. If you loaded a TEC from the standard library, the maximum operating current is shown. Note that modifying this value will not be saved the next time the panel is open. • **Desired Tc (C)**: Only available for the **Specify T and calculate I** simulation mode. • **G-factor (cm)**: If you loaded a TEC from the standard library, the G-factor is shown.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• **# of TEC Couples**: If you loaded a TEC from the standard library, the # of TEC Couples is shown. • **Update Current Interval**: If you selected **Specify T and calculate I**, enter how many iterations you would like to wait before the TEC current is updated.
Load and write properties	If you want to save the coefficients and/or TEC properties and load them in a later session, select **Coefficients** and/or **TEC properties** under **Load and Write properties** and click **Save**. Use the **TEC coefficients data** panel to navigate to a location to save the file.
Accept	Click **Accept** to run the analysis.
Reset/Cancel	Click **Reset** to restore the default values. Click **Cancel** to close the Run TEC toolkit without running the analysis.

Icepak Productivity Toolkit

Icepak has the following productivity toolkits.

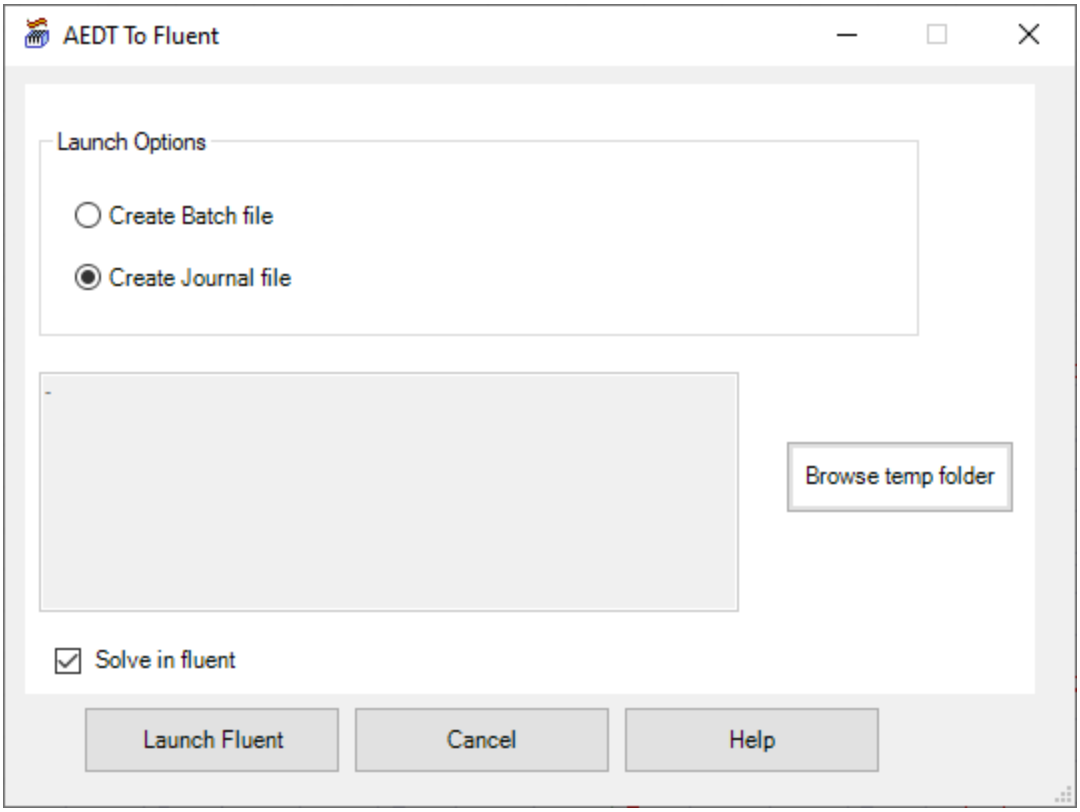
- AEDT to Fluent
- Contour File Export
- Create Conduction Design
- Debug Divergence
- Dimensionless Number Calculator
- DTM Python Verification Form
- Fluent Mesh Import
- HTC Coefficient Calculator
- Randomize Object Colors
- Snapping

AEDT to Fluent

AEDT to Fluent allows you to launch Ansys Fluent to mesh and solve a design using a batch or journal file.

> **Note:** The AEDT to Fluent toolkit is only available for the Windows version of the Electronics Desktop. Also, the following environment variables must be defined:
>
> ```
> PRESERVE_SOLVER_INPUT = 1
> PRESERVE_SOLVER_FILES = 1
> ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



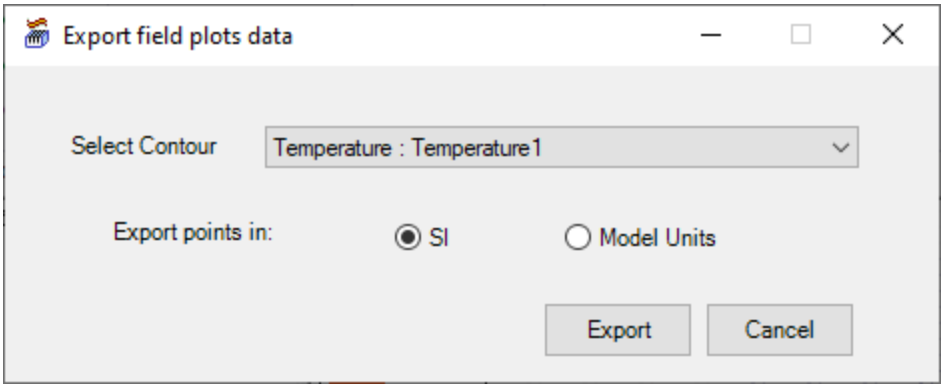
Launch Options	
Create Batch file	Select **Create Batch file** to create a batch file to execute later to launch Fluent.
Create Journal file	Select **Create Journal file** to create a journal file and launch Fluent and mesh to design. Select **Solve in fluent** to also solve the design.
Browse temp folder	Click **Browse temp folder** to select the directory in which to create the batch or journal file.
Solve in fluent	If **Create Journal file** is selected, select **Solve in fluent** to also solve the design in addition to meshing.
Launch Fluent, **Cancel**, **Help**	Click **Launch Fluent** to create the batch or journal file. Click **Cancel** to close the toolkit without launching Fluent. Click **Help** to view information about the toolkit,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contour File Export

Contour File Export creates a tabulated file containing quantity data based on the selected field plot. The data can be presented with SI units or the model's selected unit.

> **Note:** After you create the field plot, you must save the project before opening the toolkit.

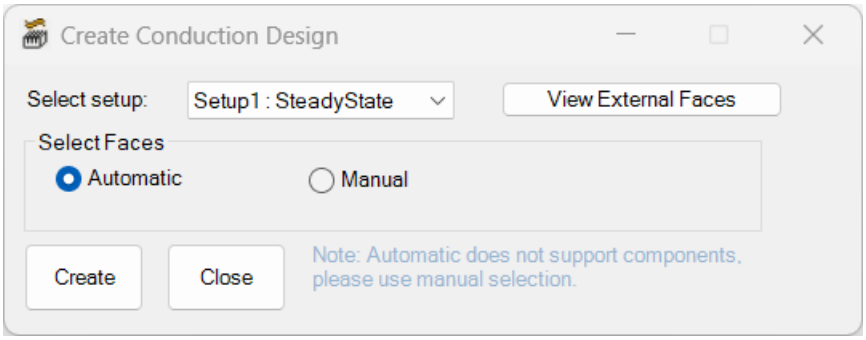



```
Scalar data "Temperature"
-9.0719999999999967e-03 3.0860000000000002e-03 -6.3000000000000000e-02  4.7736890728759668e+01
-9.0719999999999967e-03 3.0860000000000002e-03 -6.2500000000000014e-02  4.7108666992187523e+01
-9.0719999999999967e-03 3.2370454545454547e-03 -6.3000000000000000e-02  4.7652801168684441e+01
-6.8000000000000005e-02 3.0860000000000002e-03 -6.3000000000000000e-02  3.7044439936037435e+01
-6.8000000000000005e-02 3.0860000000000002e-03 -6.2500000000000000e-02  4.4821069335937523e+01
-6.8000000000000005e-02 3.2370454545454543e-03 -6.3000000000000000e-02  3.5822534043058425e+01
-2.0254500000000002e-02 9.0637272727272736e-03 -1.7100000000000001e-02  4.4937768554687523e+01
-2.1000000000000001e-02 8.4163181818181828e-03 -1.7100000000000001e-02  4.4984674072265648e+01
-2.1000000000000001e-02 9.0637272727272736e-03 -1.7100000000000001e-02  4.4934899902343773e+01
-2.1000000000000001e-02 9.7111363636363645e-03 -1.7100000000000001e-02  4.4886529541015648e+01
-1.8763499999999999e-02 9.0637272727272736e-03 -1.7100000000000001e-02  4.4942315673828148e+01
-1.8018000000000003e-02 9.7111363636363645e-03 -1.7100000000000001e-02  4.4896539306640641e+01
-1.8018000000000003e-02 9.0637272727272736e-03 -1.7100000000000001e-02  4.4943170166015506e+01
-1.8018000000000003e-02 8.4163181818181828e-03 -1.7100000000000001e-02  4.4991082763671685e+01
-2.0254500000000002e-02 1.1653363636363637e-02 -1.7100000000000001e-02  4.4727502441406273e+01
-2.1000000000000001e-02 1.1005954545454546e-02 -1.7100000000000001e-02  4.4767419433593773e+01
-2.1000000000000001e-02 1.1653363636363637e-02 -1.7100000000000001e-02  4.4722924804687523e+01
-2.1000000000000001e-02 1.2300772727272728e-02 -1.7100000000000001e-02  4.4679956054687523e+01
-1.8763499999999999e-02 1.1653363636363637e-02 -1.7100000000000001e-02  4.4735955810546898e+01
-1.8018000000000003e-02 1.2300772727272728e-02 -1.7100000000000001e-02  4.4696466064453141e+01
-1.8018000000000003e-02 1.1653363636363637e-02 -1.7100000000000001e-02  4.4738214111328006e+01
-1.8018000000000003e-02 1.1005954545454546e-02 -1.7100000000000001e-02  4.4781335449218567e+01
-2.0254500000000002e-02 1.4243000000000000e-02 -1.7100000000000001e-02  4.4541436767578148e+01
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Conduction Design

Create Conduction Design allows you to generate a conduction-only copy of your design. The toolkit creates a wall boundary condition with **Heat Transfer Coefficient** enabled for the external condition. The wall boundary condition is applied to the external faces using automatic or manual face selection.



Select setup	From the **Select setup** drop-down, select the solve setup to use in generating the results for the sub-model.
Select Faces	Under **Select Faces**, select **Automatic** to automatically select external faces of all supported geometries See "Supported and Unsupported Geometry" on the facing page for more information. Select **Manual** to manually select a fewer number of external faces in the model.
View External Faces	Click **View External Faces** to display a list of all external faces in the model. See "External Faces" on the facing page for more information. If **Automatic** is selected, the **Info** dialog box displays all faces that are automatically identified. If **Manual** is selected, the dialog box displays faces that you have manually selected.
Create	Click **Create** to generate the conduction-only design based on the selected faces.
Close	Click **Close** to close the toolkit without creating a conduction-only design.

Note:

If you encounter an unhandled exception error when attempting to run the Create Conduction Design toolkit, set the following environment variable:

```
ANS_USE_ISOLATED_CLIPBOARD = 1
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

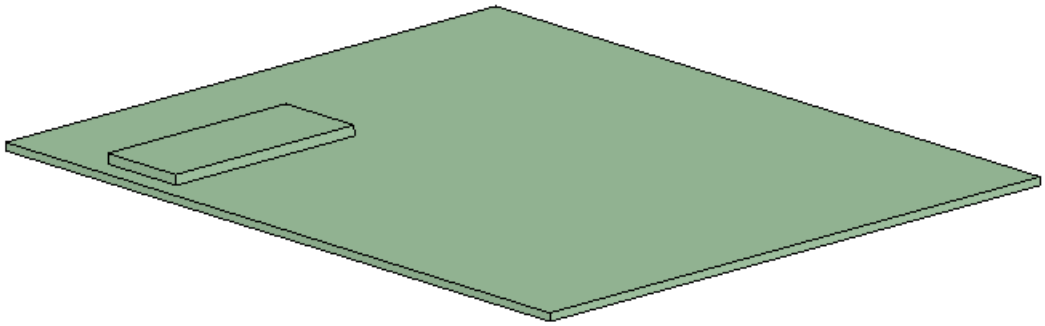
Supported and Unsupported Geometry

The Create Conduction Design toolkit supports many types of geometry but has some limitations:

- Supported: primitive objects, CAD, 3D components (Manual selection), PCB native component Bounding Box outline (Manual selection)
- Unsupported: hollow objects, subregions, and native 3D components (polyline PCBs, heatsinks, CTMs, and fans)

External Faces

External faces are sheet geometry that form the outer boundary of the model. In the following image, Box 1 is located on top of Box 2. In this example, automatic selection would identify five external faces for Box 1 since the bottom sheet coincides with the top of Box 2 and is not exposed.



Debug Divergence

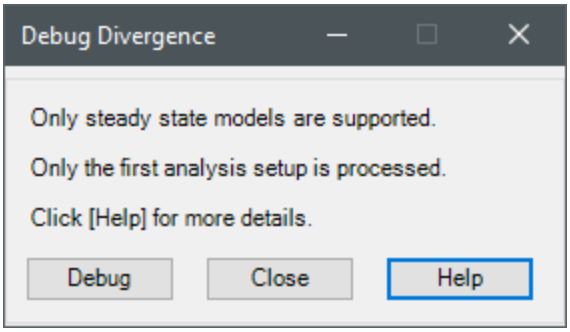
Debug Divergence creates monitor points and cut planes at the location of maximum speed and temperature. It automatically detects which variables are available and plots accordingly.

- The monitor points are located at the position of maximum speed and temperature.
- The cut planes pass through these points on the XZ and YZ planes.

The location of maximum speed and/or temperature is usually the location of solver divergence.

- The continuity and momentum equations usually diverge at the location of maximum speed.
- The energy equation usually diverges at the location of maximum temperature.

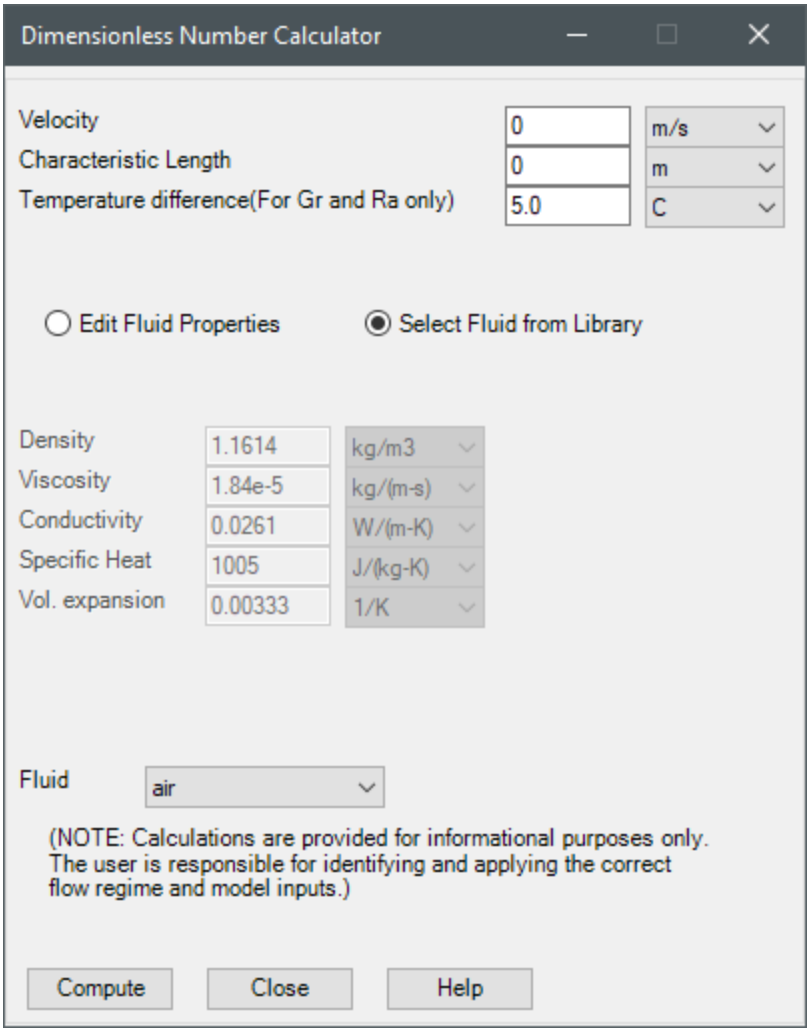
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Dimensionless Number Calculator

Dimensionless Number Calculator allows you to compute the Reynolds, Prandtl, Grashof, and Rayleigh numbers by inputting appropriate characteristic velocity, length, and temperature scales. You can define the fluid properties manually or select a fluid from the materials library.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iron Python Script – *Dimensionless_number_calculator.py*:

The following is the toolkit's scripting information.

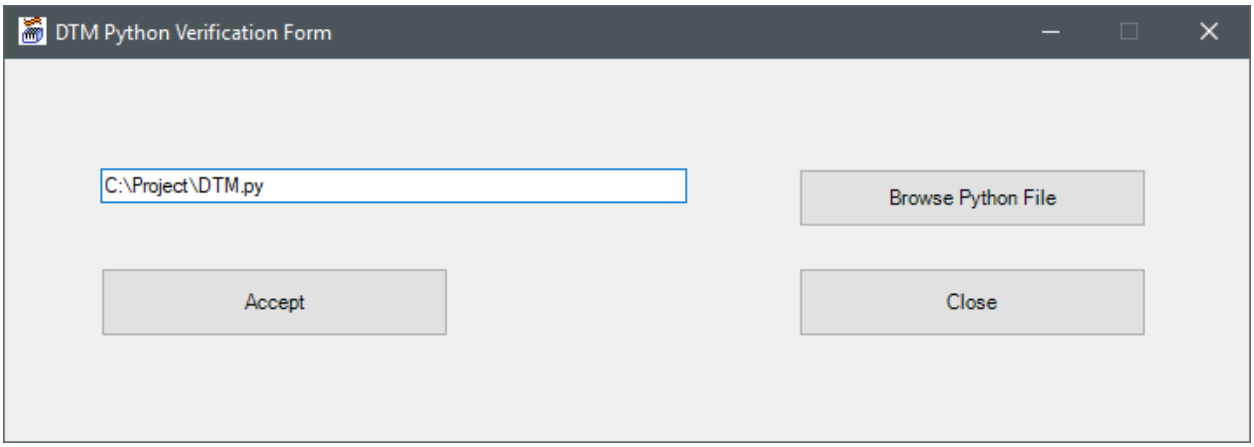
```
data = ["ng",
        'velocity_unit:m/s',
        'Characteristic Length_unit:m',
        'Temperature difference_unit:C',
        'Density_unit:kg/m3',
        'Viscosity_unit:kg/(m-s)',
        'Conductivity_unit:W/(m-K)',
        'Specific Heat_unit:J/(kg-K)',
        'Vol. expansion_unit:1/K',
        'Edit Fluid Properties:True',
        'Velocity:0',
        'Characteristic Length:0',
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        'Temperature difference:5.0',
        'Density:1.614',
        'Viscosity:1.84e-005',
        'Conductivity:0.0261',
        'Specific Heat:1005',
        'Vol. expansion:0.00333',
        'Fluid:air',
        ]
 oDesign.RunToolkit("SysLib", "Productivity/Dimensionless_
number_calculator", [])
 oDesign.RunToolkit("SysLib", "Productivity/Dimensionless_
number_calculator", ["ng"])
 oDesign.RunToolkit("SysLib", "Productivity/Dimensionless_
number_calculator", data)
```

Debug Divergence

DTM Python Verification Form allows you to confirm that the Dynamic Thermal modeling python script you created using the Thermostat for Source Control toolkit contains no errors prior to using it in a transient simulation.



Browse Python File	Click **Browse Python File** to select the control program script file you want to verify.
Accept	Click **Accept** to run the python script verification. Verification results are displayed in the Messages panel.

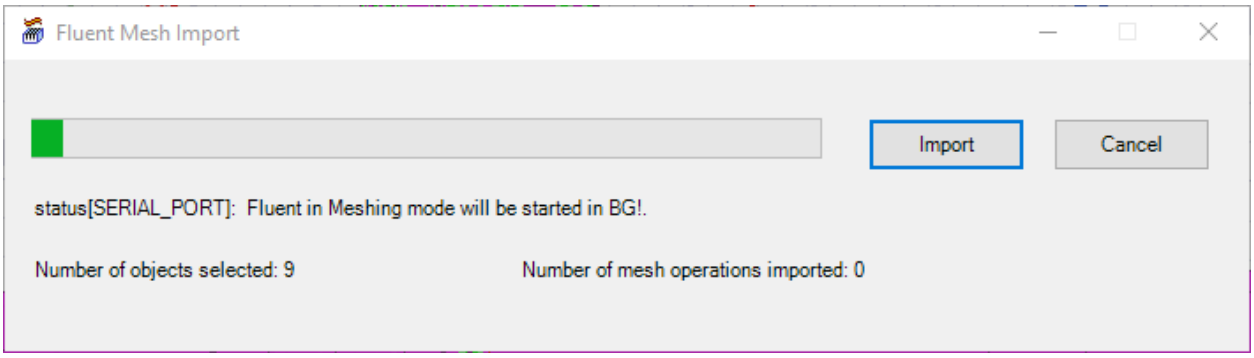
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent Mesh Import

> **Note:** To use the Fluent Mesh Import toolkit, you must install the Geometry Interfaces from the Ansys Workbench installation.

Fluent Mesh Import allows you to generate mesh for selected objects. The mesh generation occurs in the background while you continue to work in the Electronics Desktop.

Select objects in the History tree or the **3D Modeler** window and then click Import to generate the mesh files.



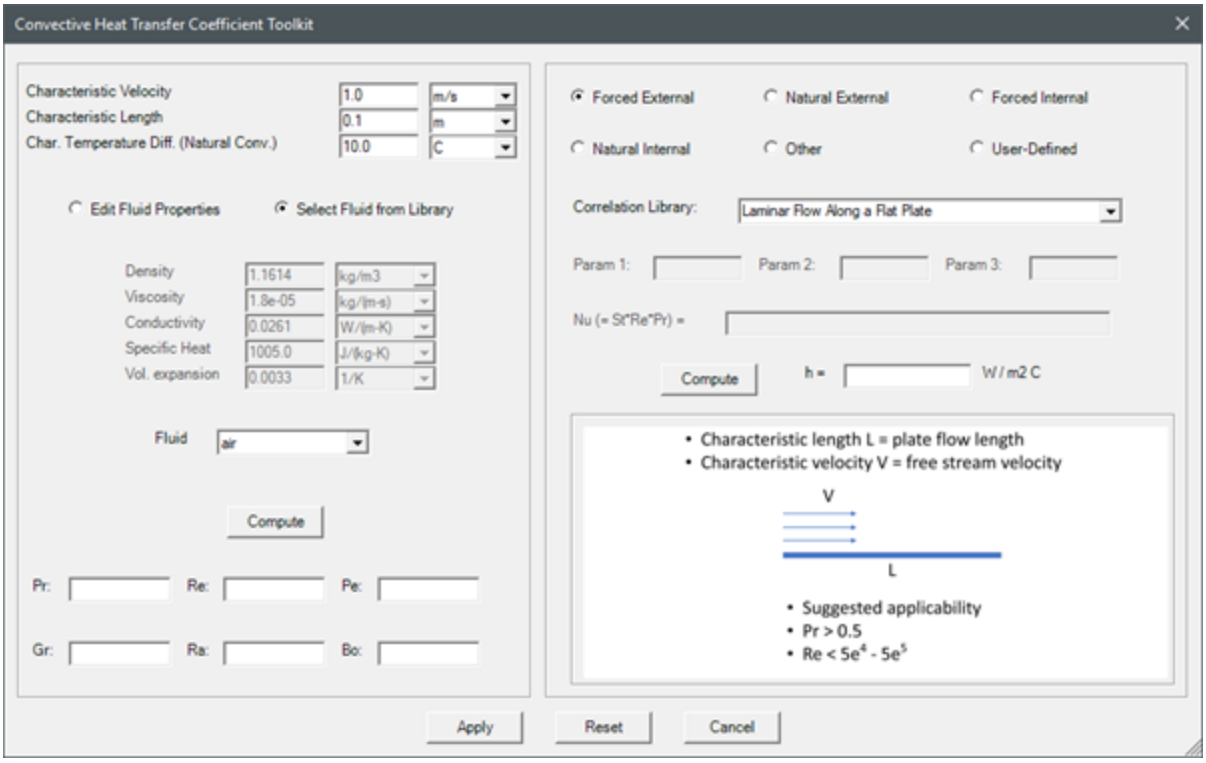
HTC Coefficient Calculator

HTC Coefficient Calculator enables you to calculate average heat transfer coefficients using well established correlations or user-defined correlations. After the heat transfer coefficient is computed, you can apply it to the selected model face. A wall thermal boundary condition is created with the heat transfer coefficient defined.

> **Note:** In Linux, you must select faces before opening the toolkit in order to assign the wall boundary condition.

> **Note:** In addition to status updates, the **Messages** window displays reference information related to the correlations used.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Characteristic Values	The **Characteristic** values specify the velocity, length, and temperature difference (natural convection).
Fluid Properties	Select **Edit Fluid Properties** and manually enter the fluid specifications, or select a fluid from the project's material library in the Fluid drop-down list.
Dimensionless Numbers Calculator	Click **Compute** to calculate the dimensionless numbers based on the fluid properties.
Flow Configuration	Select a flow configuration option to define the **Correlation Library**.
Correlation Library	The **Correlation Library** lists the correlations available for the selected flow configuration. The schematic displayed provides guidance on the suggested applicability of the correlation. Use the schematic to verify the characteristic length, velocity, and temperature.
Param 1, **2**, **3**	If needed, specify any additional parameters in **Param 1**, **Param 2**, and **Param 3** based on the selected correlation.
User-Defined Correlation	To define a custom correlation, select the **User-Defined** flow configuration and enter the correlation values in the enabled field.
Compute	Click **Compute** to calculate the heat transfer coefficient value.
Apply	Click **Apply** to assign the heat transfer coefficient to the selected

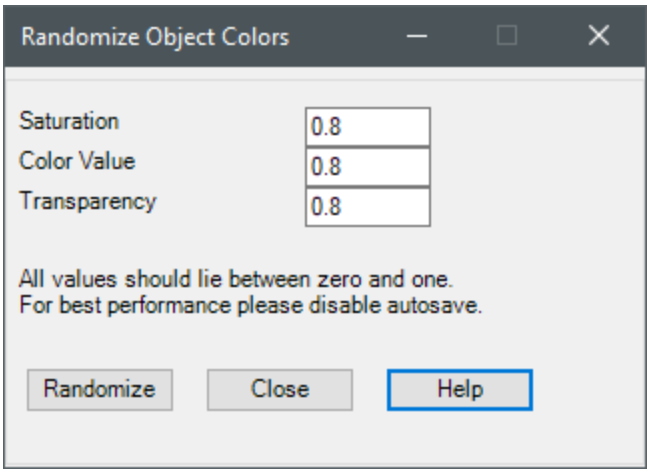
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	model face. A wall thermal boundary condition is created with the heat transfer coefficient defined.

Randomize Object Colors

Randomize Object Colors assigns random colors to all objects in the model to avoid identical colors from repeating. The same **Saturation**, **Color Value**, and **Transparency** are applied to all objects; however, the hue is different for each object.

> **Note:** For models with many objects, it is best to disable autosave. From the **Tools** menu, select **Options** > **General Options**. In the **Options** dialog box, expand **General** and select **Desktop Configuration**. Under **Save Options**, disable **Do Autosave**.



Iron Python Script – *Randomize_Object_Colors.py*:

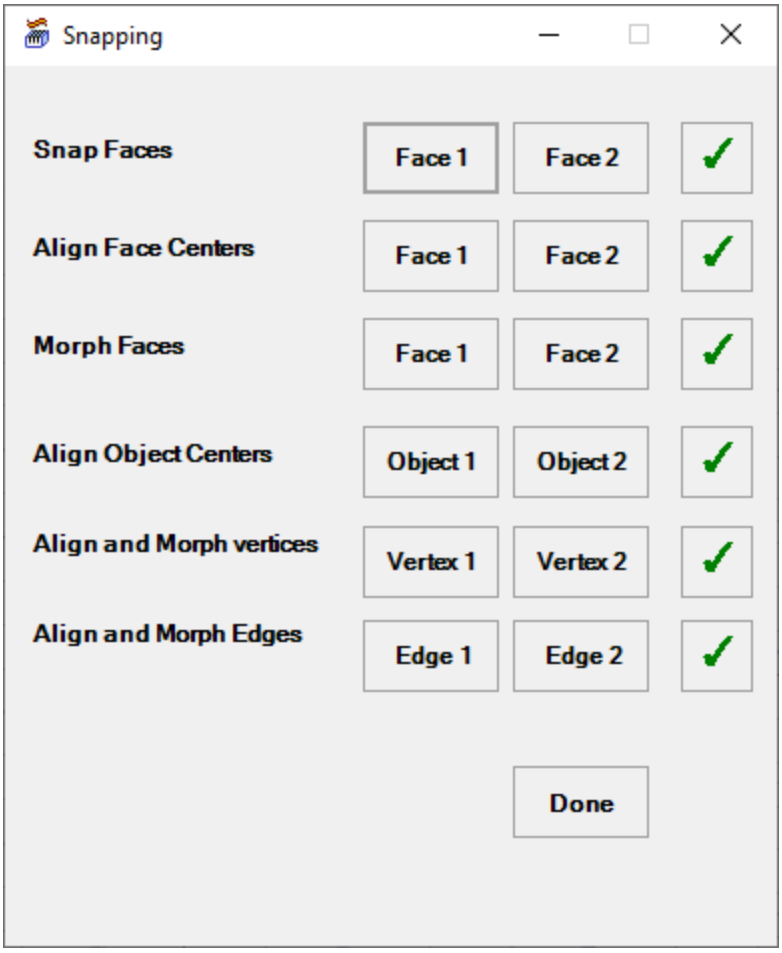
The following is the toolkit's scripting information.

```
 data = ["ng",
     'saturation:0.8',
     'color:0.8',
     'transparency:0.8',
     ]
 oDesign.RunToolkit("SysLib", "Productivity/Randomize_Object_
Colors", [])
 oDesign.RunToolkit("SysLib", "Productivity/Randomize_Object_
Colors", ["ng"])
 oDesign.RunToolkit("SysLib", "Productivity/Randomize_Object_
Colors", data)
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Snapping

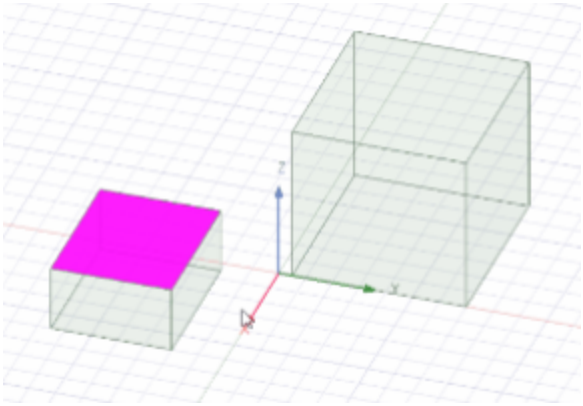
Snapping provides multiple ways to reposition geometry objects, faces, and edges.

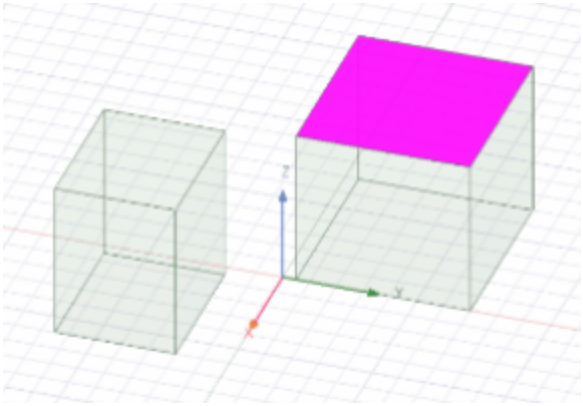


Snap Faces	Face 1 is aligned with Face 2.
Align Face Centers	Face 1 center is aligned with Face 2.
Morph Faces	Face 1 is morphed to match the size of Face 2.
Align Object Centers	Object 1 center is aligned with Object 2 center.
Align and Morph Vertices	Vertex 1 is aligned with and morphed to match the size of Vertex 2.
Align and Morph Edges	Edge 1 is aligned with and morphed to match the size of Edge 2.
Done	Click **Done** to close the toolkit.

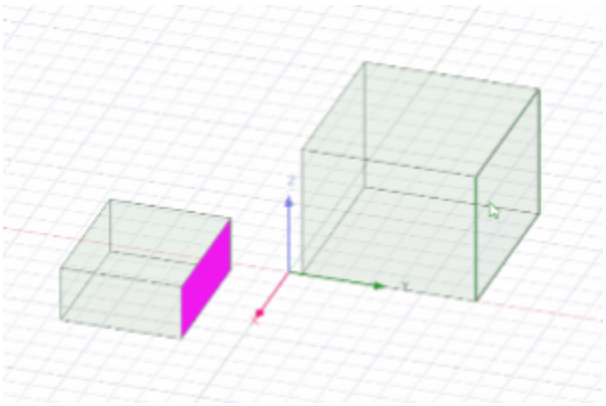
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Snap Faces

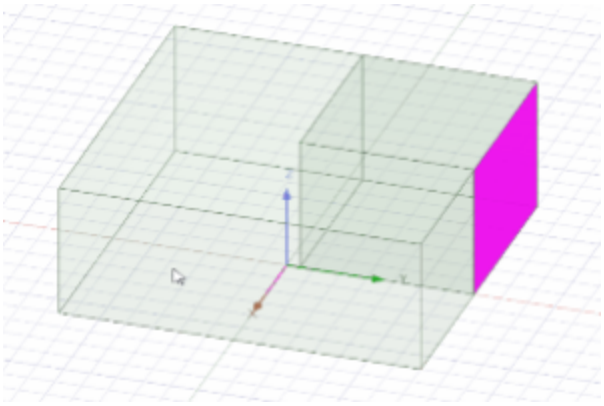




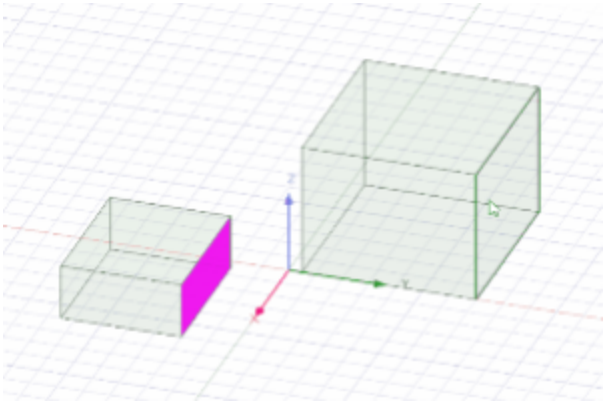
Align Face Centers

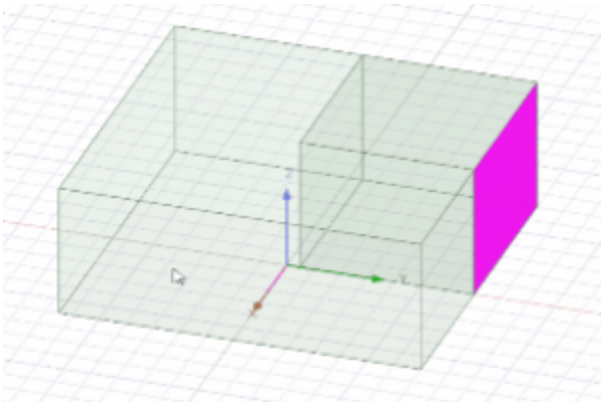


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



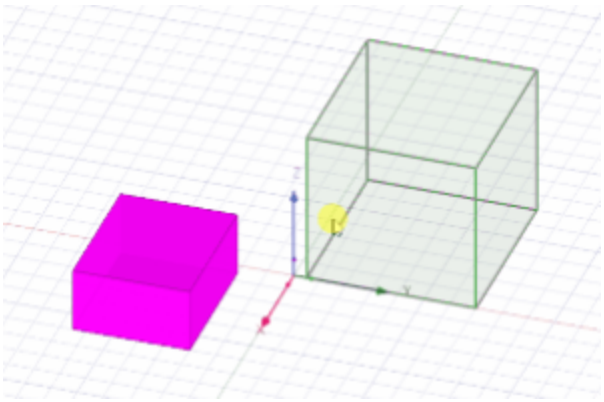
Morph Faces

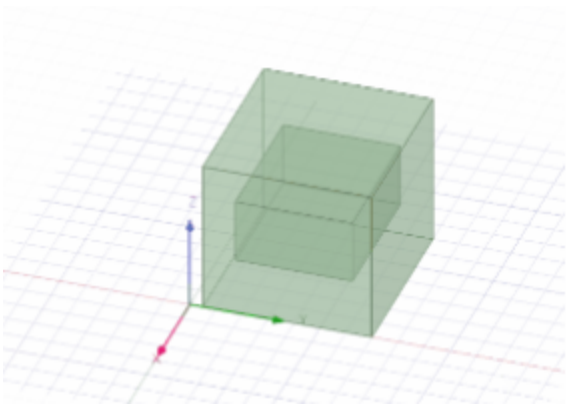




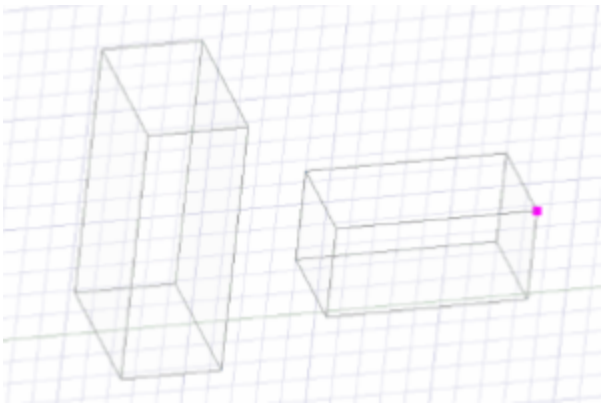
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Align Object Centers

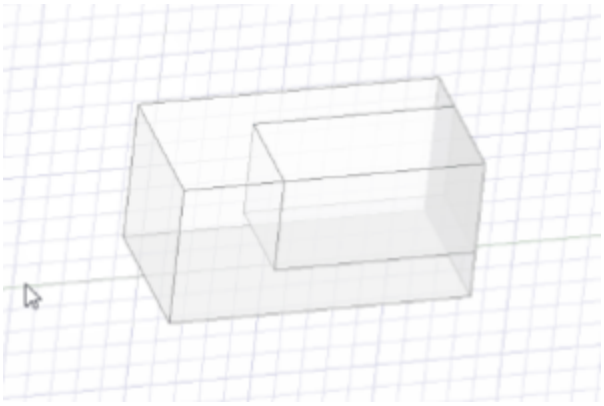




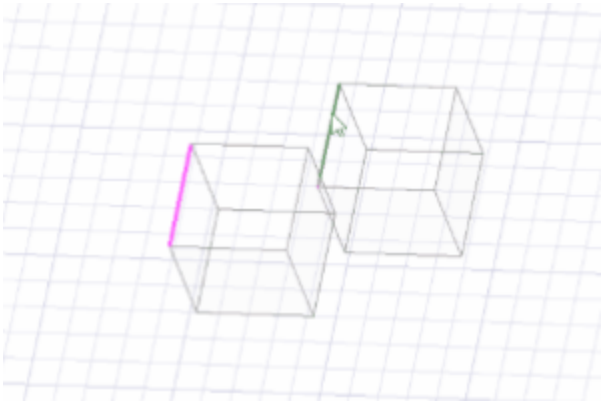
Align and Morph Vertices

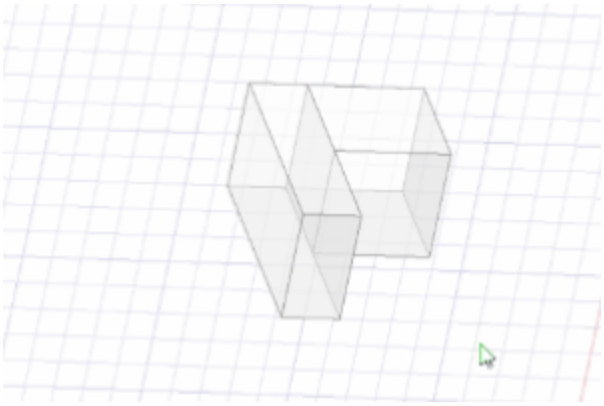


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Align and Morph Edges

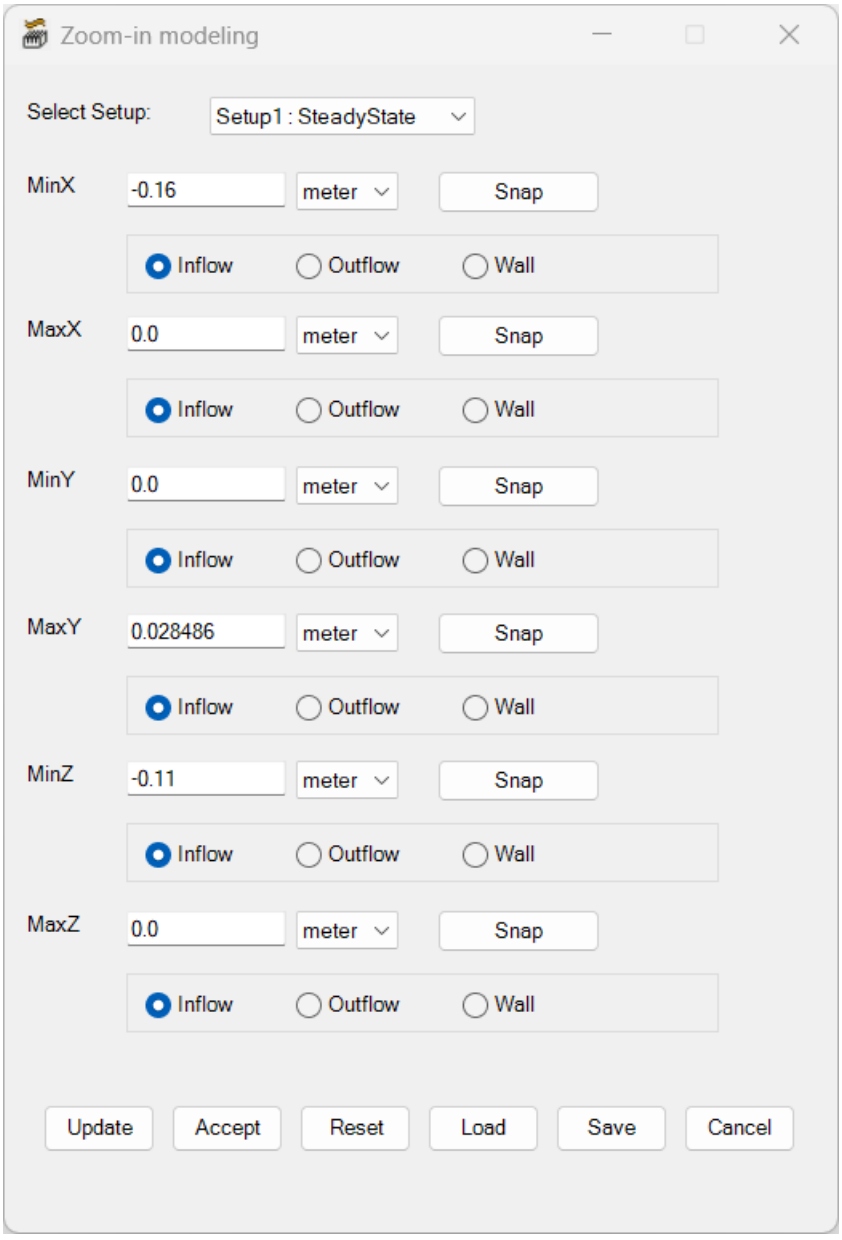




Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zoom-In Modeling

Zoom-in modeling allows you to generate a new design containing a sub-model from a larger model in a source design. Specify a bounding box to surround the geometry you want to include in the sub-model. The new design contains the geometry and its associated boundary conditions and results from the source model. The results are imposed as spatial boundary conditions in the sub-model's design.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Setup	From the **Select Setup** drop-down, select the solve setup to use in generating the results for the sub-model.
Min/Max X, Y, Z	Specify the minimum and maximum X, Y, and Z coordinate locations and units for the bounding box.
Inflow, Outflow, or Wall	For each side of the bounding box, specify whether it should be modeled as **Inflow**, **Outflow**, or a **Wall**.
Snap	Optionally, select a single object, face, edge, or vertex and then click **Snap** to align the bounding box side with the selection.
Update	Click **Update** to update the bounding box geometry based on the values specified the minimum and maximum X, Y, and Z fields.
Accept	Click **Accept** to generate the sub-model in a new design.
Reset	Click **Reset** to restore the default values.
Load	Optionally, click **Load** to open a dialog and select a previously zoom-in modeling file to populate the minimum and maximum X, Y, and Z coordinate location values.
Save	Optionally, click **Save** to save the current values as a zoom-in modeling file for later use.
Cancel	Click **Cancel** to close the dialog box without generating a sub-model.

Icepak Reporting Toolkits

Icepak has the following reporting toolkits.

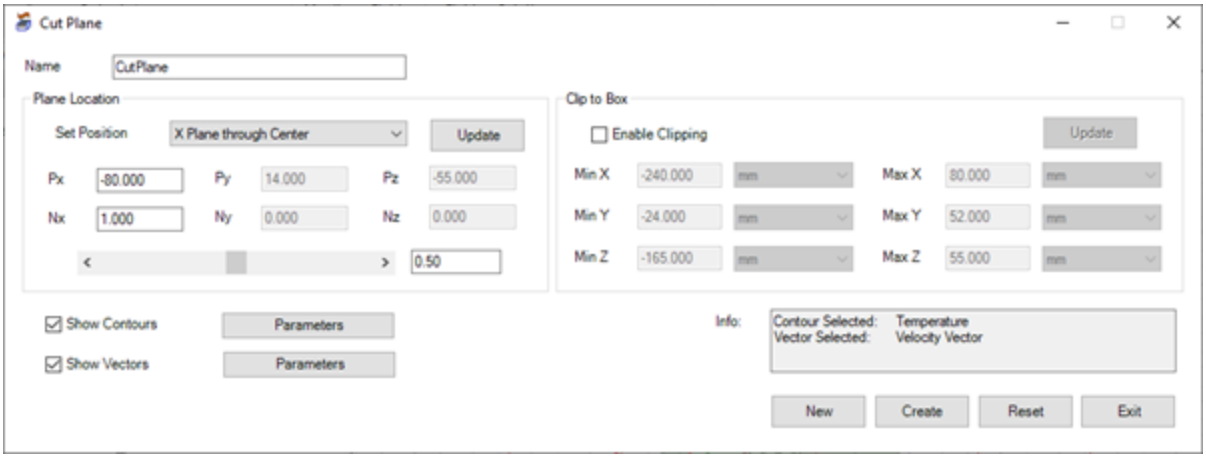
- Cut Plane
- Write Detailed Report
- Min/Max Locations
- Power Budget

Cut Plane

Cut Plane allows you to create a plot thermal or flow data on contours or vectors on a plane. Cut planes allow you to examine areas between objects in the model (in which the fluid alone exists), as well as regions inside objects that contain a fluid or solid material.

> **Note:** The Cut Plane toolkit creates plots that display data in decimal notation.

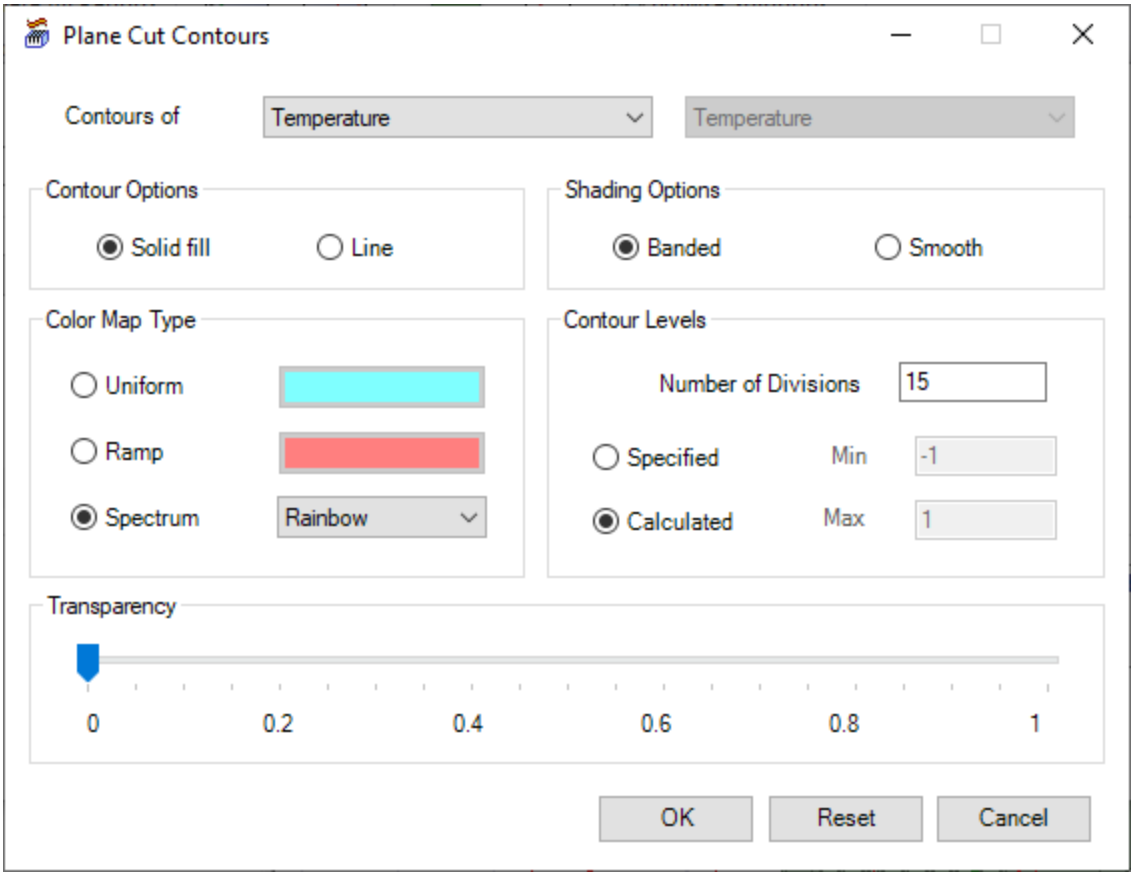
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** appears in the **Project Manager** window, history tree, and **3D Modeler** window.
Plane Location	Under **Plane Location**, specify the location of the plane using one of two methods. Select **X plane through Center**, **Y plane through Center**, or **Z plane through Center** in the **Set position** drop-down list under **Plane location**. The cut plane created is perpendicular to the specified axis. For example, selecting **Y plane through Center** creates a plane in the x-z plane located in the center of the model. You can also specify the location by selecting **Point and Normal**. Then, enter the coordinates of a point on the plane (**Px**, **Py**, **Pz**). Enter a vector defining the direction normal to the plane (**Nx**, **Ny**, **Nz**). For example, entering (1, 0, 0) for the vector will define a normal pointing in the x direction. The positioning slider bar is only available when creating a cut plane using X, Y, and Z plane through center.
Clip to Box	If needed, under **Clip to Box**, select **Enable Clipping** to create a cut plane that spans a specified sub-range. Enter values in the **Min** and **Max** fields and click **Update**.
Show Contours	Select **Show Contours** and click **Parameters** to display the **Plane Cut Contours** dialog box. See below for more information.
Show Vectors	Select **Show Vectors** and click **Parameters** to display the **Plane Cut Vectors** dialog box. See below for more information.
Info	The **Info** dialog displays which quantities are plotted on the contours and/or vectors.
New, **Create**, **Reset**, **Exit**	Click **New** button to create a cut plane (if not already created) based on selected options and then show the preview of new cut plane with all values reset to default values. Use the **New** button control to adding multiple contours without closing the toolkit. Click **Create** to create the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

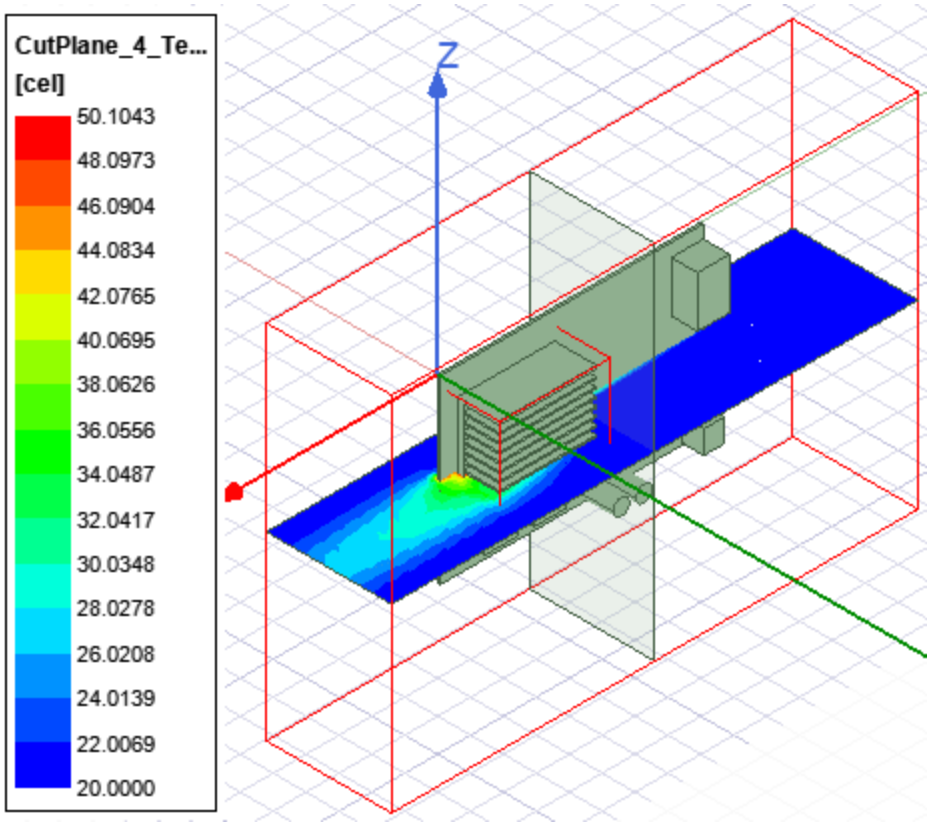
	cut plane based on the current selections. Click **Reset** to restore default values. Click **Exit** to close the **Cut Plane** dialog box.



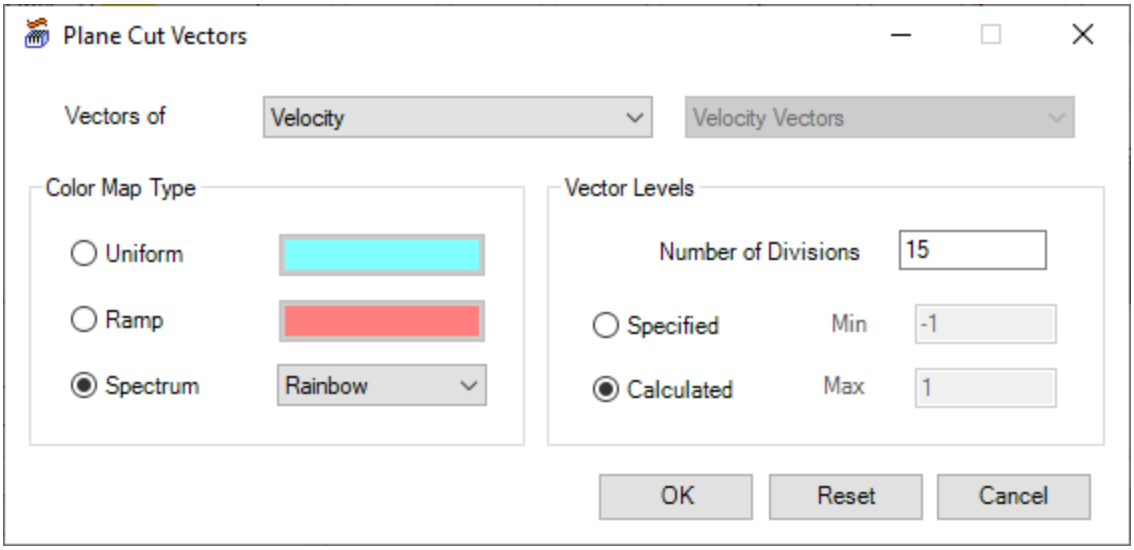
Contours of	In the **Contours of** drop-down list, select the quantity of data to plot on the cut plane and its components (if any).
Contour Options	Under **Contour Options**, specify **Solid fill** if you want a solid contour plot or **Line** if you want a line contour plot. For solid fill contour plots, specify the **Shading Options**.
Shading Options	Under **Shading Options**, select **Banded** to have the colors change abruptly at the contour boundaries. Select **Smooth** to vary the colors smoothly in continuous gradations from one boundary of the contour to the other.
Color Map Type	Under **Color Map Type**, select **Uniform** to display one color on the cut plane. Select **Ramp** to display a gradient of the selected color. Select **Spectrum** and a color option to display a range of colors.
Contour Levels	Under **Contour Levels**, enter a value for **Number of Divisions** to

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	specify the density of the lines. Select **Specified** to specify the minimum and maximum values in the displayed value range, or select **Calculated** for Icepak to automatically select the values based on the calculated minimum and maximum.
Transparency	Under **Transparency**, use the slider bar to adjust the transparent or opaque the cut plane contours appear.
OK, **Reset**, **Cancel**	Click **OK** to save the settings and close the **Cut Plane Contours** dialog box. Click **Reset** to restore default values. Click **Cancel** to close with dialog box without saving.

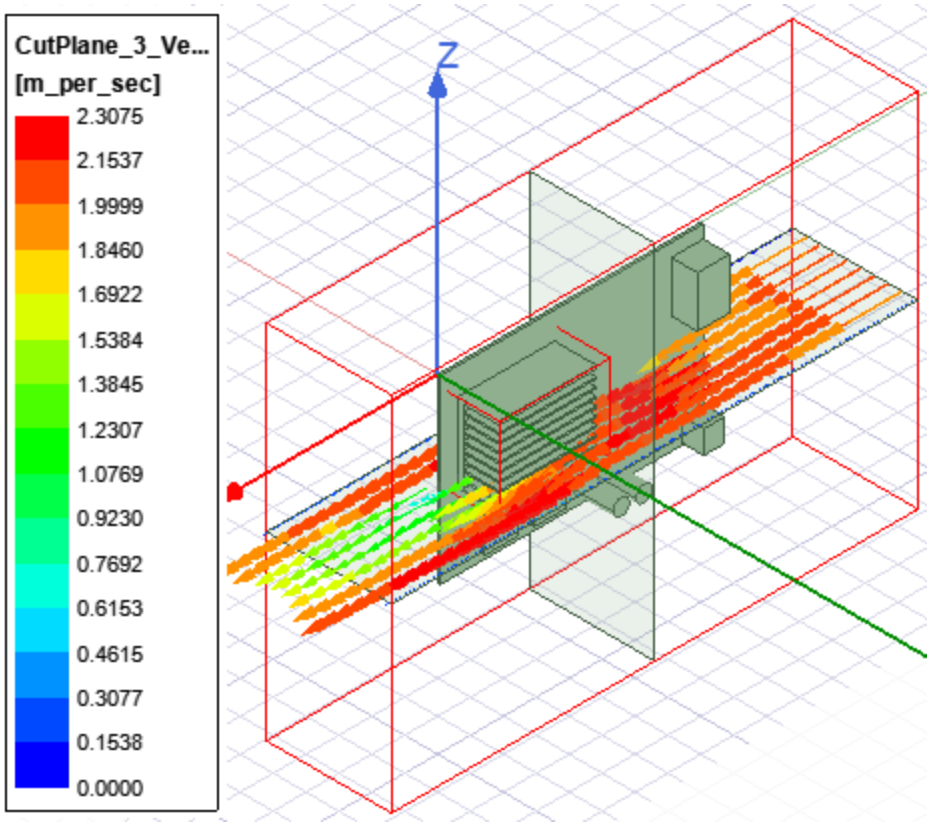


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Vectors of	In the **Vectors of** drop-down list, select the quantity of data to plot on the cut plane and its components (if any).
Color Map Type	Under **Color Map Type**, select **Uniform** to display one color on the cut plane. Select **Ramp** to display a gradient of the selected color. Select **Spectrum** and a color option to display a range of colors.
Vector Levels	Under **Vector Levels**, enter a value for **Number of Divisions** to specify the density of the lines. Select **Specified** to specify the minimum and maximum values in the displayed value range, or select **Calculated** for Icepak to automatically select the values based on the calculated minimum and maximum.
OK, **Reset**, **Cancel**	Click **OK** to save the settings and close the **Cut Plane Vectors** dialog box. Click **Reset** to restore default values. Click **Cancel** to close with dialog box without saving.

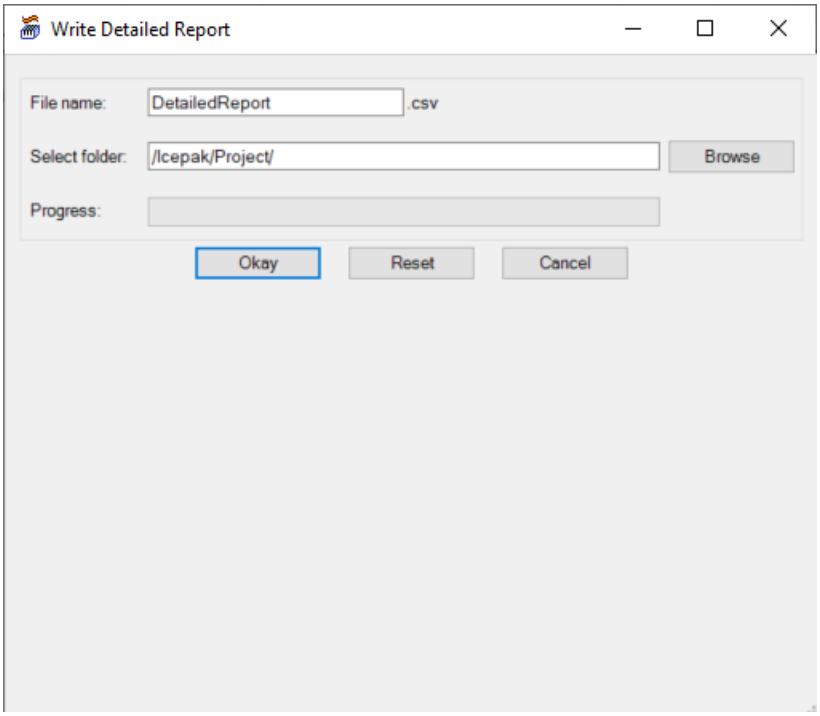
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Write Detailed Report

Write Detailed Report allows you to create a report featuring thermal data related to the components in your model. The report is saved as a comma separated values file (.csv), which you can open a spreadsheet application.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The report contains the following information about each component:

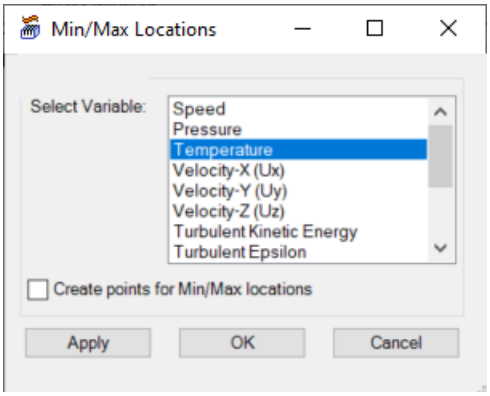
- Power
- Max Temperature
- Material
- Density
- Specific Heat
- Conductivity Type
- Conductivity

Object	Power [W]	Max Temperature [C]	Material	Density [Kg/m^3]	Specific Heat [J/(Kg-C)]	Conductivity type	Conductivity [W/(m-C)]
ALPHA_MAIN_PCB		74.6123	PCB_Material	1	1	Anisotropic	
CAPACITOR		31.69	Al-Extruded	2800	900	Isotropic	205
CAPACITOR_1		24.9127	Al-Extruded	2800	900	Isotropic	205
CPU	25	51.3169	Ceramic_material	3970	910	Isotropic	15
HEAT_SINK		50.1506	Al-Extruded	2800	900	Isotropic	205
KB		21.8064	Al-Extruded	2800	900	Isotropic	205
MEMORY1	5	75.1374	Ceramic_material	3970	910	Isotropic	15
MEMORY1_1	5	68.523	Ceramic_material	3970	910	Isotropic	15
Region		75.1374	air	1.1614	1005	Isotropic	0.0261
SERIAL_PORT		21.4008	Al-Extruded	2800	900	Isotropic	205

Min/Max Locations

Min/Max Locations allows you to identify the minimum and maximum locations of a selected quantity, such as temperature. You can also create monitor points at the identified locations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



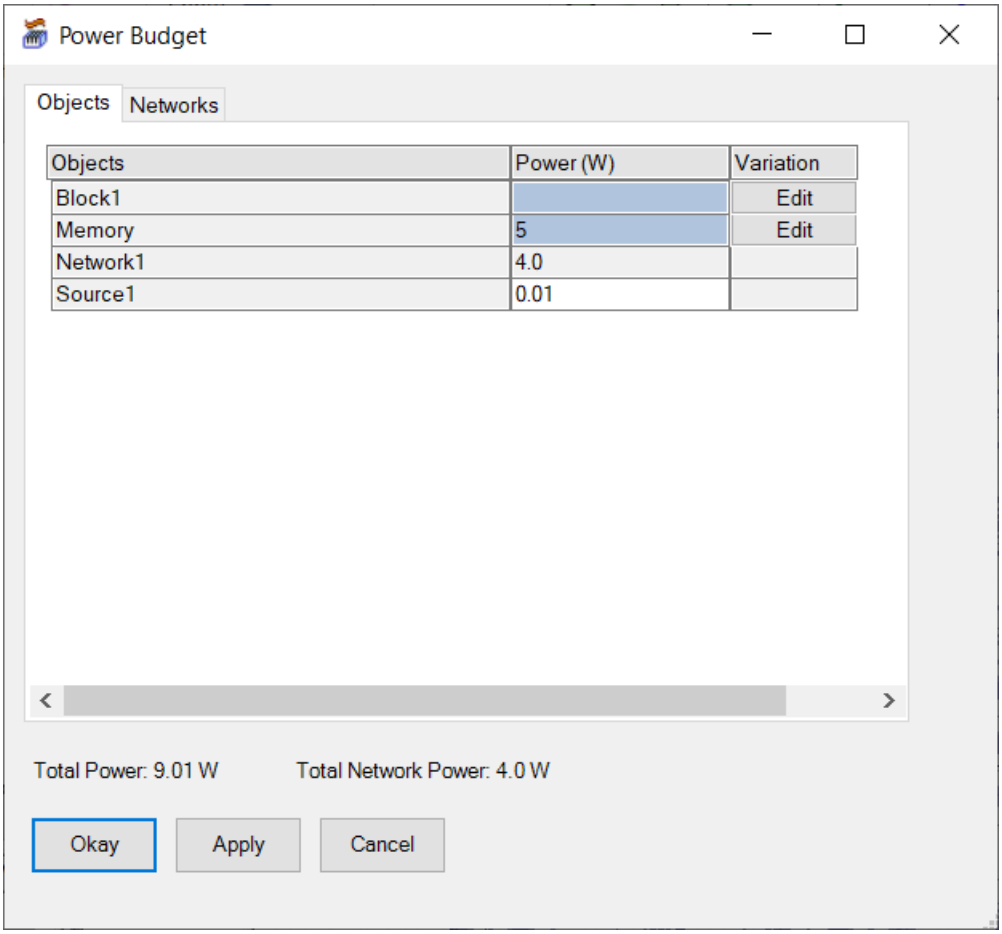
Select Variable	In the **Select Variable** field, select the quantity for which you want to identify the minimum and maximum locations.
Create points for Min/Max locations	Enable **Create points for Min/Max locations** to create monitor points at the identified minimum and maximum locations.
Apply	Click **Apply** to locate the min and max locations for the selected quantity and, if enabled, create monitor points at the locations.
OK	Click **OK** to locate the min and max locations for the selected quantity and, if enabled, create monitor points at the locations. Clicking **OK** closes the Min/Max Locations dialog box.
Cancel	Click **Cancel** to close the **Min/Max Locations** dialog box.

Power Budget

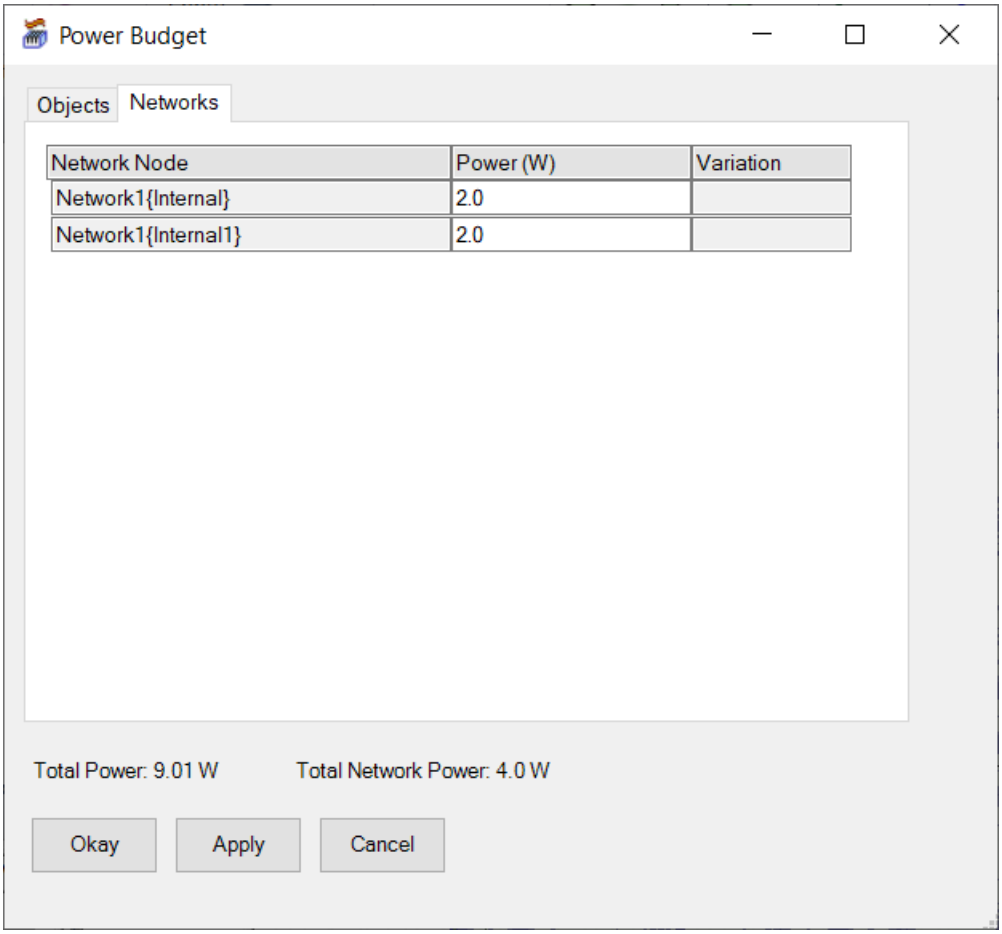
Power Budget displays each boundary condition with a power assignment as well as the total cumulative power and total network power. It also allows you to modify the values if necessary.

> **Note:** If a source boundary condition has a **Surface Flux** thermal condition, the flux is converted to a power value as illustrated in the following image.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



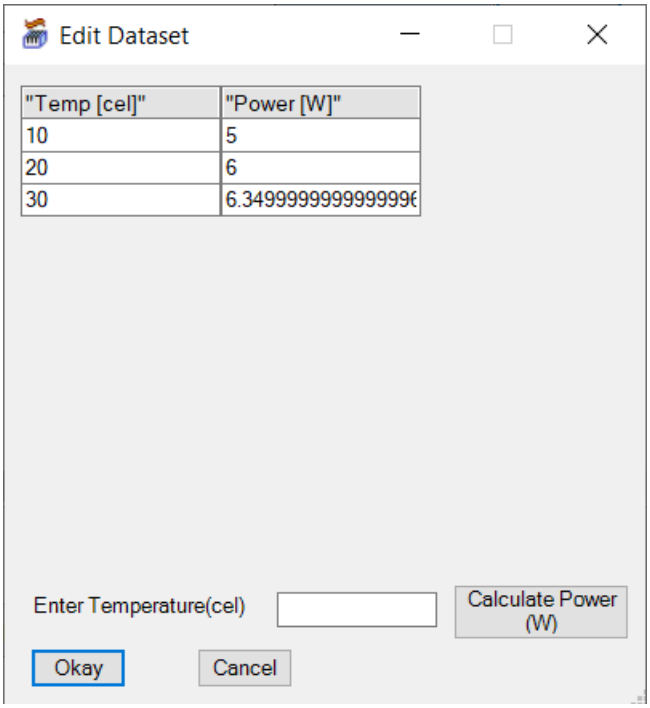
Objects/Networks Tabs	The **Objects** tabs shows all boundary conditions with a power assignment, including the sum of all network power assignments. The **Networks** tab displays only network boundary conditions with power assignments.
Total Power/ Total Network Power	**Total Power** displays the sum of power values assigned to all boundary conditions. **Total Network Power** displays the sum of power values assigned to only network boundary conditions.
Okay	Click **Okay** to update power values if you made changes and close the **Power Budget** dialog box.
Apply	Click **Apply** to update power values.
Cancel	Click **Cancel** to close the **Power Budget** dialog box. Any changes that were not applied are not saved.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

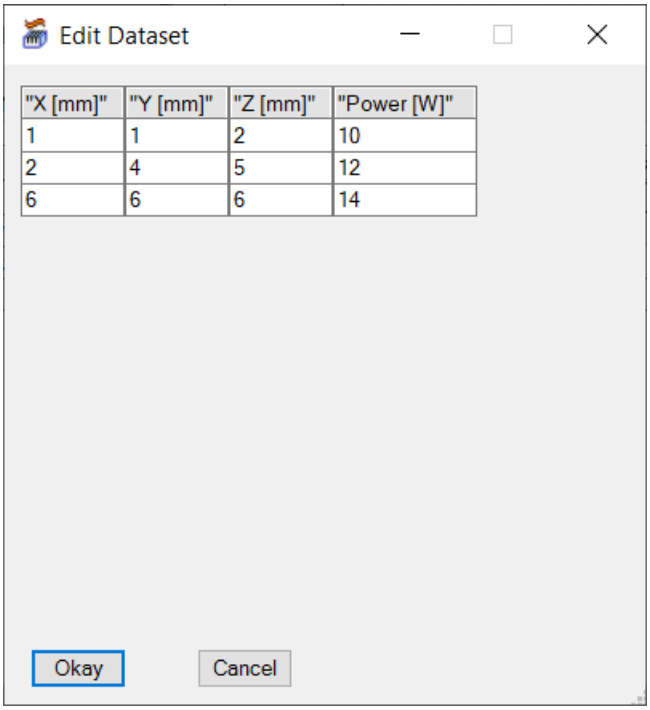
Variations

If your design contains a boundary condition with temperature-dependent dataset or spatial dataset assigned for power, click **Edit** in the **Variation** column to view and edit the dataset.

> **Note:** To calculate a new value for a temperature-dependent dataset, enter the temperature and click **Calculate Power (W)**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Assigning Materials and Surface Materials

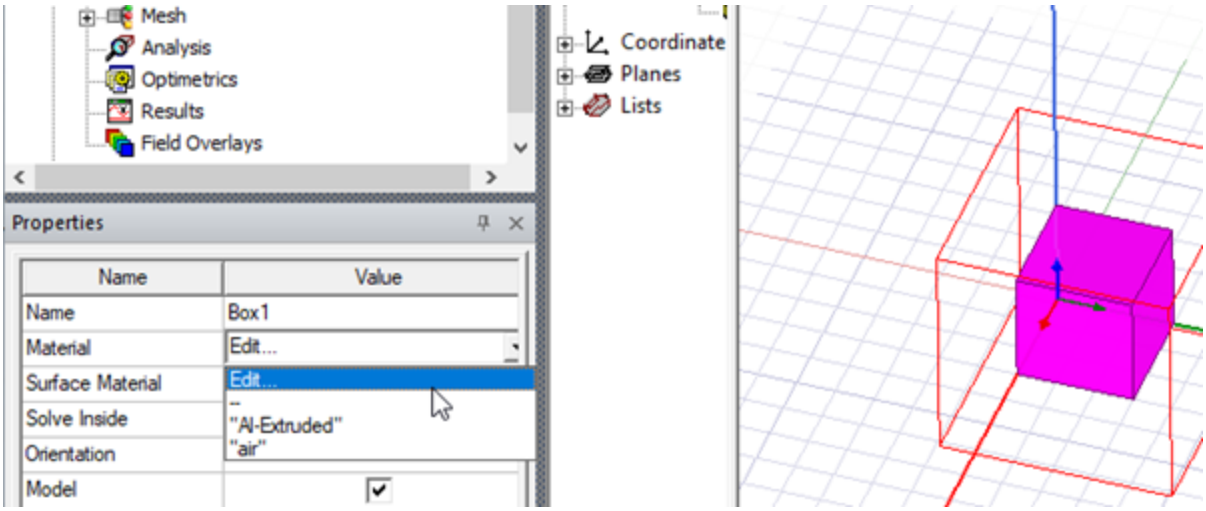
You can add, remove, and edit materials in two main ways:

- Using the **Tools** > **Edit Libraries** > **Materials** or **Surface 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.

To assign a material to an object, follow this general procedure:

1. Select the object or object face to which you want to assign a material or surface material.
2. Click **Modeler** > **Assign Material** or **Assign Surface Material** or select the Material field in the **Properties** window for the selected object, and select **Edit...** from the drop-down menu:

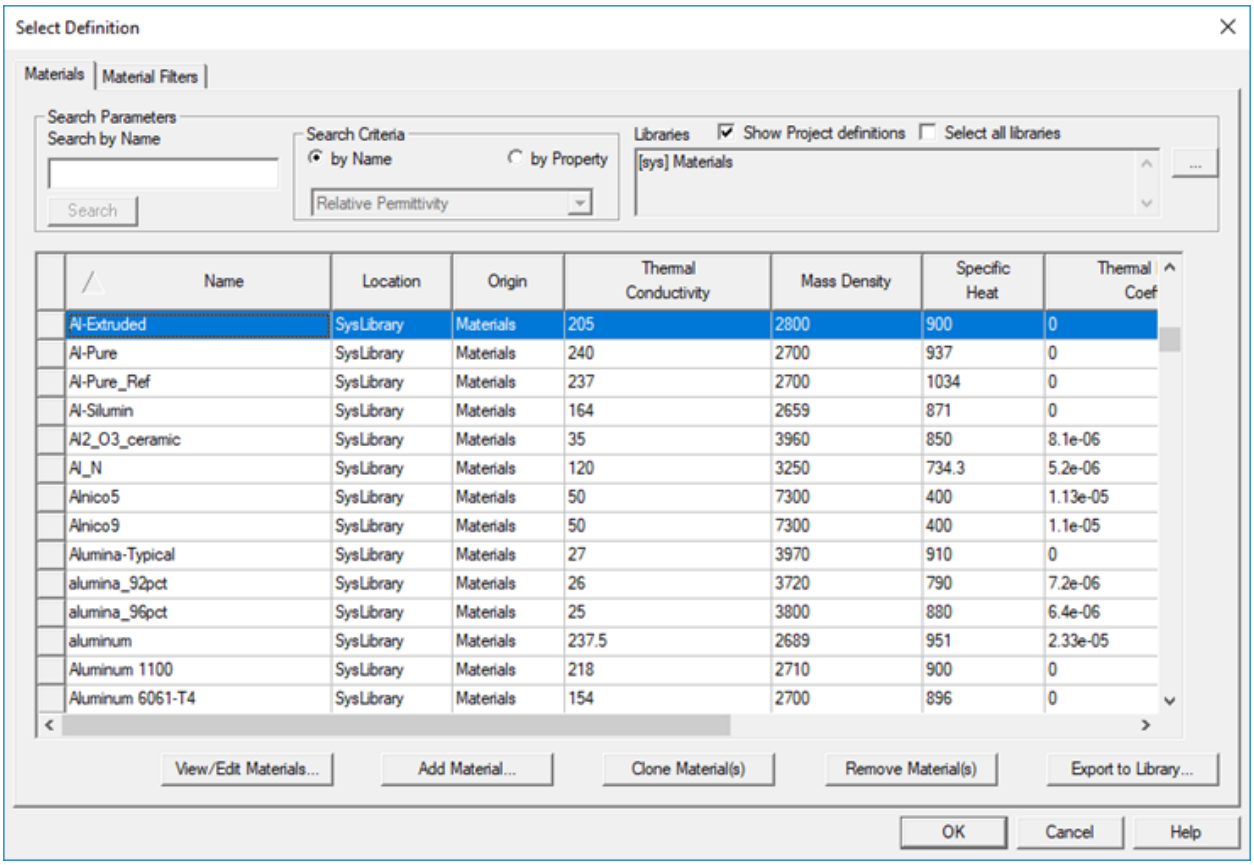


The **Select Definition** window appears. The current material is highlighted, with the Name, Location, Origin library, and parameter values shown.

> **Note:** You can resize the dialog box to show more of the materials and/or material properties without having to use the scroll bars.

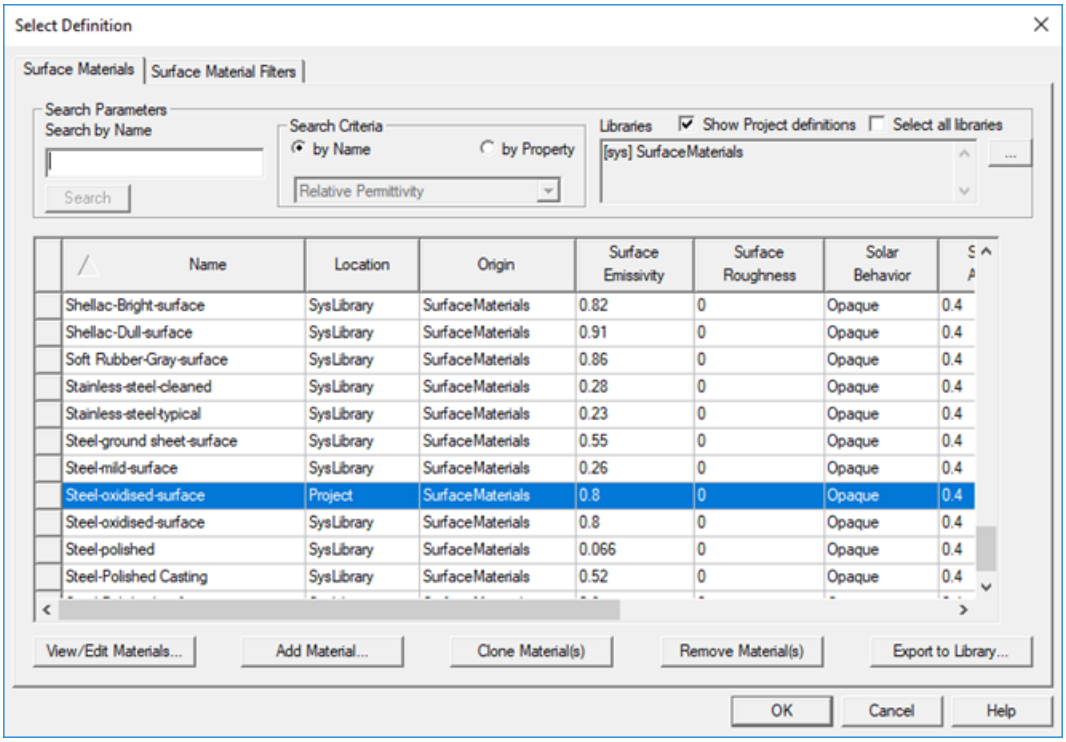
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Definition Dialog Box - Materials

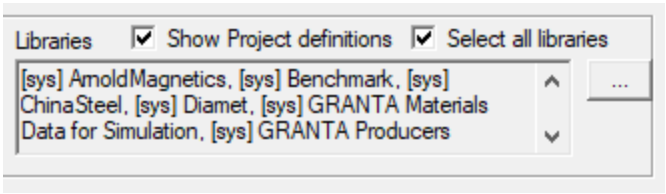


Select Definition Dialog Box - Surface Materials

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

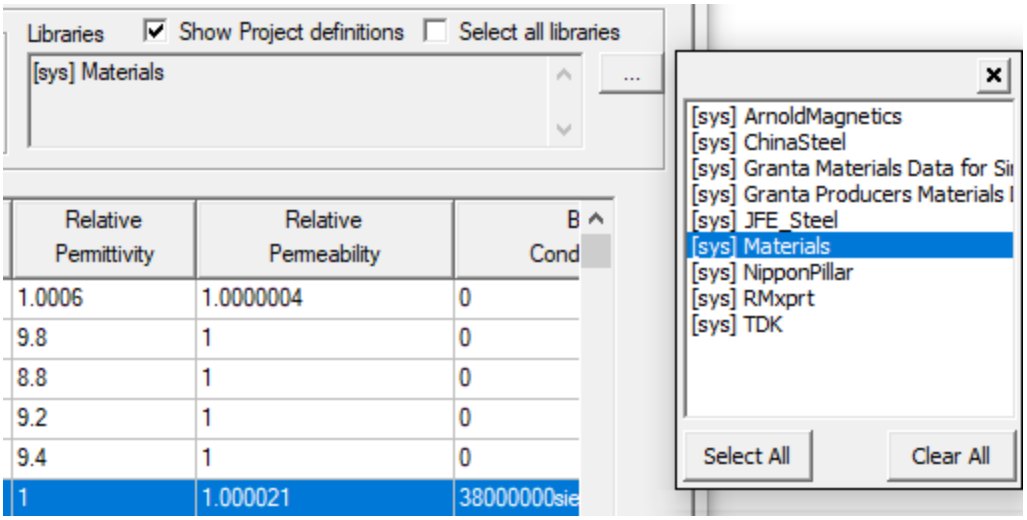


When the **Select all libraries** check box is selected, the window lists all of the materials in Ansys Electronics Desktop's global material library as well as the project's local material library

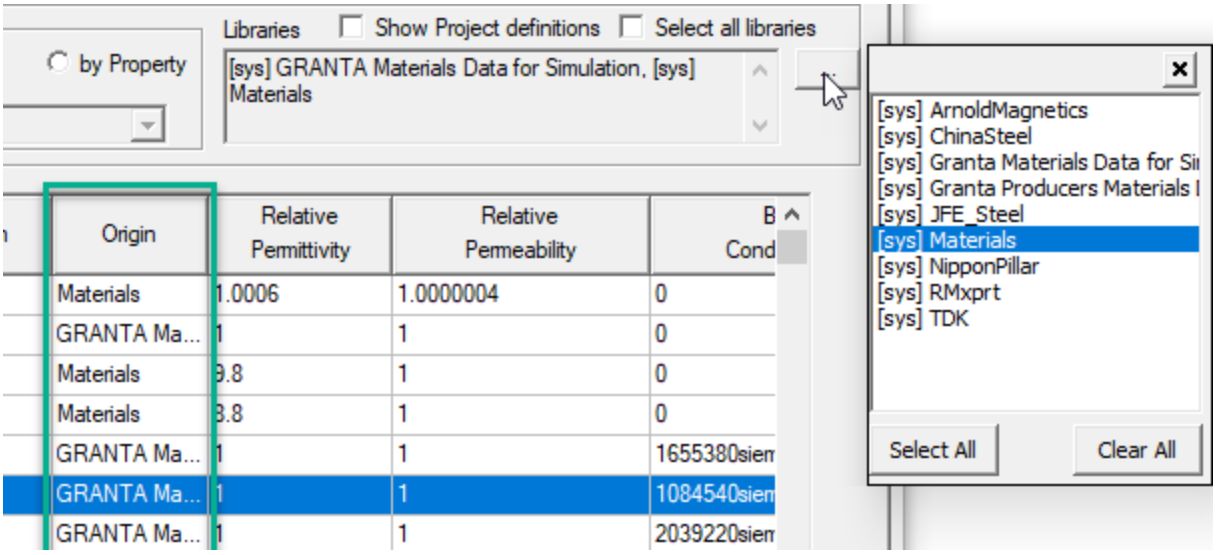


If you click the ellipsis button **[...]**, you see a list of all available libraries. Any selected libraries are highlighted. Shift-Click allows you select a range. Ctrl-Click allows you to select any libraries.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Origin** column shows the originating library for each material, whether the **sys** library, or one of the additional libraries listed in the Libraries pane.



For further information on the materials and their intended uses, you can refer to the published information on materials from those libraries. For example, the Schott materials are described in detail in the Wily Series in Materials for Electronic & Optoelectronic Applications, *Microwave Dielectric Materials and Applications*, edited by M. T. Sebastian, Rick Ubic, and Heli Jantunen, volumes 1 and 2.

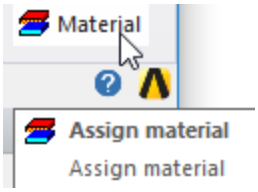
GRANTA Materials Data for Simulation (MDS) is an optional, licensed feature, including more than 700 generic materials and over 2100 producer-specific magnetic materials and PCB materials. When you use or view the GRANTA licensing libraries, the GRANTA

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

license is checked out and held for 30 minutes. Viewing the contents of a GRANTA library in the Materials dialog box or modifying an object to use a GRANTA material causes a license check out and hold. If the license is already checked out, the hold time is reset to 30 minutes.

You can also open the **Select Definition** window in one of the following ways:

- In the **Properties** dialog box for the object, click the material name under the **Attributes** tab. A drop-down menu shows an **Edit...** button that opens the **Select Definition** window.
- With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.

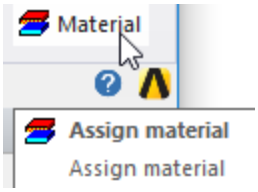


The menu also lists materials included in the current project. Selecting one of these materials provides another way to assign materials to an object.

- Right-click **Model** in the project tree, and then click **Assign Material** on the shortcut menu.
- Right-click the object in the history tree, and then click **Assign Material** on the shortcut menu.

Select Definition Dialog Box - Materials

3. With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.



4. Select a material from the list.

> **Note:**
>
> You can search the listed materials by name or property value.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the material you want to assign is not listed, add a new material to the global or local material library, and then select it.

5. Click **OK**.

   The material you chose is assigned to the object.

> **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.

You can edit the color and transparency values for materials in the View/Edit Material window.

Materials and Surface Materials

Ansys Electronics Desktop provides two different system material libraries: Materials and Surface Materials. Materials in the Materials system library are bulk materials for assignment to 3D geometry (e.g., 3D geometry selected in object selection mode). Bulk materials are defined by volume-related properties such as thermal conductivity. Surface materials in the Surface Materials system library are for assignment to the surfaces of 2D or 3D geometry (for example, the face of 3D geometry selected in face selection mode). Surface materials properties are defined by surface-related properties such as surface roughness and surface emissivity.

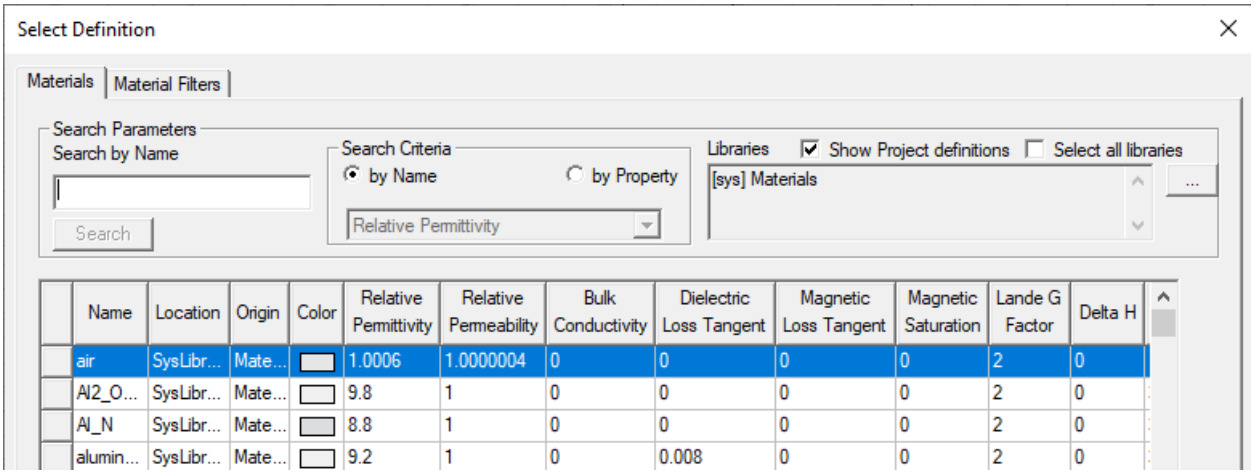
The Materials system library can be viewed and edited in the **Edit Libraries** dialog accessed by clicking **Tools**>**Edit Libraries**>**Materials**.

The Surface Materials system library can be viewed and edited in the **Edit Libraries** dialog accessed by clicking **Tools**>**Edit Libraries**>**Surface Materials**.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

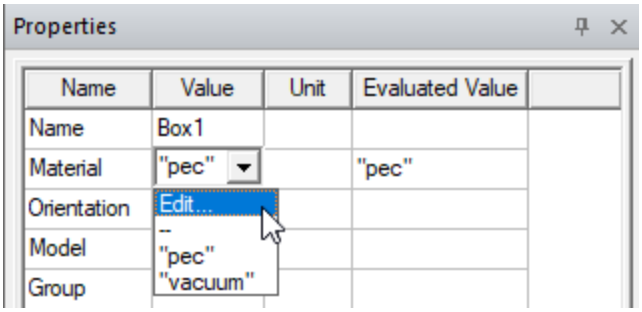


To search for a material **by Name**:

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 **Modeler** > **Assign Surface Material**.
   - Click the **Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.

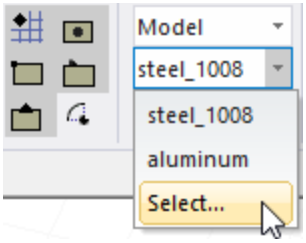


   - Click the **Surface Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.
   - On the **Draw** ribbon tab, click **Assign surface material**.

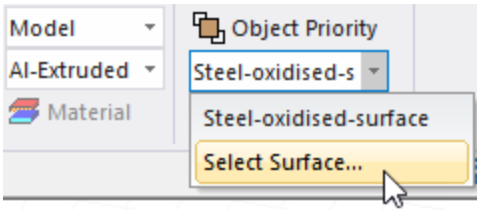
   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:

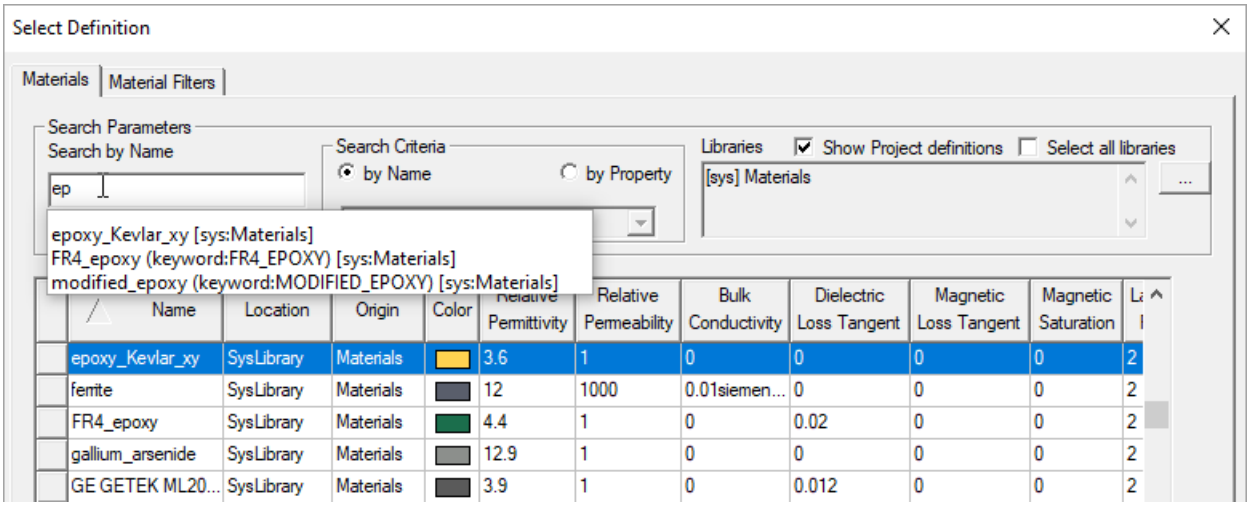
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- On the **Draw** ribbon tab, choose **Select** from the **Default surface material** drop-down menu:



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.



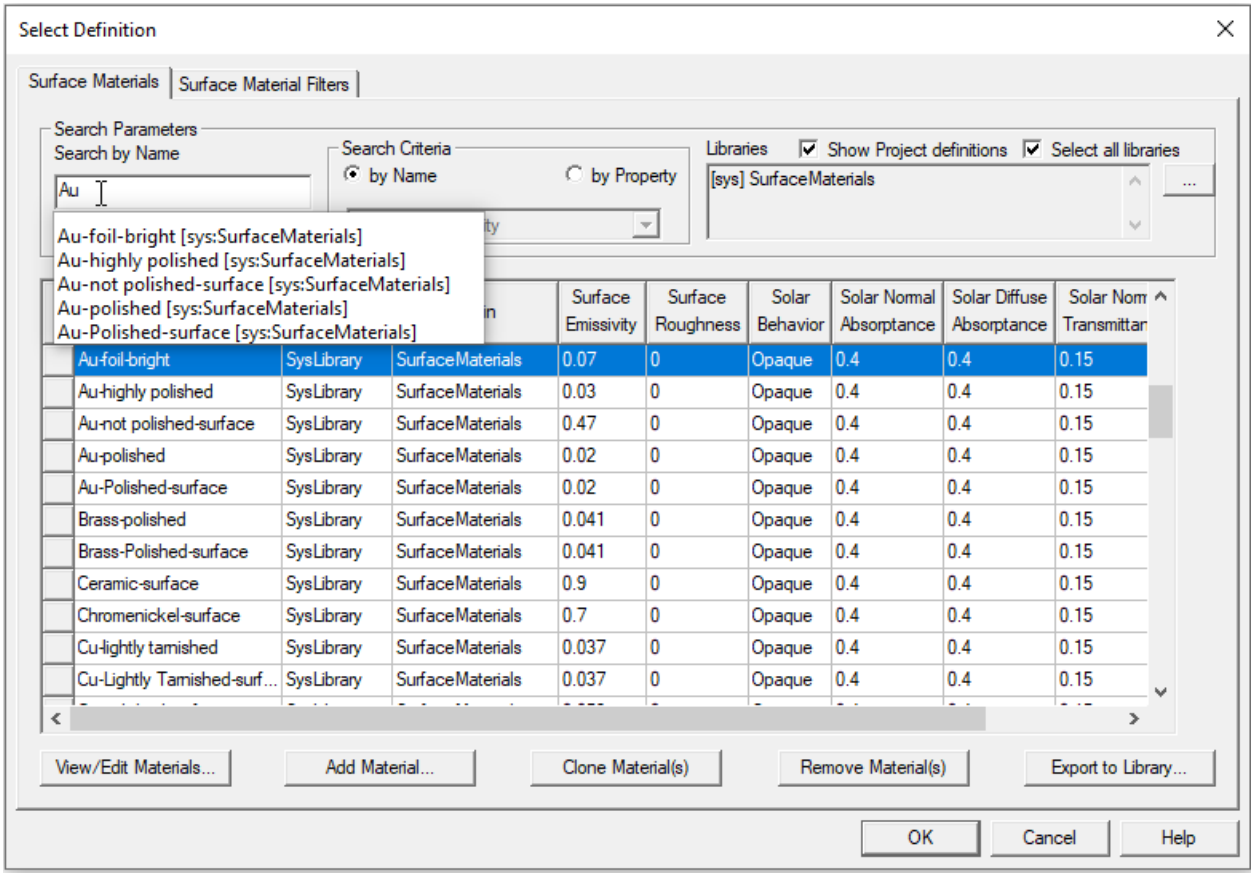
A drop-down menu under the Search Parameters field will show the a list of the materials from selected libraries matching any part of the current text, and, if the Granta Materials library is selected, matching keywords in the material definitions. The first row containing the material name most similar 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:

Ansys Electromagnetics Suite 2026 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 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 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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Searching by Material Property

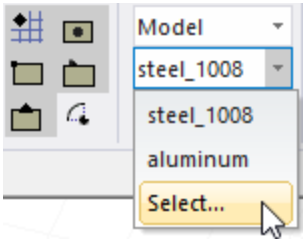
1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

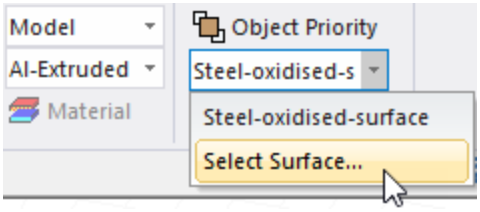
   - Click **Modeler** > **Assign Material**.
   - Click **Modeler** > **Assign Surface Material**.
   - Click the **Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - Click the **Surface Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.
   - On the **Draw** ribbon tab, click **Assign surface 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:



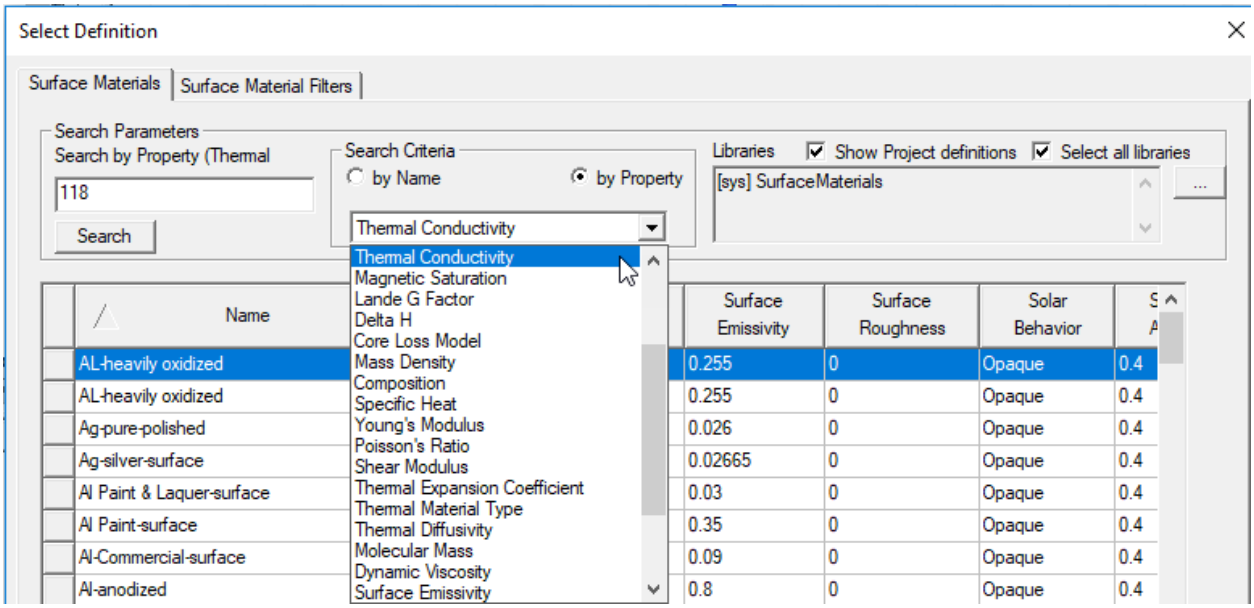
   - On the **Draw** ribbon tab, choose **Select** from the **Default surface material** drop-down menu:



2. In the **Search Criteria** section, select **by Property**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select a material property from the drop-down menu:



> **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**.

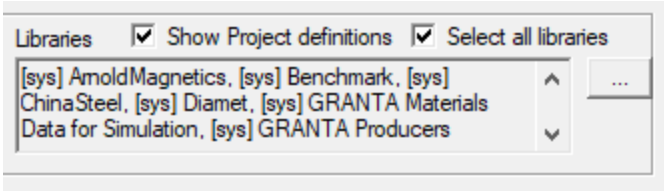
   The materials are sorted according to the property you selected. Additionally, the material with the property value closest to the one you typed, but without exceeding it, is selected.

   If the selected material is not the one you are searching for, do one of the following:

   - 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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** window, 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**or **Surface Materials**. In the project tree, you can also right-click **Materials**, and select **Edit Library**.

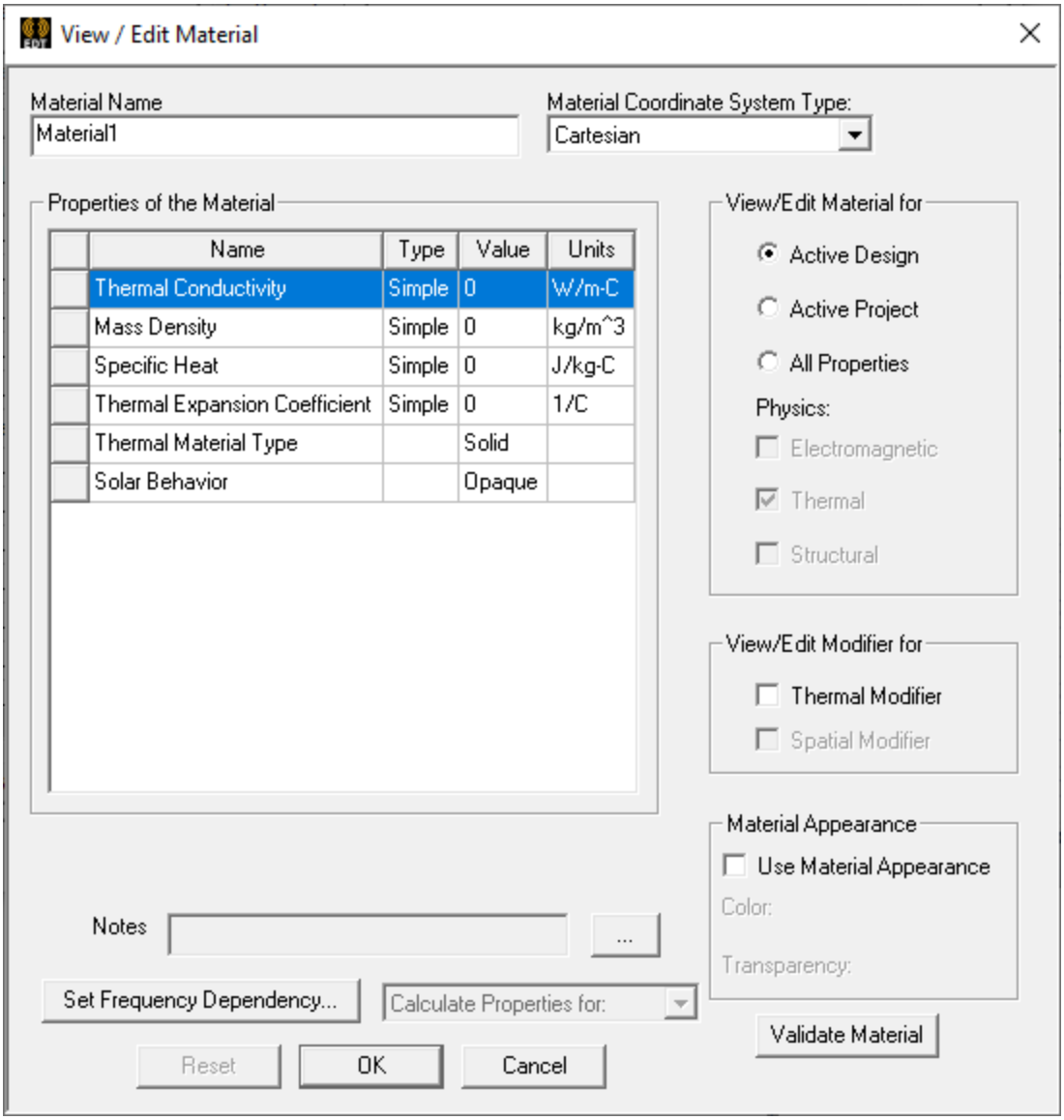
  The **Edit Libraries** window appears.

To add a new material:

1. From either the **Select Definition** window or the **Edit Libraries** window, click **Add Material**.

   The **View/Edit Material** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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** or **Surface Material Name** text box, or accept the default.
3. Select a **Material Coordinate System Type** from the drop-down menu:
   - **Cartesian** (default)
   - **Cylindrical**
   - **Spherical**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

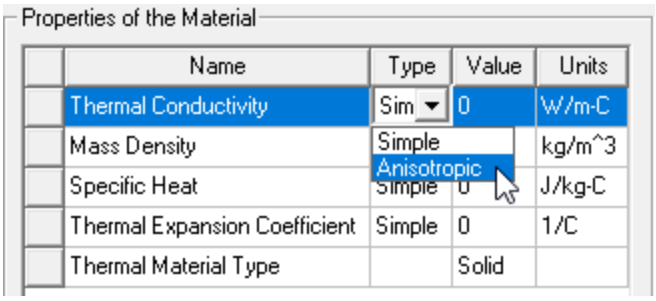
> **Note:**
>
> Spherical coordinate systems are not supported for the Icepak design type.

4. 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).

   > **Note:**
   >
   > If a material is edited in a design type for which the **Physics** type has not been set (e.g., an Icepak design but Thermal physics type was not set), the **Physics** type will be automatically set in the material.
   >
   > **Physics** classification options are unavailable for surface materials.

   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.

5. Enter a **Material Name** or **Surface Material Name**, or accept the default.
6. 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. For thermal materials, select **Solid** or **Fluid** for the **Thermal Material Type**.



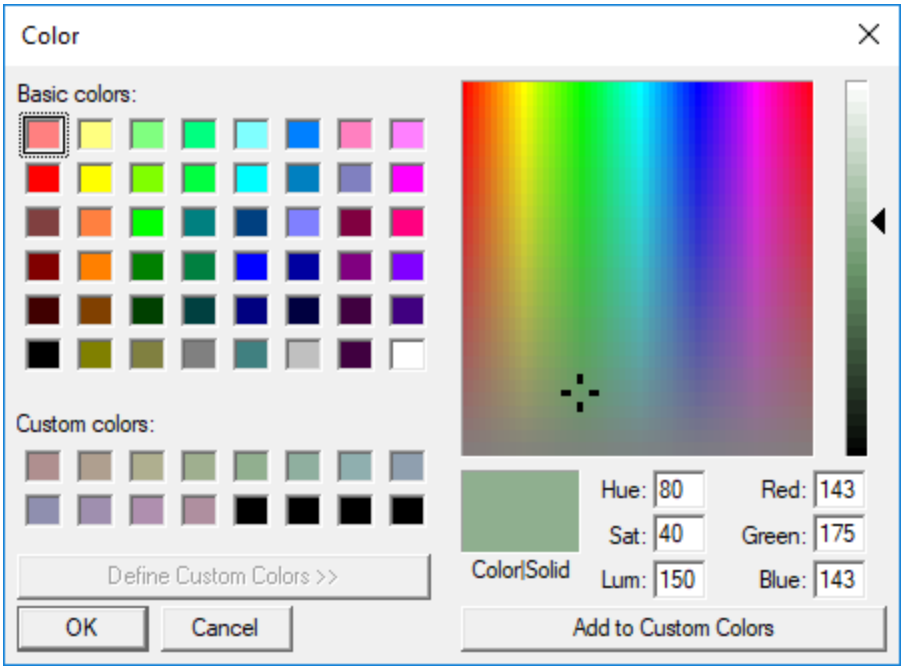
7. Enter a **Value** for the following material properties:
   - Thermal Conductivity
   - Mass Density
   - Specific Heat
   - Thermal Expansion Coefficient (fluid materials only)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Thermal Diffusivity (fluid materials only)
- Molecular Mass (fluid materials only)
- Dynamic Viscosity (fluid materials only)
- Solar Behavior (fluid materials only)
- Absorption Coefficient (transparent fluid materials only)
- Scattering Coefficient (transparent fluid materials only)
- Refractive Index (transparent fluid materials only)

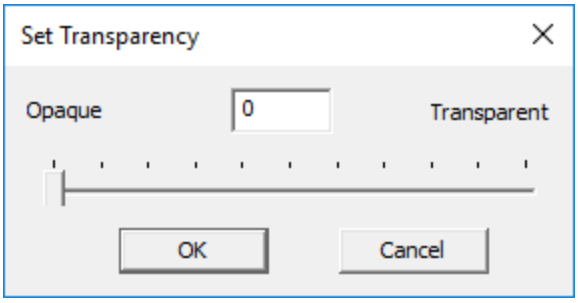
> **Note:**
>
> You may enter a variable name or mathematical expression in the **Value** box.

8. 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.
9. To modify the units for a material property, double-click the **Units** box and select a new unit system.
10. 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:

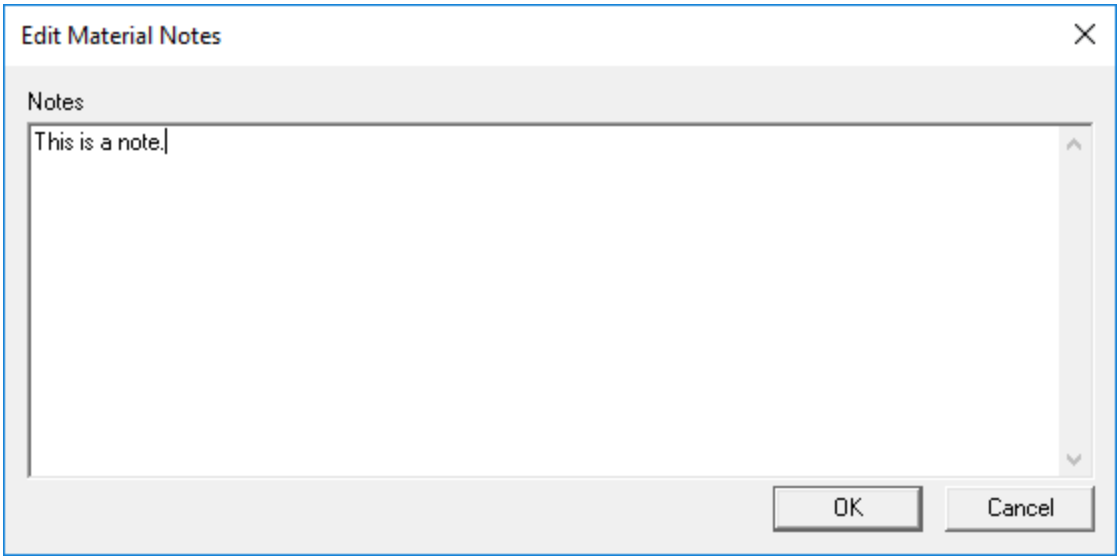


11. Clicking the Transparency box opens a Transparency dialog box with a text field and slider bar for selection.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



12. Click **OK**.
13. To modify the units for a material property, double-click the **Units** box and select a new unit system.
14. Click **OK**.
15. 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.



16. Click **OK** to add the Notes.
17. Click **OK** on the **View/Edit Materials** window to add the new material to the material library.

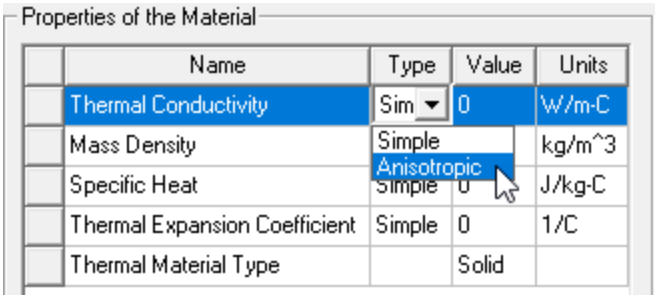
Assigning Material Property Types

Material properties can be assigned using the **View/Edit Material** window, using one of the following material property types, based on the applicability to the material property:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction. This models only the diagonal.

Select a material property type for each property from the **Type** drop-down menu. Some properties only use the Simple type

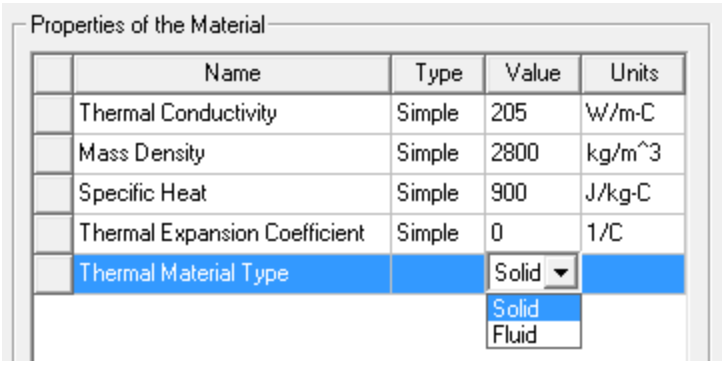


If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic thermal conductivity and thermal expansion coefficient. Each diagonal represents a tensor of your model along an axis.

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.

Thermal Properties for Solid Materials

The thermal properties a material has is determined by the **Thermal Material Type** selection. When the material is a **Solid**, the following properties must be set:



- **Thermal Conductivity** is the thermal conductivity of the material.
- **Mass Density** is the density of the material.
- **Specific Heat** is the specific heat capacity of the material.

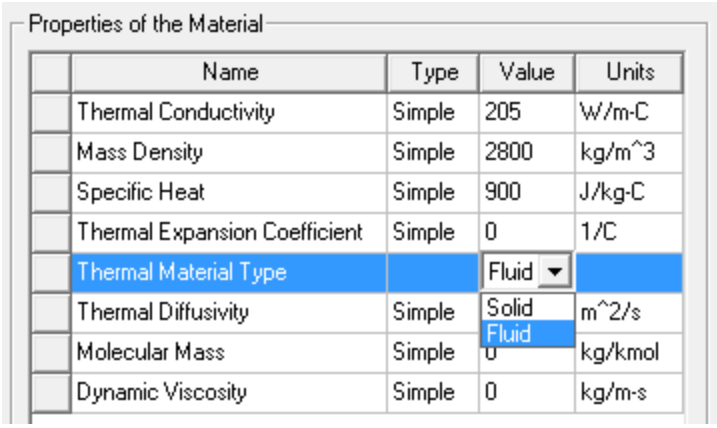
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For solid materials, Thermal Conductivity and Specific Heat can be assigned a Thermal Modifier to define a temperature dependency. See Specifying Thermal Modifiers for more information.

Thermal Properties for Fluid Materials

Note: For fluid materials, Thermal Conductivity, Specific Heat, Thermal Diffusivity, and Dynamic Viscosity can be assigned a Thermal Modifier to define a temperature dependency. See Specifying Thermal Modifiers for more information.

The thermal properties a material has is determined by the **Thermal Material Type** selection. When the material is a **Fluid**, the following properties must be set:



- **Thermal Conductivity** is the thermal conductivity of the material.
- **Mass Density** is the density of the material.
- **Specific Heat** is the specific heat capacity of the material.
- **Thermal Expansion Coefficient** is the volumetric expansion coefficient of the material.
- **Thermal Diffusivity** is the diffusivity of the fluid.
- **Molecular Mass** is the molecular weight of the fluid.
- **Dynamic Viscosity** is dynamic viscosity of the fluid.

Note: If you use the ideal gas law and you have created a new fluid material, make sure that you specify the correct molecular weight for the new material.

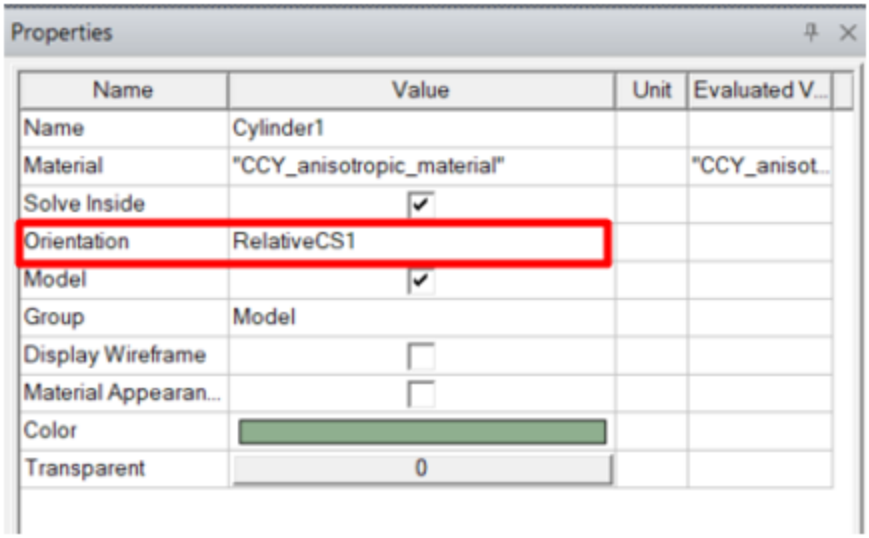
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a thermal modifier is applied to mass density with ideal gas law enabled, the thermal modifier is used in the simulation.

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.



Important:

Anisotropic materials are supported in 2D models but are not available for Q3D.

Assigning Anisotropic Tensors

To assign anisotropic tensors to thermal conductivity and thermal expansion coefficient:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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 thermal conductivity and thermal expansion coefficient. along each axis of the material's tensor.
   - For **Thermal Expansion Coefficient** – Enter the thermal expansion coefficient along the direction of each tensor axis. These tensors can be based on a Cartesian, cylindrical, or spherical *Material Coordinate System Type*.
4. Click **OK** to save the values and return to the **Select Definition** window.

Defining Anisotropic Thermal Conductivity

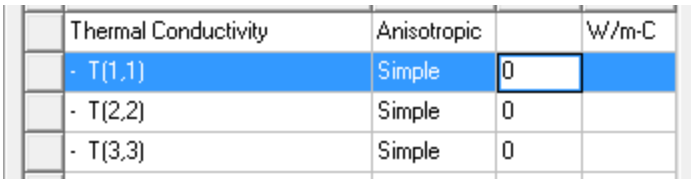
If the thermal conductivity is anisotropic, either the Cartesian or the Cylindrical coordinate systems can be selected to define the characteristics of its anisotropy tensor. For Cartesian-orthotropic materials you must define three diagonals for the conductivity. Each diagonal represents a tensor of your model along a Cartesian coordinate axis. Cylindrical-orthotropic materials are defined by the axial, circumferential and radial values for the conductivity.

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.

1. In the **Thermal Conductivity** row in the **View/Edit Material** window, select **Anisotropic**.

For **Cartesian**:

- Three rows named **T(1,1)**, **T(2,2)**, and **T(3,3)** are added below the **Thermal Conductivity** row.



- Enter the material's conductivity along each tensor axis in the **Value** box of the **T(1,1), T(2,2) and T(3,3)** rows.

For **Cylindrical**:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three rows named **z**, **theta**, and **r** are added below the **Thermal Conductivity** row.

	Name	Type	Value	Units
	Thermal Conductivity	Anisotro...		W/m-C
	z	Simple	0	
	theta	Simple	0	
	r	Simple	0	

- Enter the material's conductivity along each direction in the **Value** box of the **z** (axial), **theta** (circumferential), and **r** (radial) row.

Geometry Axis and Point of Origin

When applying cylindrical orthotropic thermal conductivity, Electronics Desktop determines the axis and point of origin differently based on the selected geometry.

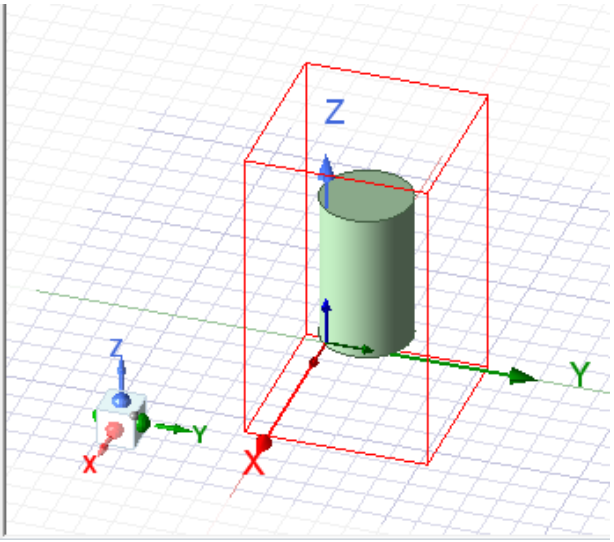
- Cylinder, frustum, sphere, and polyhedron geometry have an **Axis** defined in the **Properties** window. The point of origin for the cylindrical orthotropic thermal conductivity is the **Center Position** displayed in the **Properties** window.

> **Note:** For inclined geometry, define a relative coordinate system to ensure that the coordinate axis and geometry axis are aligned.

Properties

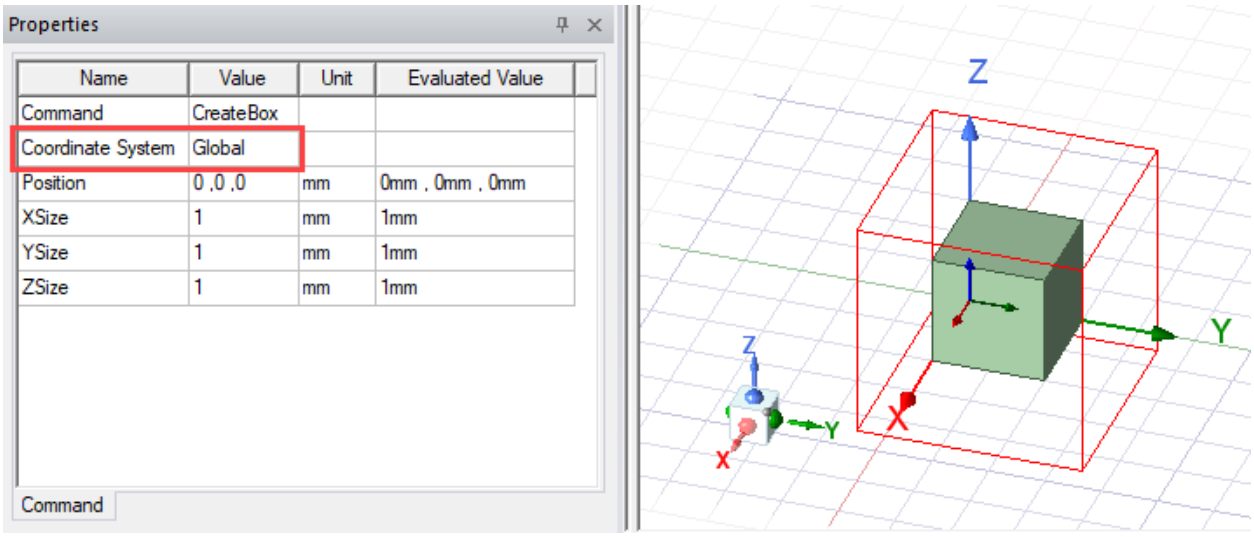
Name	Value	Unit	Evaluated Value
Command	CreateCyli...		
Coordinate System	Global		
Center Position	-0.2 ,0.2 ,0	mm	-0.2mm , 0.2mm , 0mm
Axis	Z		
Radius	0.3	mm	0.3mm
Height	1	mm	1mm
Number of Segme...	0		0

Command



- Box and CAD geometry axis information is determined by the assigned **Coordinate System**, which is displayed in the **Properties** window. The point of origin for the cylindrical orthotropic thermal conductivity is the centroid of the geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: When opening Icepak Classic projects (.tzr files) with objects that are assigned cylindrical orthotropic thermal conductivity, Electronics Desktop creates coordinate systems for the geometry based on the axis and origin information defined in the Icepak Classic project.

Limitations for Applying Cylindrical Thermal Conductivity

- Assignment to the native geometry shapes torus, helix, spiral, and bondwire is not supported.
- Assignment to parts of a PCB native component is not supported.
- Assignment to fluid materials is not supported.
- Assignment to geometries with arbitrary rotations relative to the global coordinate system. For example, a cylinder created in a rotated coordinate system consisting of non-orthogonal rotations, such as 45° along X axis, 45° along Y axis, and 45° along Z axis, is not supported.

Surface Material Properties

Specify the surface material to be used for each object or surface. This surface material defines the following properties:

Roughness

In fluid dynamics calculations, it is common practice to assume that boundary surfaces are perfectly smooth. In laminar flow, this assumption is valid, because the length scales of typical rough surfaces are much smaller than the length scales of the flow. In turbulent flow, however,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the length scales of the flow eddies are much smaller than laminar length scales; therefore, it is sometimes necessary to account for surface roughness. Surface roughness acts to increase resistance to flow, leading to higher rates of heat transfer.

Icepak assumes, by default, that all surfaces of an object are hydrodynamically smooth, and applies standard no-slip boundary conditions. For turbulent-flow simulations in which roughness is significant, however, you can specify a roughness factor for the entire object or for each individual side. The roughness factor is defined as part of the properties of the surface material for the object. The purpose of the roughness factor is to approximate the average height of the surface texture on the object.

> **Note:**
>
> Fluid flows over rough surfaces are encountered in diverse situations. See Wall Roughness Effects in Turbulent Wall-Bounded Flows in the Fluent User Guide for more information.
>
> - Surface roughness is only applicable to turbulent models that use wall function. That is, Two equation, Enhanced two equation, Enhanced RNG, Enhanced realizable two equation used with the mesh size with Wall Y Plus > 35. The turbulent models other than that ignore wall roughness.
> - With the applicable turbulent model, the first mesh size must be smaller than half of the specified surface roughness.
> - Small roughness size requires a smaller mesh size, but this may generate an inappropriate mesh size (approximately Wall Y Plus > 35) for turbulent model with wall function. This leads to the incorrect results.

Emissivity

Emissivity defines the surface's ability to emit thermal radiation.

Solar Behavior

Solar Behavior specifies the solar classification of a material. **Opaque** indicates that the surface will not allow solar radiation to pass through it. **Transparent** indicates that the surface will allow a portion of the solar radiation to pass through it.

Solar Absorption

Solar Normal Absorptance is the fraction of direct solar irradiation energy that is absorbed by the semi-transparent surface for a direct solar irradiation beam normal to the surface. Solar Diffuse Absorptance is the fraction of diffuse solar irradiation energy that is absorbed by the semi-transparent surface.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solar Transmittance

Solar Normal Transmittance is the fraction of direct solar irradiation energy that passes through the semi-transparent surface for a direct solar irradiation beam normal to the surface. Solar Diffuse Transmittance is the fraction of diffuse solar irradiation energy that passes through the semi-transparent surface.

Surface Diffuse Fraction

Surface Diffuse Fraction is the fraction of reflected radiation flux that is to be treated as diffuse. The default setting is 1, indicating that all of the radiation is diffuse. A diffuse fraction equal to 0 indicates purely specular reflected radiation. A diffuse fraction between 0 and 1 indicates partially diffuse and partially specular reflected energy.

Defining Variable Material Properties

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window 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 **MySpecificHeat** and define its value as **4**. To assign this property value to a material, type **$MySpecificHeat** in the **Specific Heat Value** box for the material. Be sure to include the prefix **$** 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.

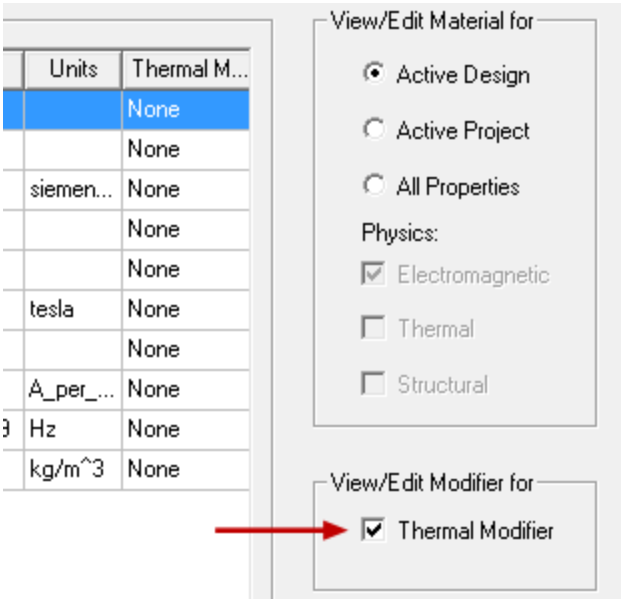
Specifying Thermal Modifiers

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

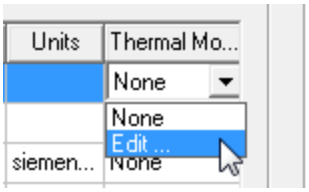
1. In the **View/Edit Material** window, you must enable the **Thermal Modifier** check box:



This option causes the properties table to expand to include a Thermal Modifier column.

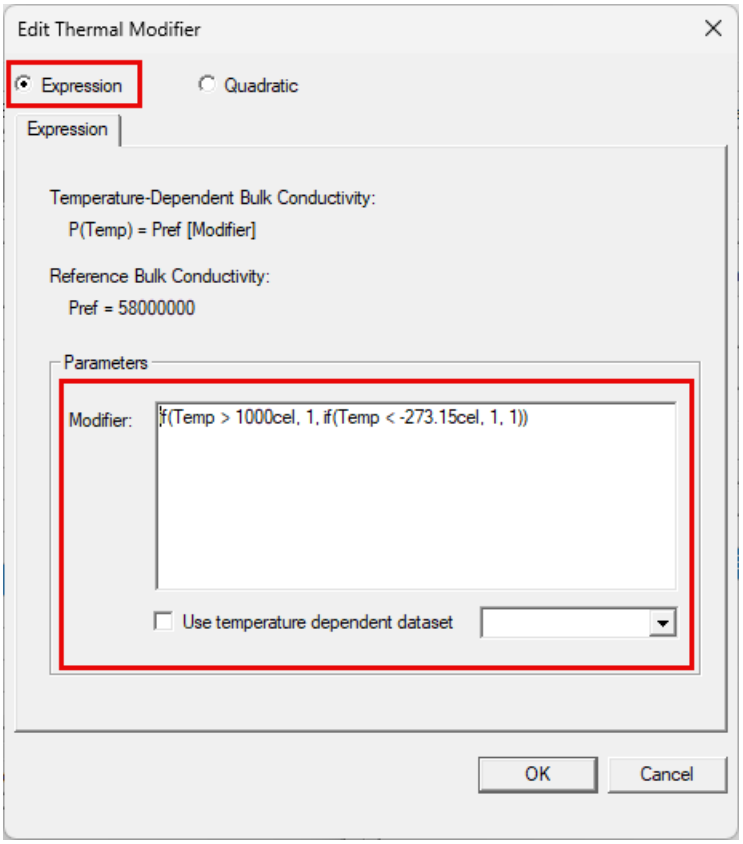
By default, the **Thermal Modifier** option is set to **None** for all material properties.

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.

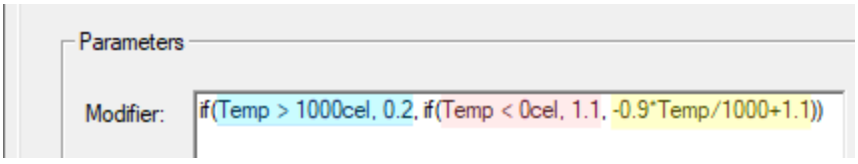
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

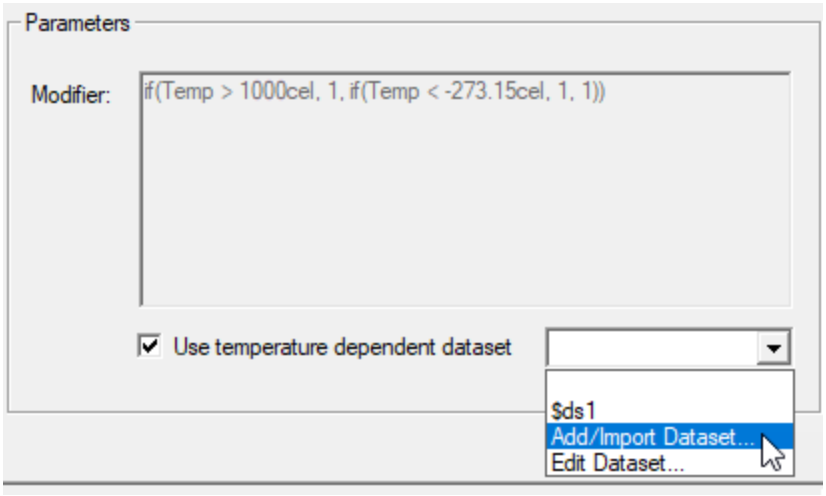


  In this example, the linear equation **-0.9 * Temp / 1000 + 1.1** (highlighted in yellow) defines the multiplier for the range 0° C ≤ Temp ≤ 1000° C.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

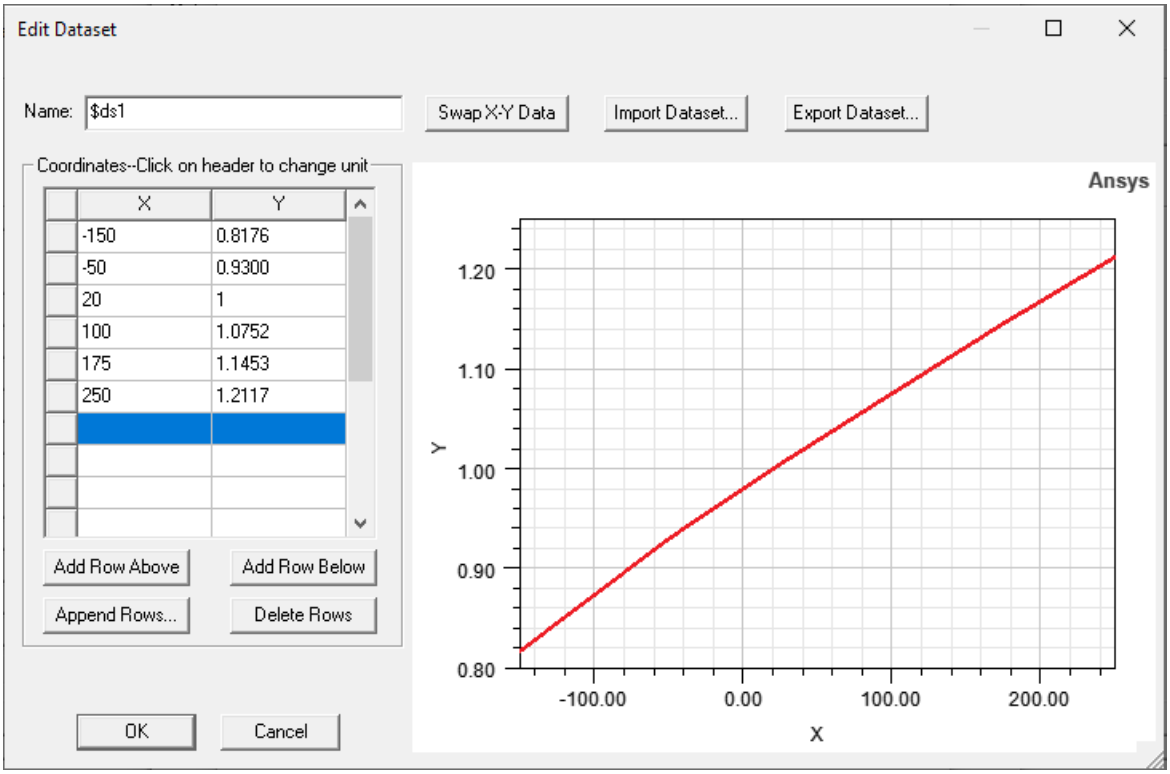


This command lets you define the thermal modifier dataset (a set of X and Y values). Alternatively, if a suitable project dataset already exists, it will be available for selection from the same drop-down menu.

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 2026 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 2026 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 Celsius 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 Celsius for correct results. The values in the **Y** column are multipliers and therefore dimensionless.

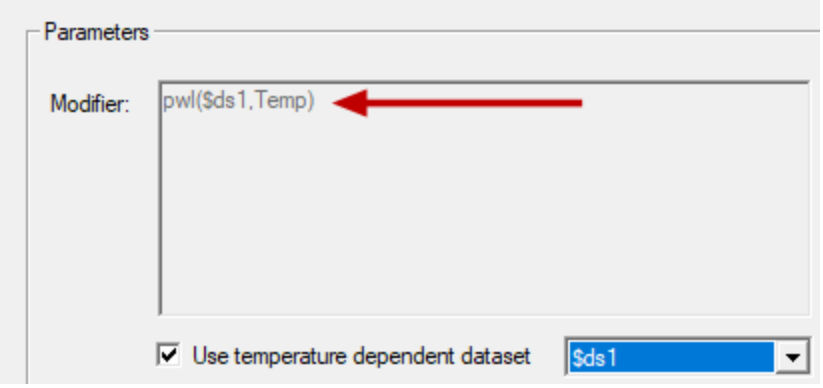
Alternatively, you can create a dataset based on any desired temperature units before defining your temperature-dependent material properties. To do so, click **Project** > **Datasets** from the menu bar and then choose **Add**. If your project includes a design type that supports 3D datasets, a submenu 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.

Warning:

If the dataset you want to import has comments, the data import will fail.

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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

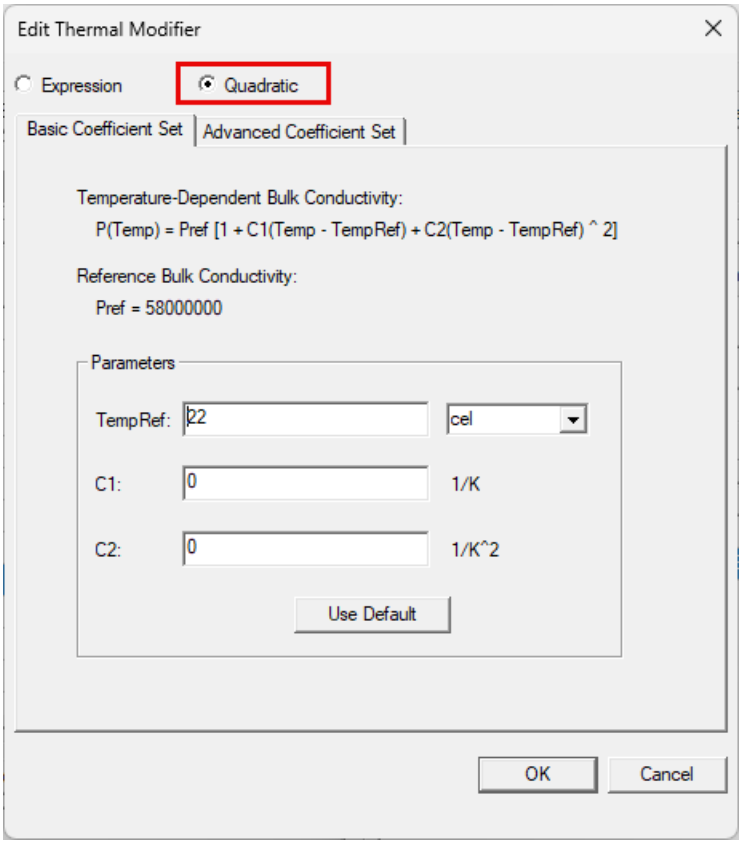
The syntax of the dataset expression is explained as follows:

- **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**.

3. Alternatively, select the **Quadratic** option to display the tabs for **Basic Coefficient Set** and **Advanced Coefficient Set** (as shown below):



- Under the **Basic Coefficient Set** tab, you can edit fields for the TempRef and units, and fields for C1 and C2 (see preceding illustration) for the following equation:

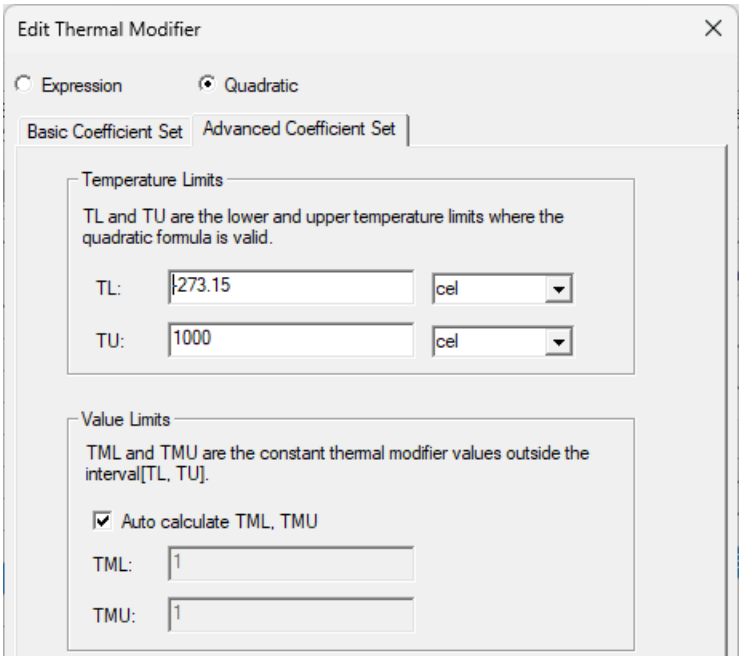
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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.



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).

4. Click **OK** to accept your edits and return to the View/Edit materials window.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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

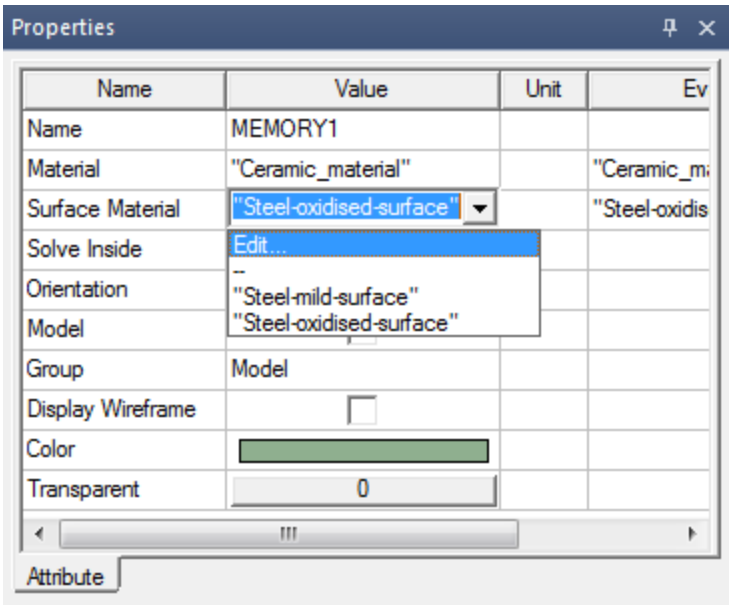
Any material property that can be specified by entering a constant can also be specified using a mathematical function. This is useful when you are defining a material property whose value is given by a mathematical relationship — for instance, one relating it to frequency or another property's value. 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 or surface 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 or surface materials to assign the selected material to that object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

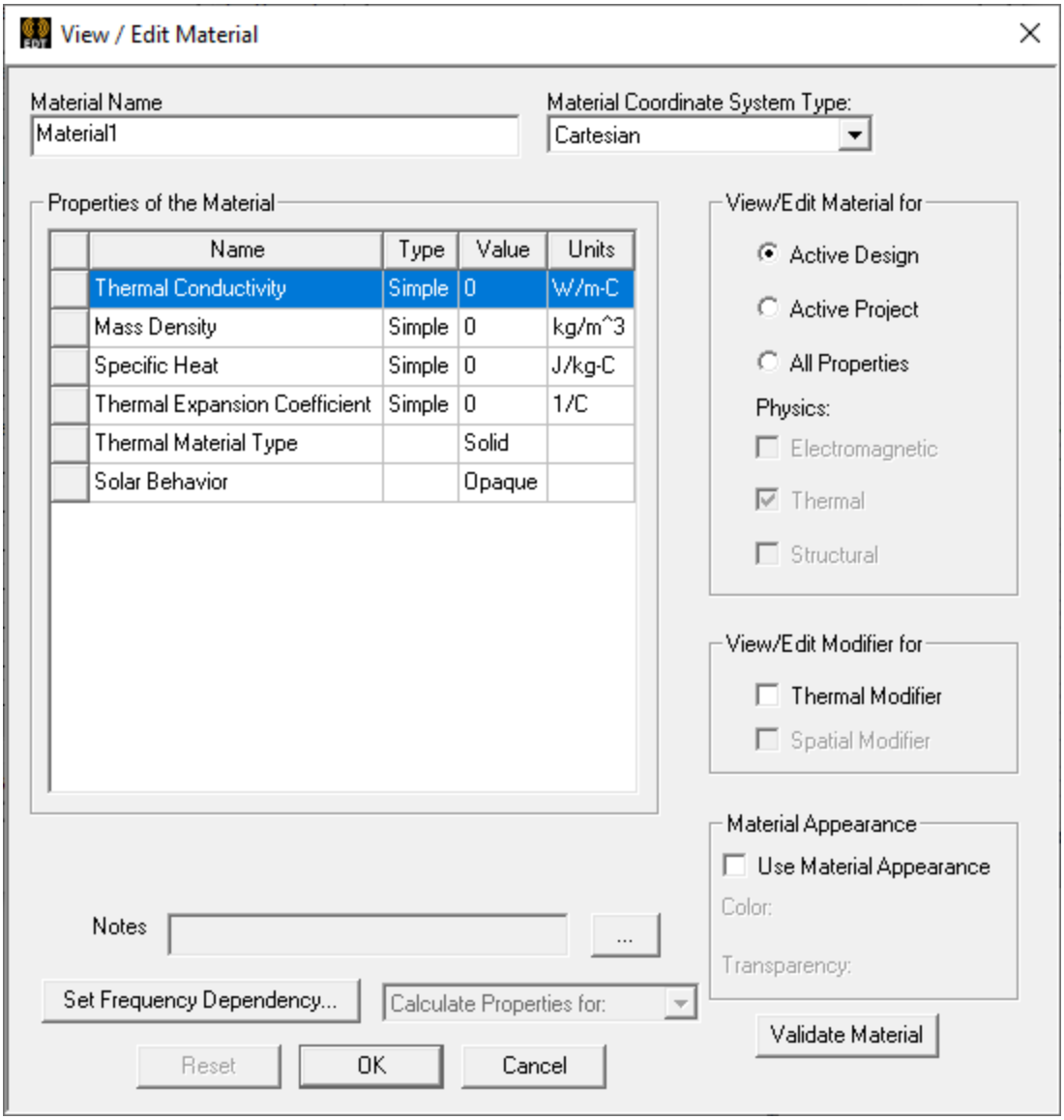


Viewing and Modifying Material Attributes

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 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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 Icepak design but Thermal physics type was not set), the **Physics** type will be automatically set in the material.
>
> **Physics** classification options are unavailable for surface materials.

3. You can modify the material as follows:
   a. Provide a new name for the material in the **Material Name** or **Surface Material Name** text box.
   b. Depending on material property selected, the types available may be **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. Enable the **Use Material Appearance** check box to specify a color and transparency.

> **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** window to add the new material to the material library.
6. Click **OK** to save the changes and return to the **Select Definition** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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 the Electronic Desktop global material library.
- Materials that have been assigned to objects in the active project.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 listed in descending order (9 to 1, Z to A), click the column heading again. The arrow will point down.

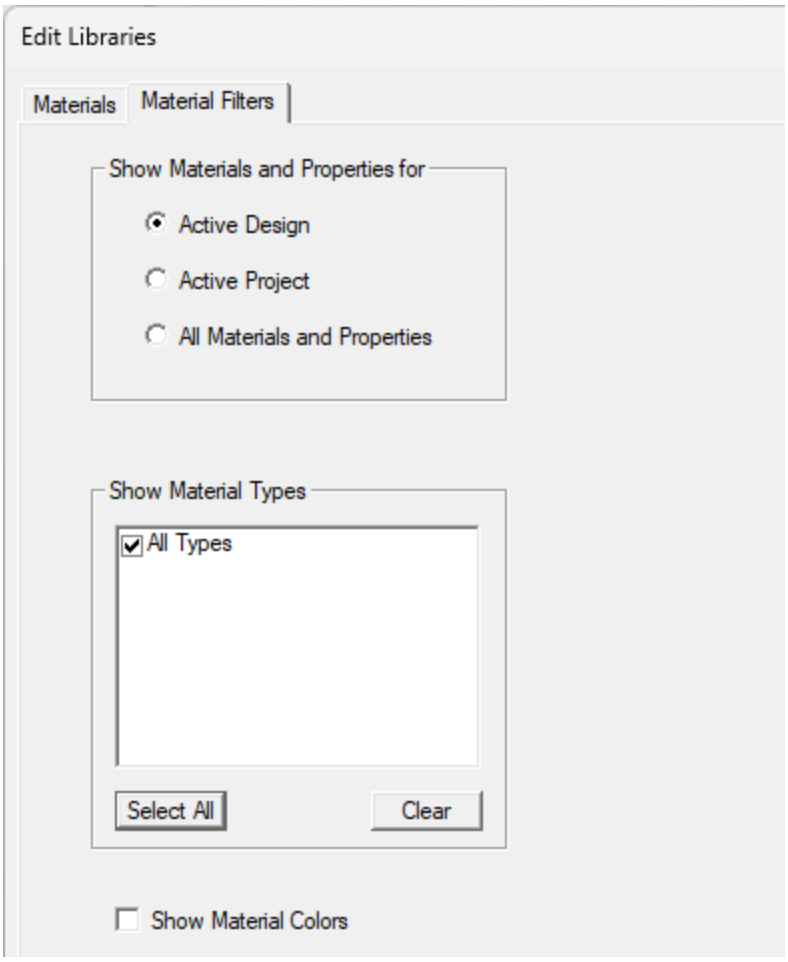
> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

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** or **Surface Material Filters** tab. You can filter out materials based on the product or library with which they are associated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To filter out or show additional materials/properties in the **Materials** or **Surface Materials** tab:

1. Click the **Material Filters** or **Surface 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** or **Surface 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** or **Surface 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 Icepak: 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Creating a Model

After you insert a design into the current project, you can create 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, or use the 3D component libraries, a 3D Layout Component, or a 2.0 Component.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

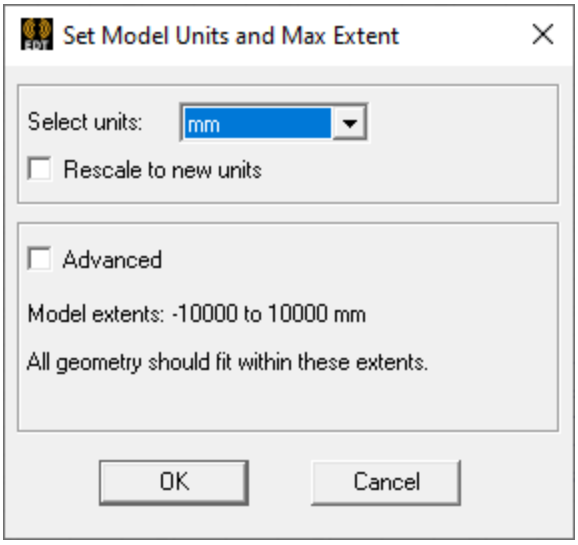
Setting Units of Measurement for the Model

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. 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**.

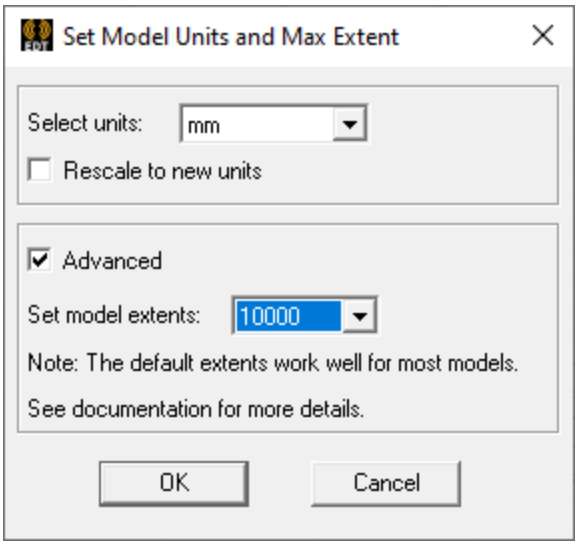
   The **Set Model Units** dialog box appears.



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 2026 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 2026 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 2026 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

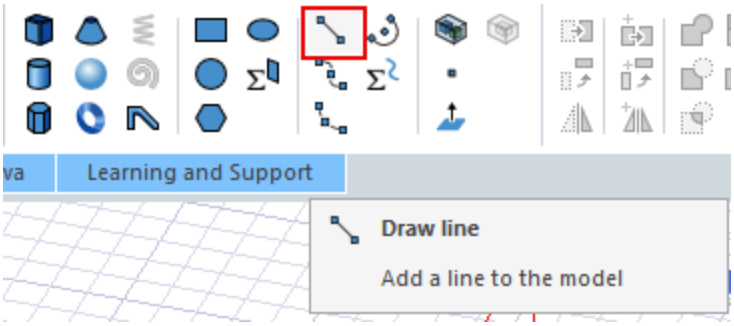
After you draw an object in the **3D Modeler** window, you can modify the object's properties, such as its position, dimensions, or color, in the **Properties** dialog box. Most model object properties can be assigned as Design variables when can then be manipulated during the solve to test their effect 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

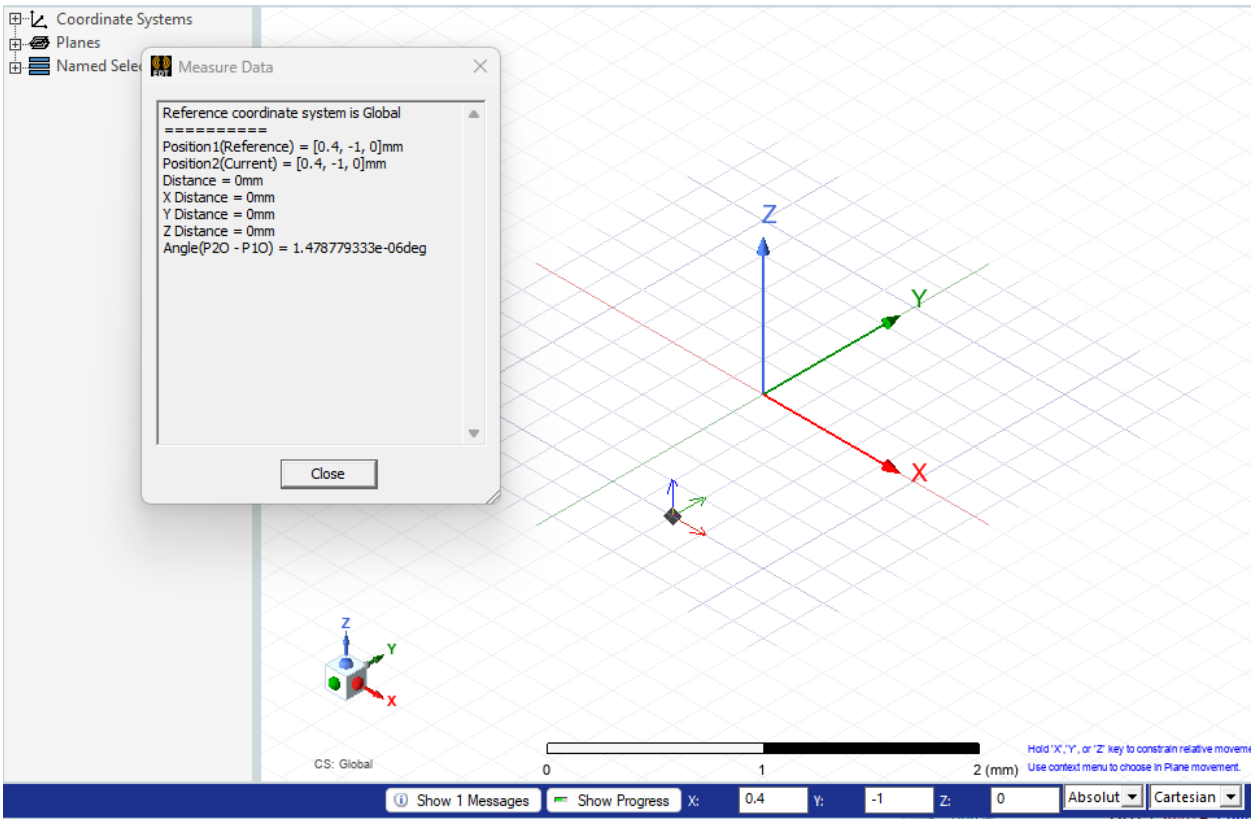
To create an object with one or more straight line segments, use the **Draw** > **Line** command.

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 dialog box for the precise coordinates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

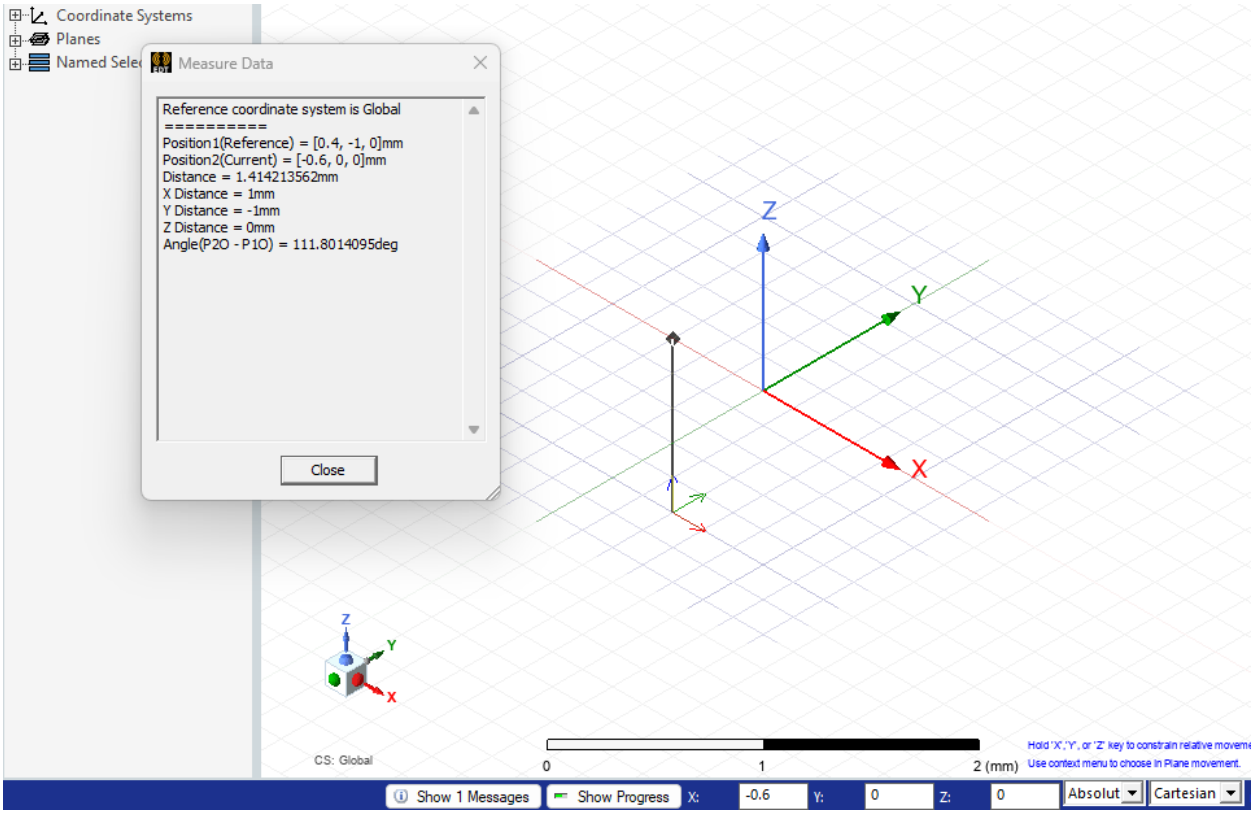


- You can accept the first point or change it by editing its coordinates in the X, Y, and Z boxes in the status bar. Note that the Status bar includes selections for Absolute or Relative Coordinates, as well as for Cartesian, Cylindrical and Spherical Coordinates.

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.

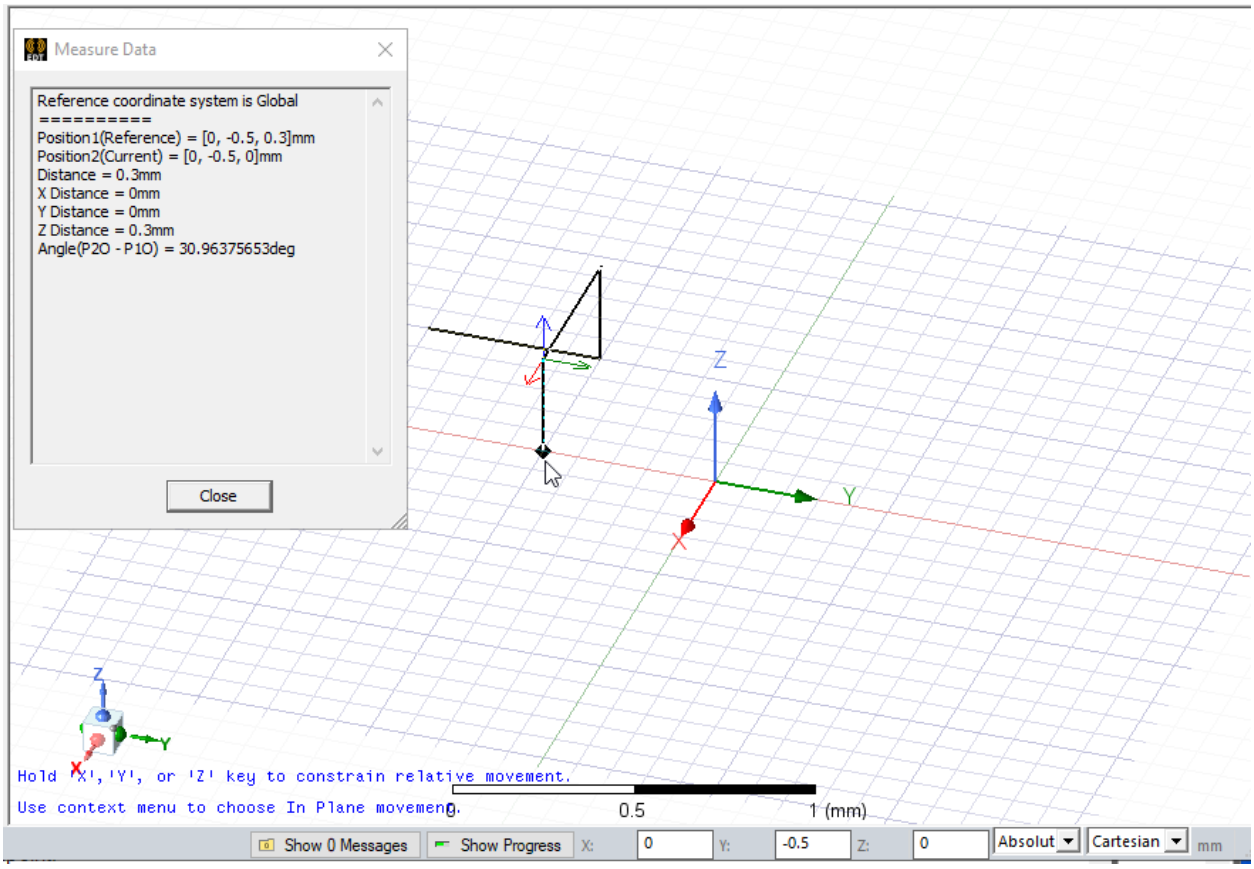
3. Select the next point of the line by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2026 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.

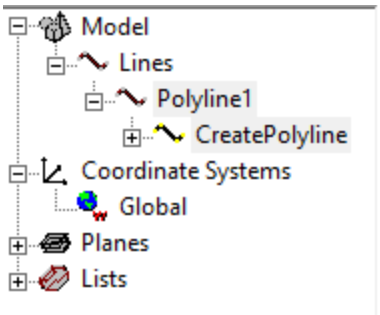
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

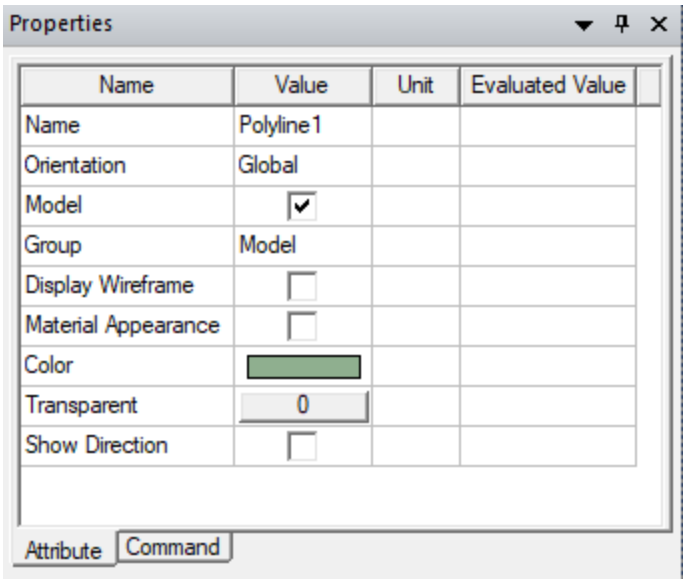
4. Complete the line in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the context (right-click) menu.
   - Press **Enter**.

   The new line appears selected in the Modeler window and shown in the History tree.

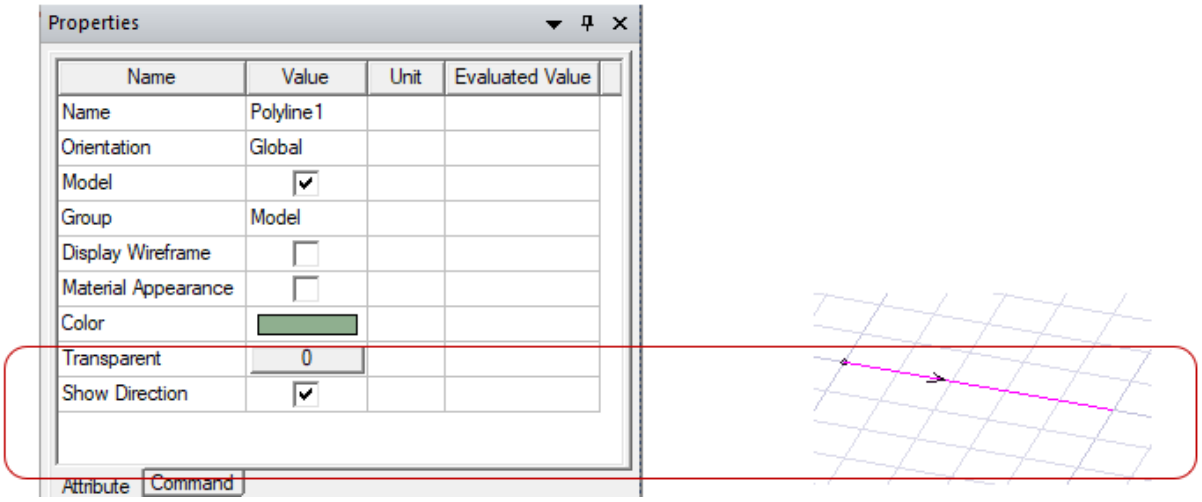


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

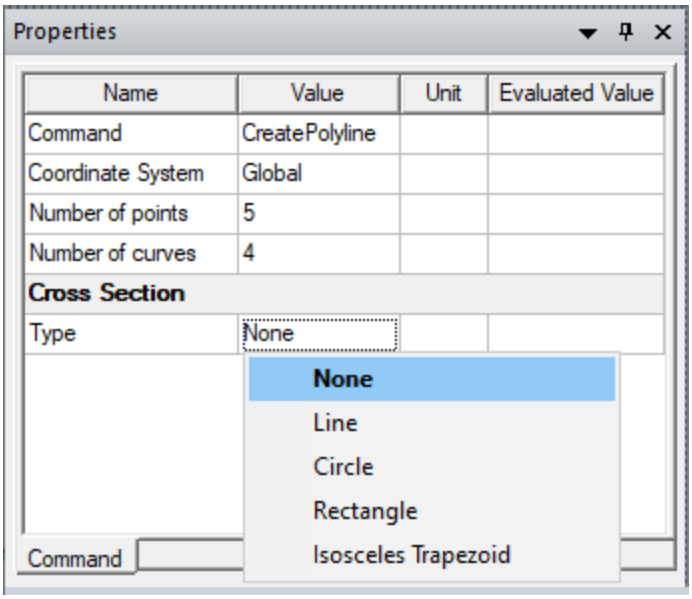


Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.



Ansys Electromagnetics Suite 2026 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.

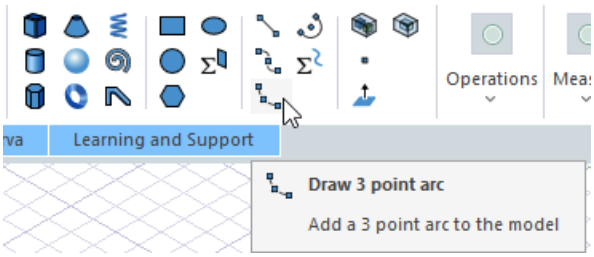
> **Note:** While drawing a polyline, you can switch between straight line, arc line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Three-Point Arc Line

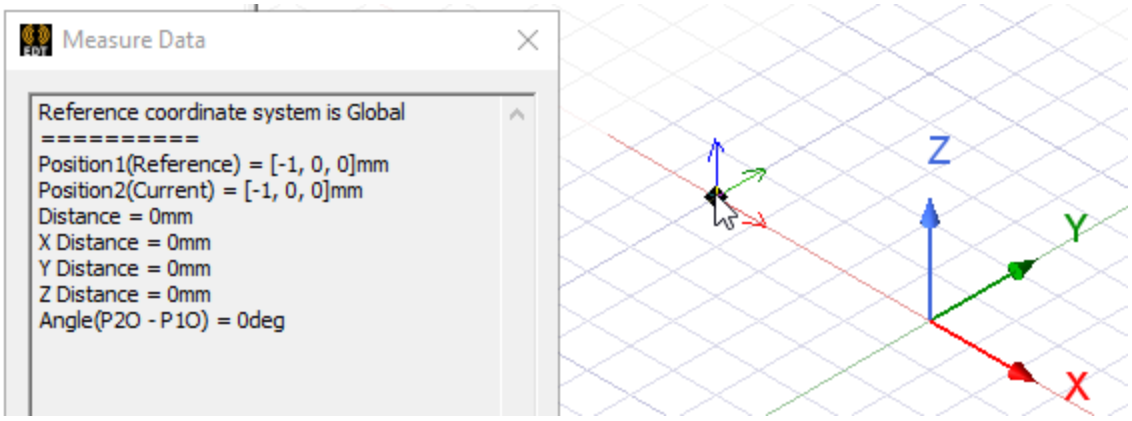
In the modeler, a three-point arc line segment is an arced line defined by three points on its curve. Use the **Draw** > **Arc** > **3 Point** command to create a polyline object with one or more arc line segments. Before you draw a three-point line arc, you can specify the coordinate system, and you 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:

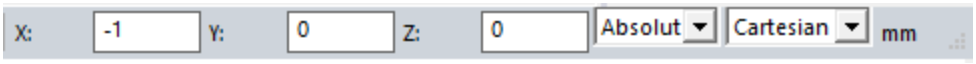
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



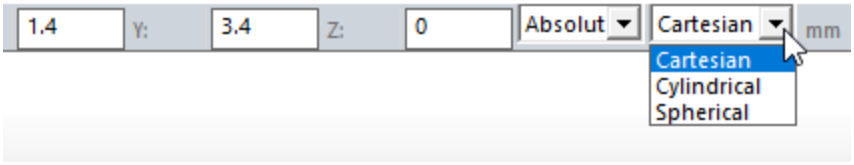
2. Select the start point of the arc in one of the following ways:
   - Click the point.



   - 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 2026 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.

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.

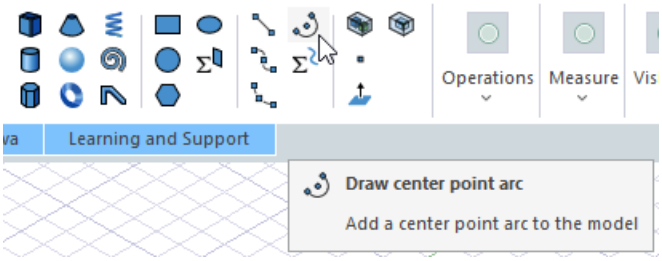
> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Center-Point Arc Line

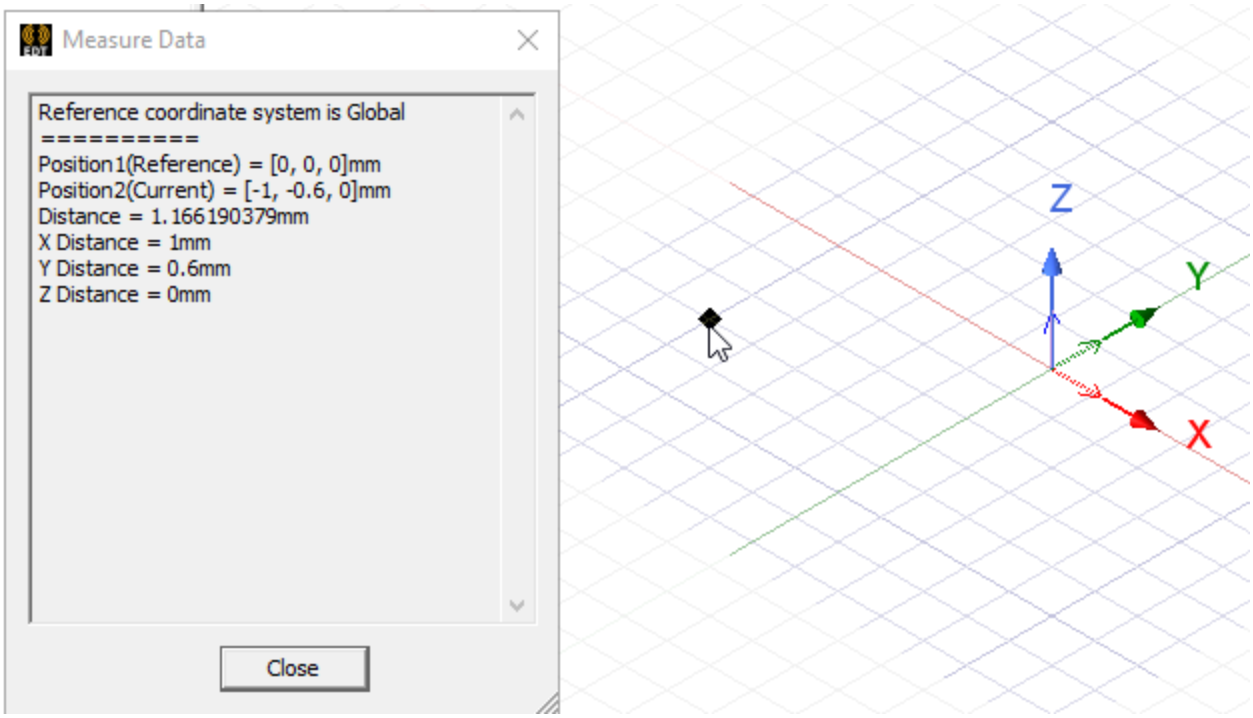
In the modeler, a center-point arc line segment is an arced line defined by a center point, start point and angle. Use the **Draw** > **Arc** > **Center Point** command to create a polyline object with one or more center-point arc line segments. Before you draw a center-pint arc line, you can specify 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 2026 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.

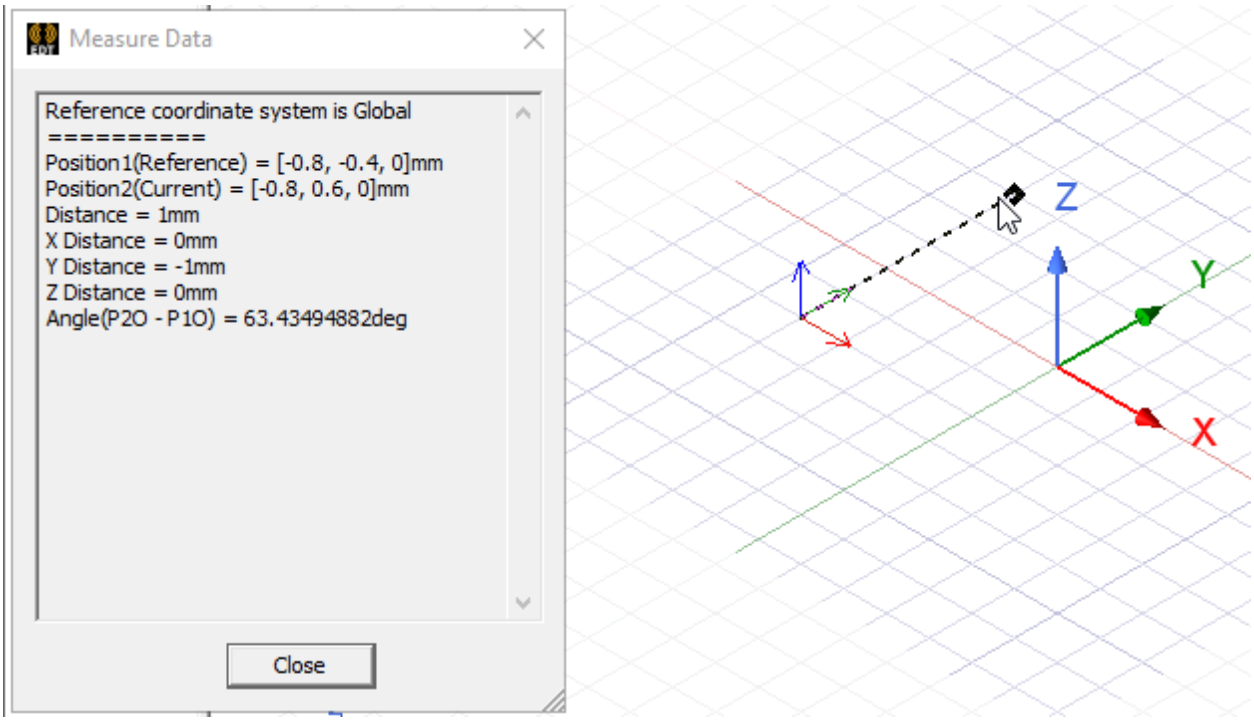


   - Type the point's coordinates in the text boxes in the status bar.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

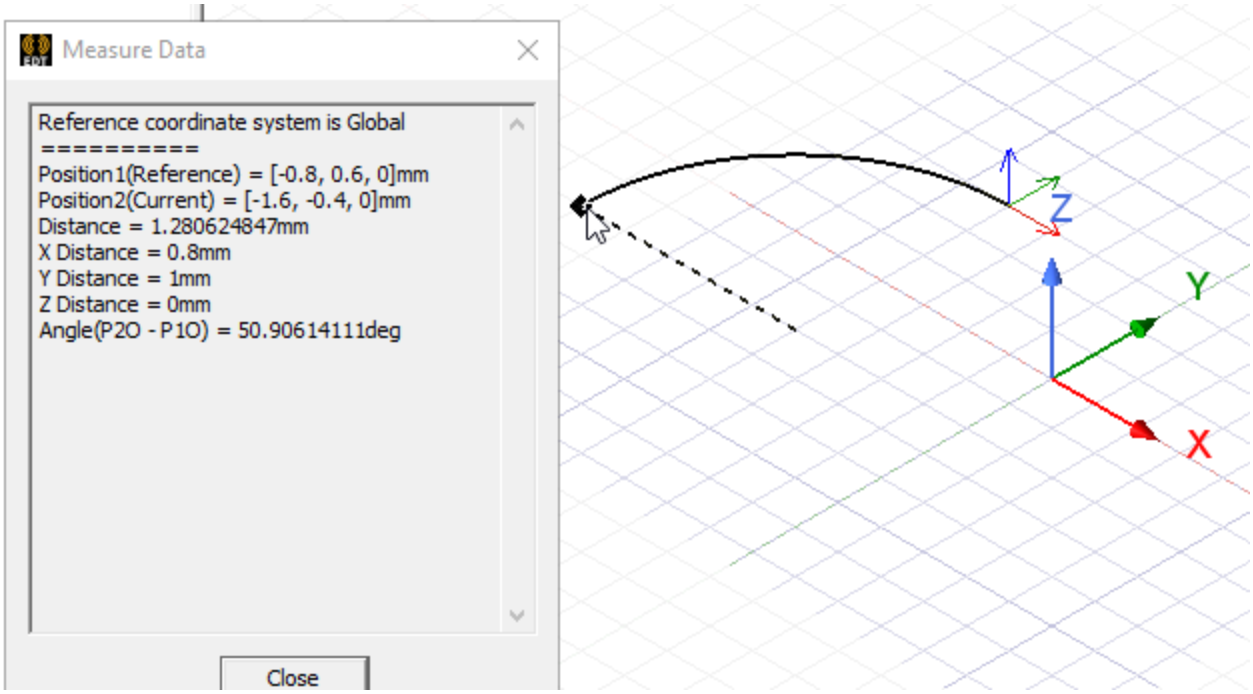


To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

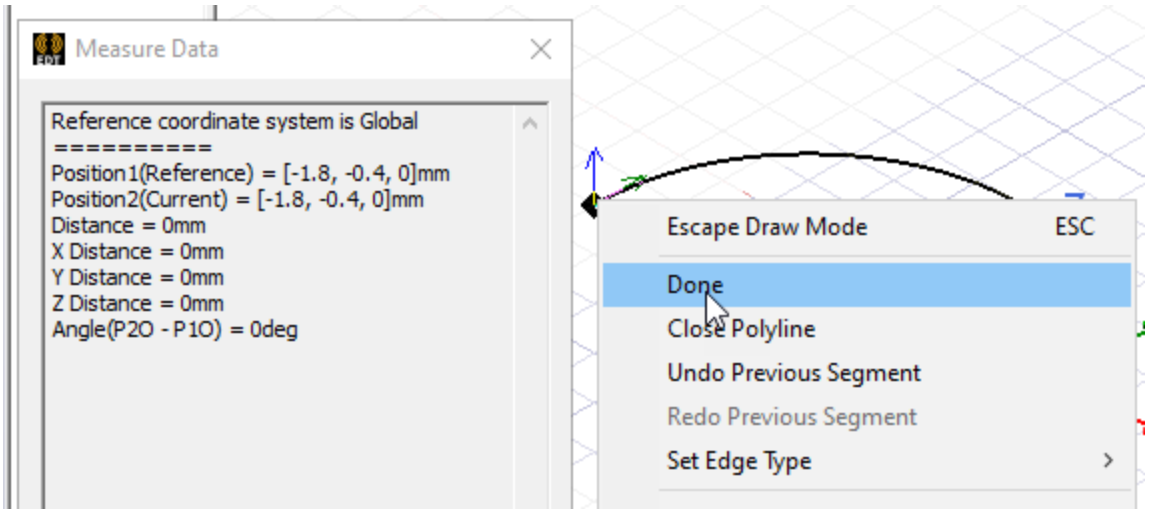
To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Sweep the angle or the endpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.



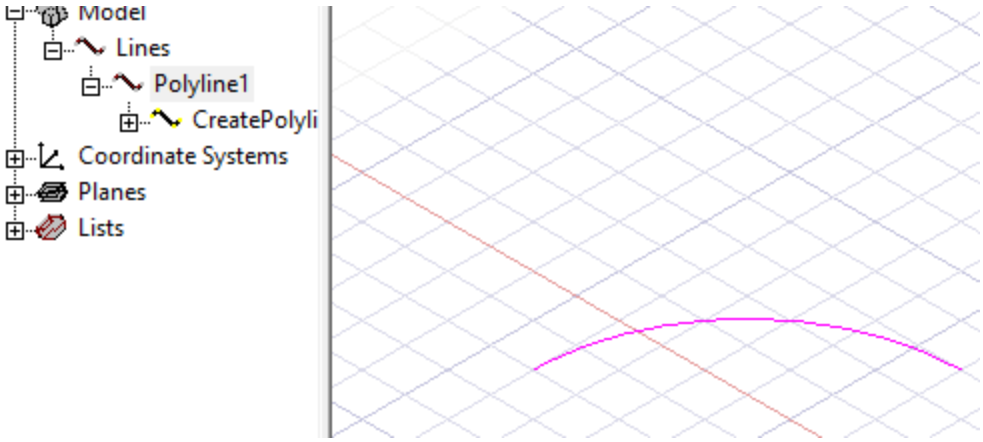
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.



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 2026 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.



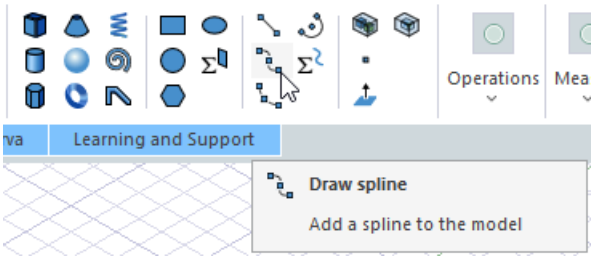
6. Click **OK**.

> **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.

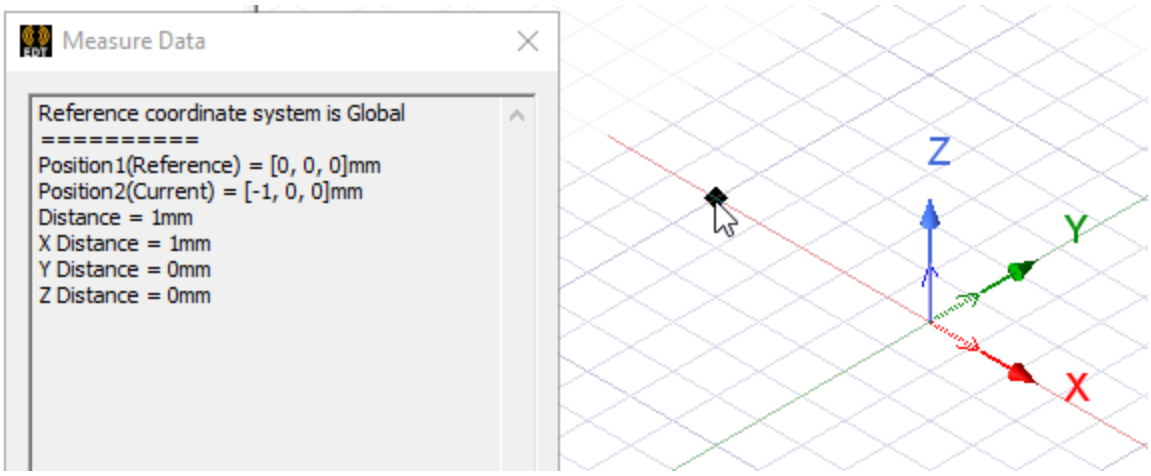
1. From the menu bar, click **Draw** > **Spline**, or on the **Draw** ribbon tab, click the **Draw spline** icon:



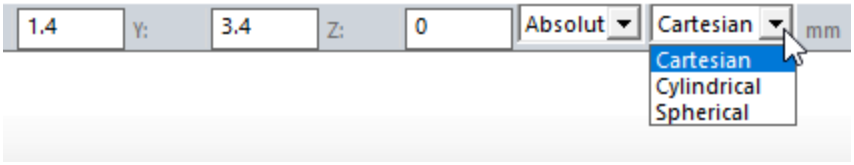
2. Select the spline's start point in one of the following ways:

Ansys Electromagnetics Suite 2026 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.

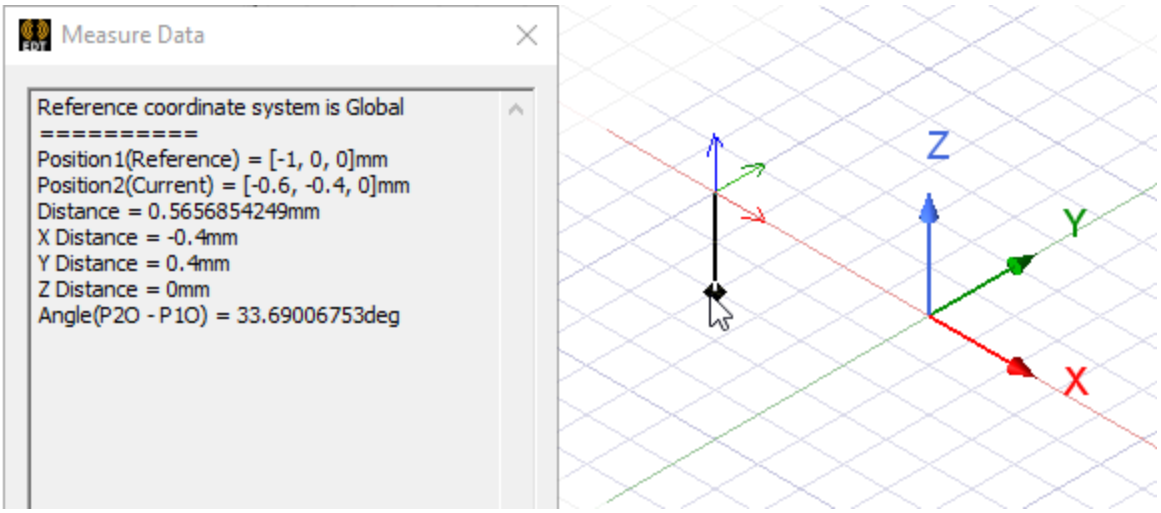


To delete the last point entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

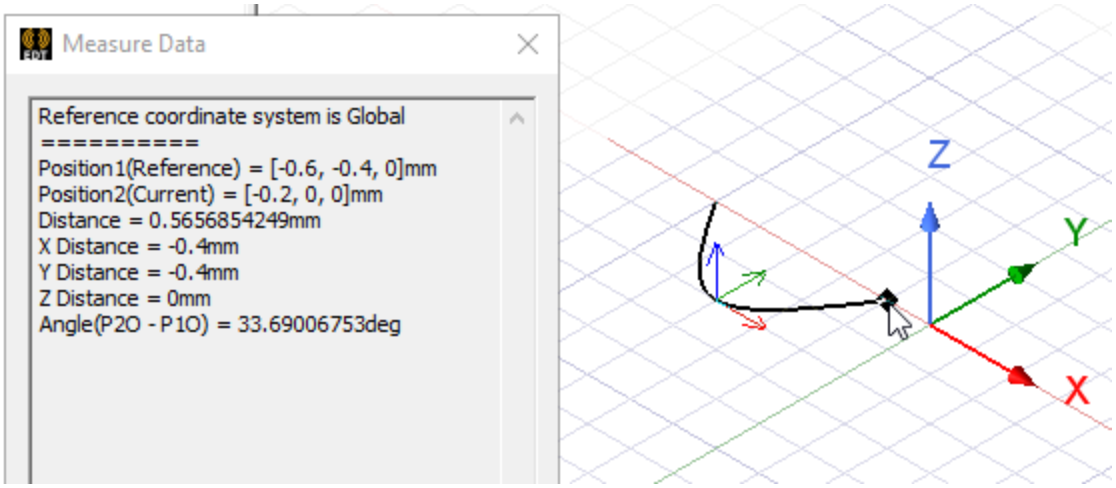
To delete all selected points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2026 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.

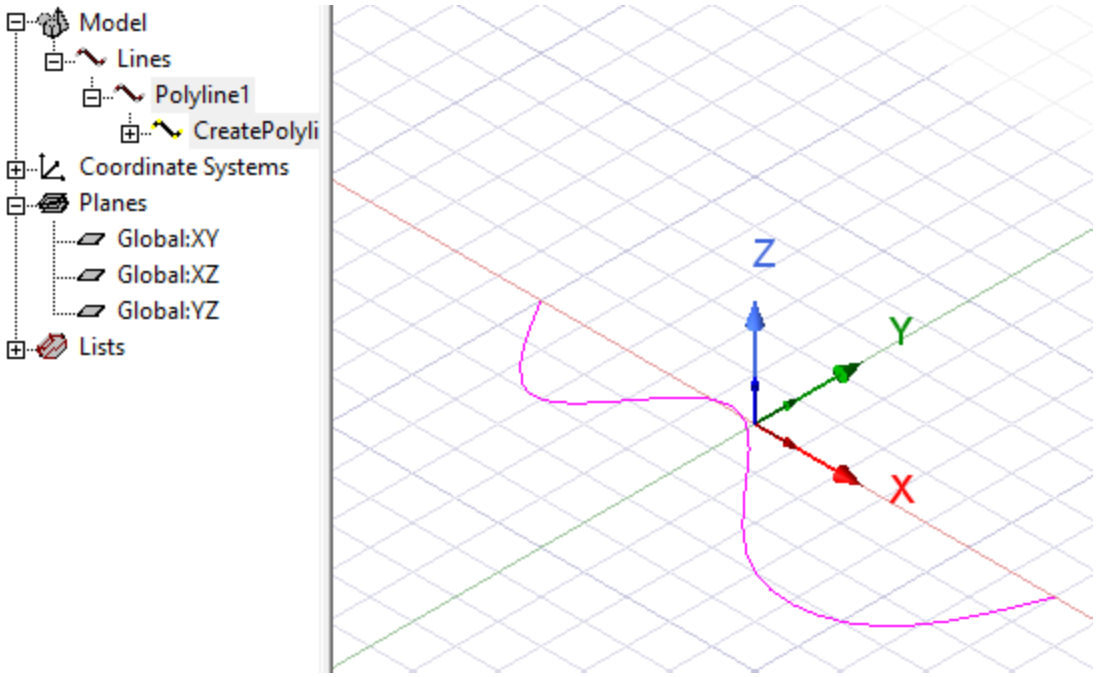


The endpoint serves as the start point for a subsequent spline segment. You can complete the spline at this point, or add additional segments.

5. Complete the spline in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.

Ansys Electromagnetics Suite 2026 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.

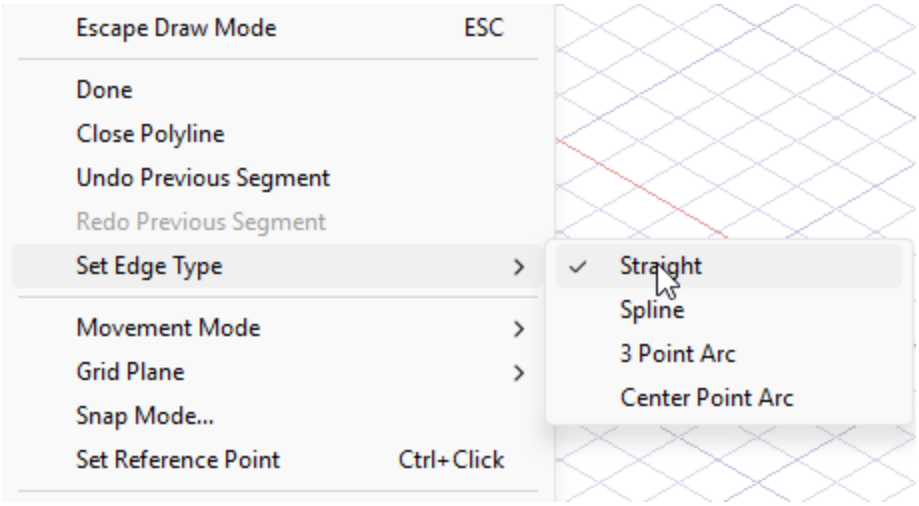
6. If you have edited Properties, click **OK**.

> **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

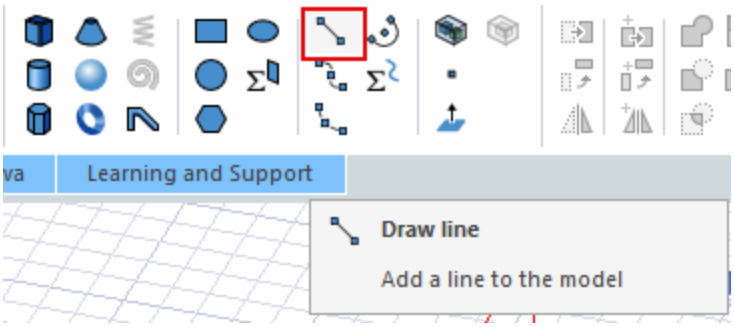


Before drawing a polyline, determine how closed polylines will be handled in the 3D Modeler options, as follows:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   - 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. 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.

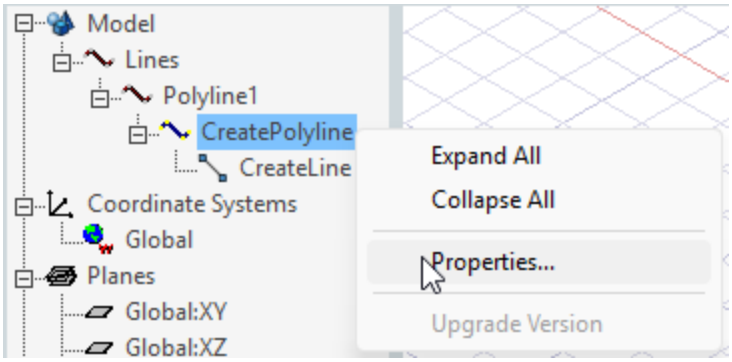
Assigning a Cross Section and Dimensions to a Polyline

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:

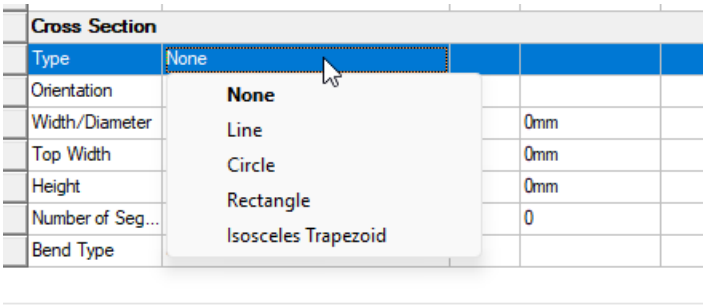
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the History Tree of the Modeler window, select the **CreatePolyline** entry for the polyline that you want to give a cross section.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

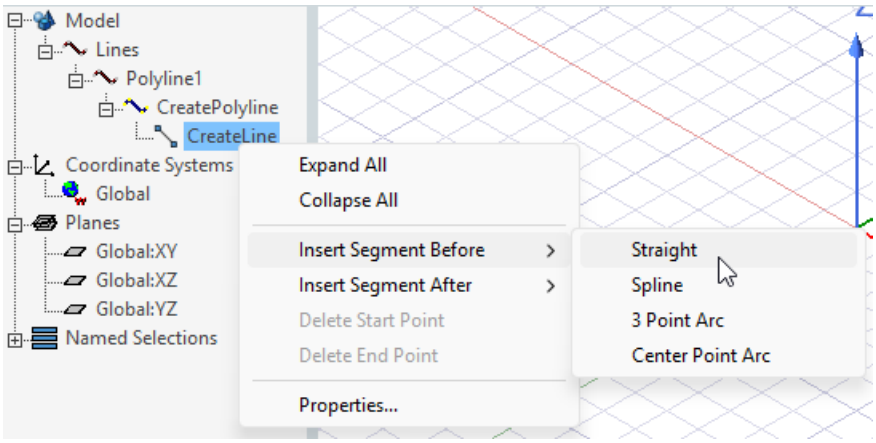
5. Specify the dimensions and select the units for the Cross section.

   Type a value in the dimension field(s) and select units from the drop-down menu.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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.

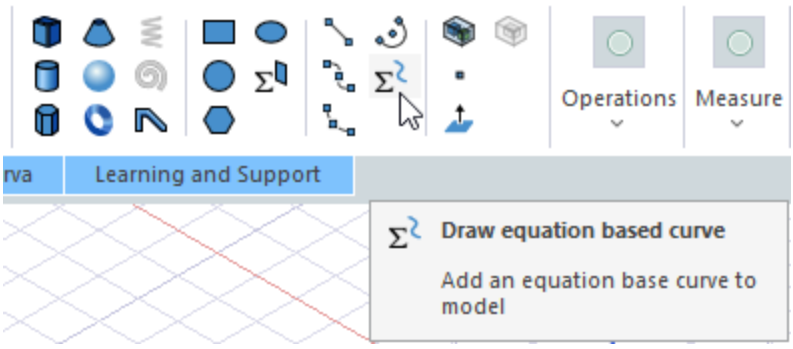
> **Note:**
>
> When inserting straight segments or arcs, the insertion operation will terminate as soon as you complete the definition of a single new segment. For splines, you can insert a spline with any number of defining points, and the insertion operation terminates as soon as you terminate the 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

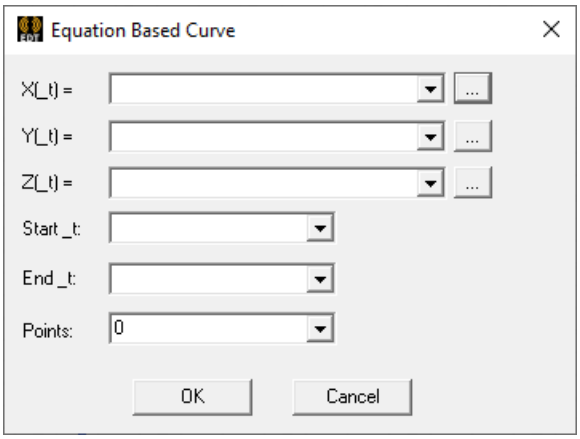
Any line that can be described by an equation in three dimensions can be drawn. Before you draw am equation-based curve, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

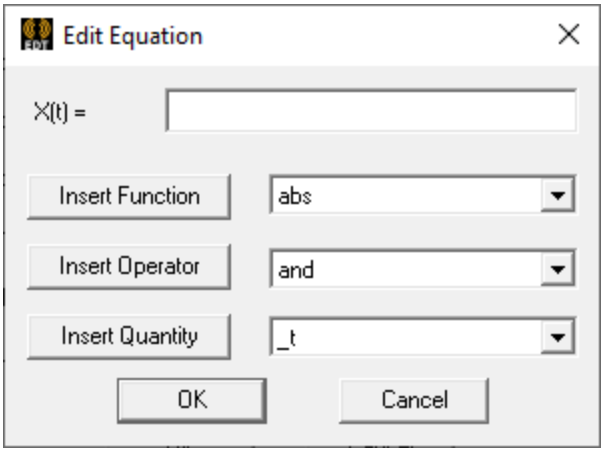


2. Type equations for **X(_t)**, **Y(_t)**, and **Z(_t)**

   You can also define an equation by doing the following:

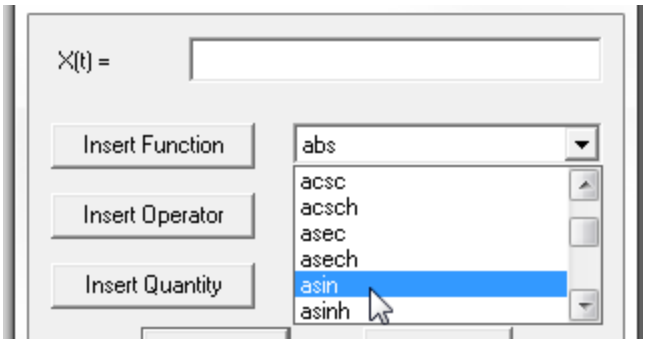
   a. Click an ellipsis button **[...]**.

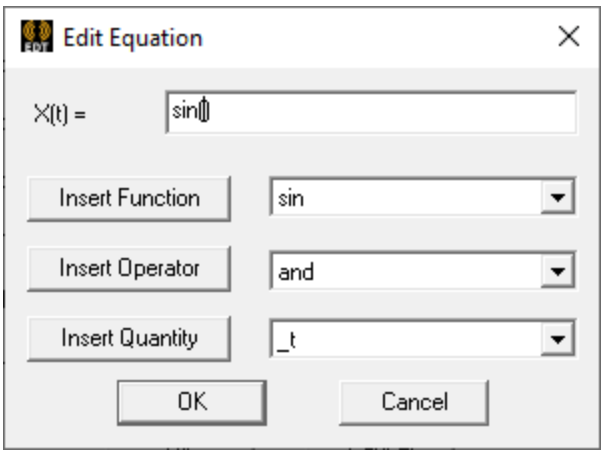
      The **Edit Equation** dialog box appears:



   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:

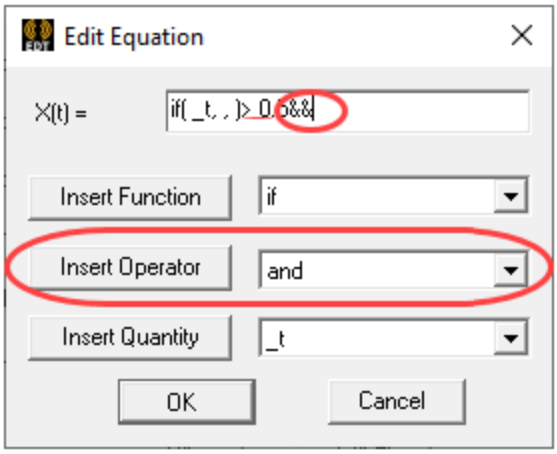
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

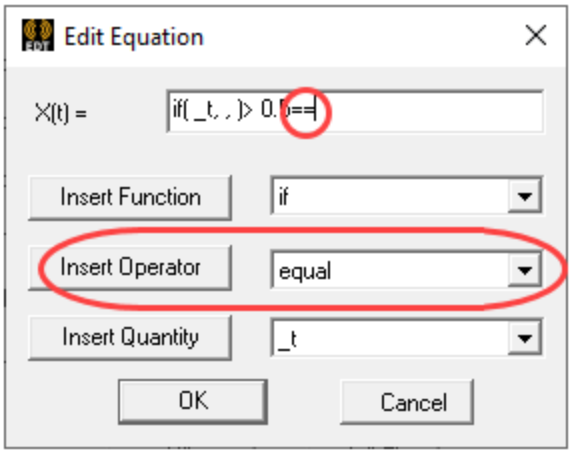




Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

- The **Insert Operator** button inserts corresponding operator:





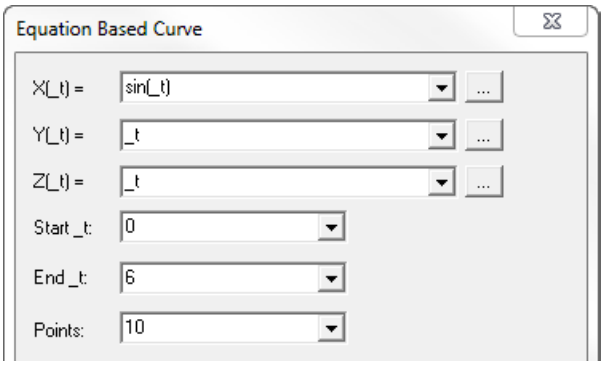
- Use a combination of the preceding two methods to build complex equations.

Ansys Electromagnetics Suite 2026 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.

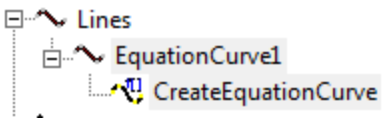
> **Note:**
>
> - The equations are taken to be meters. Use of units in equations is not properly supported.
> - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
> - Equation based curves depend 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.

   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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

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
tanh	Hyperbolic tangent
Operators	

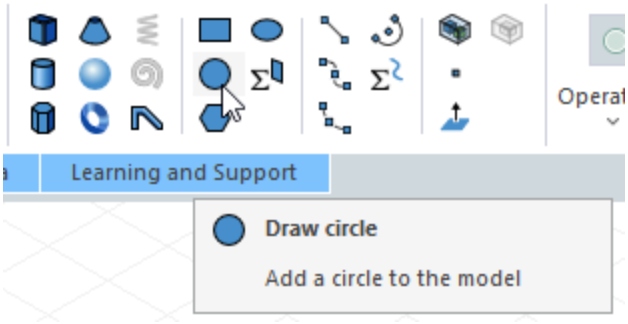
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Definition
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

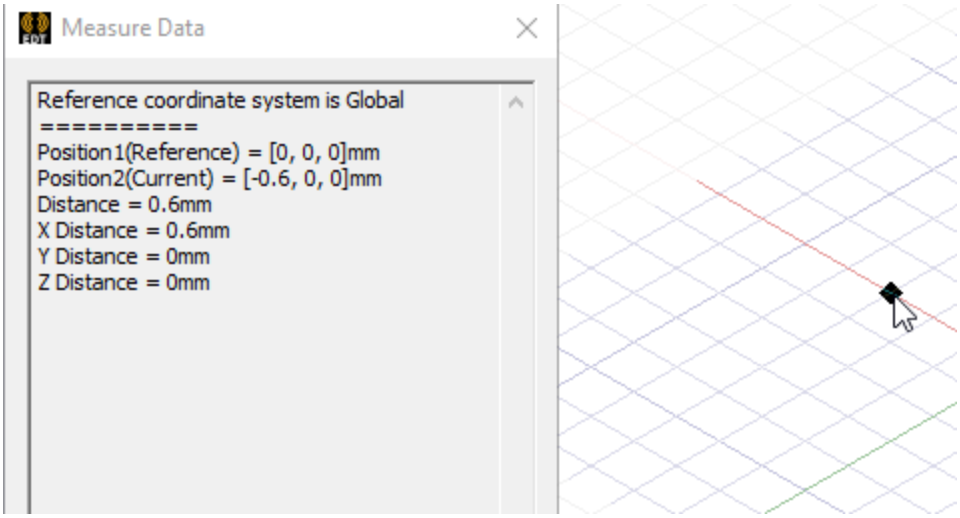
Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces in the modeler. Before you draw a circle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Circle** or on the **Draw** ribbon tab, click the **Draw circle** icon:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

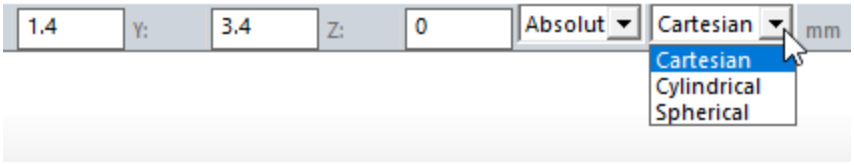
2. Select the center point of the circle in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.



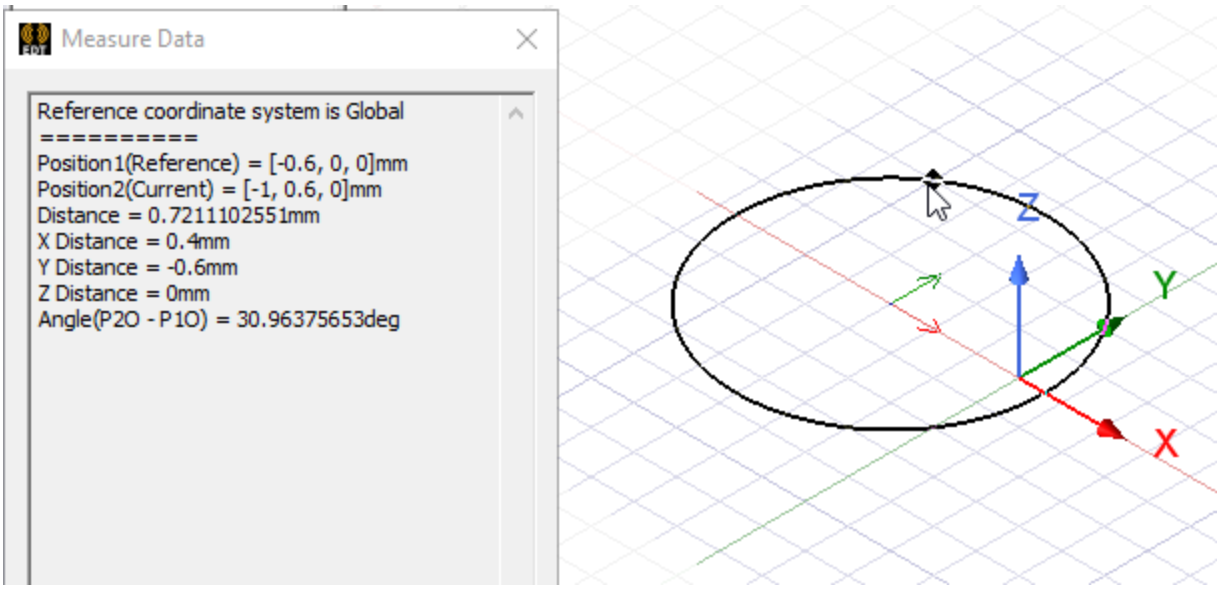
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the circle in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the radius by selecting a point on the circle's circumference in one of the following ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

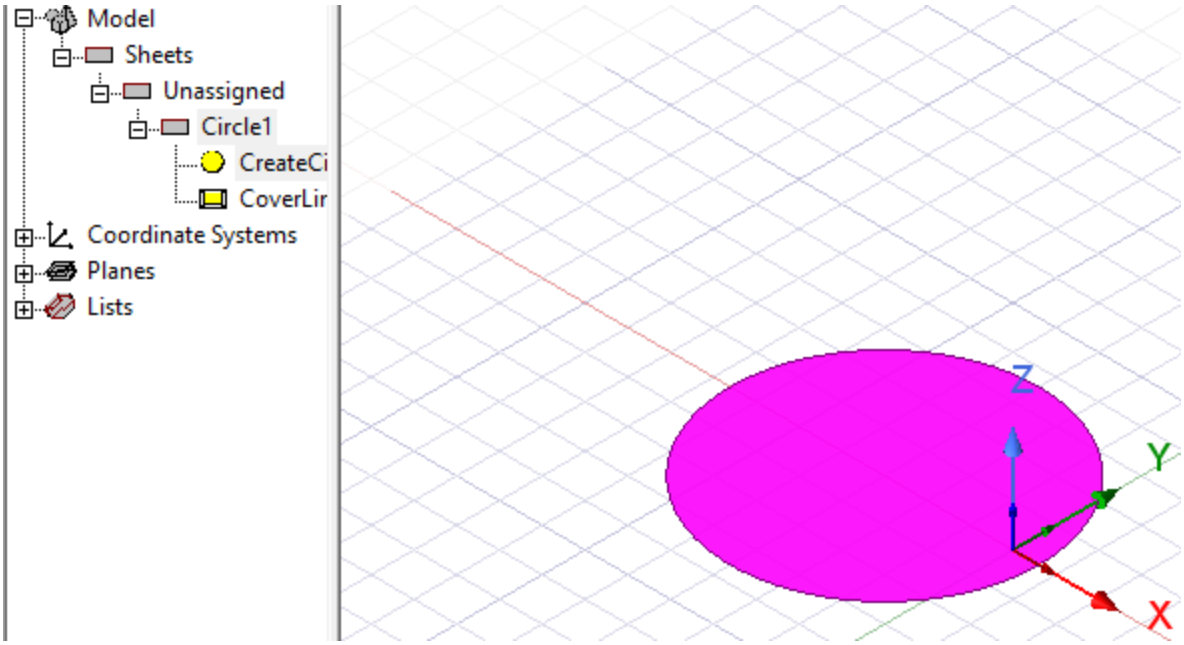
- Click the point.



- 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.

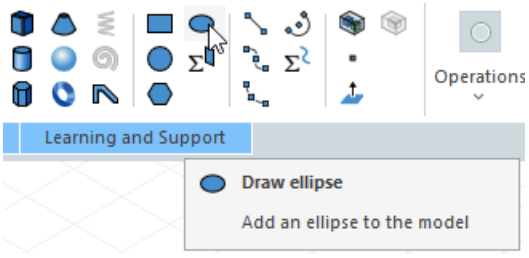
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing an Ellipse

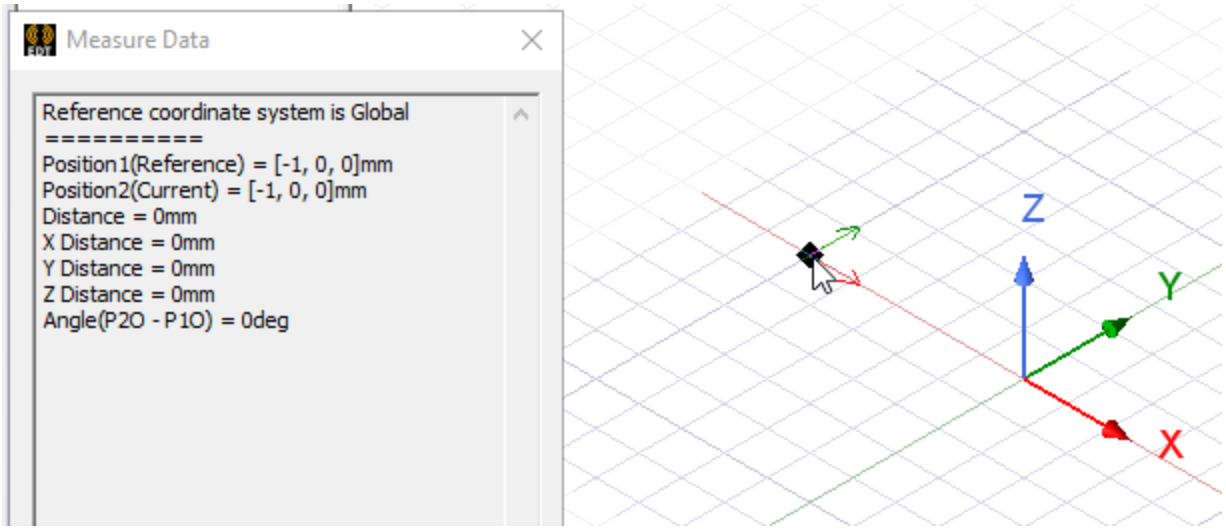
Draw an ellipse by specifying a center point, base radius, and secondary radius. Before you draw an ellipse, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Ellipse**, or on the **Draw** ribbon tab, click the **Draw ellipse** icon:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

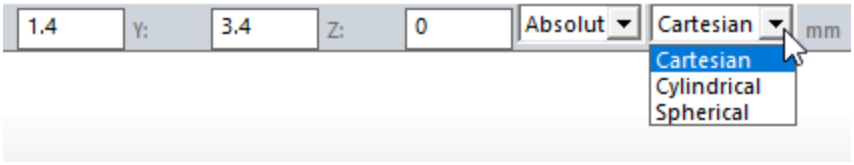
2. Select the center point of the ellipse in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

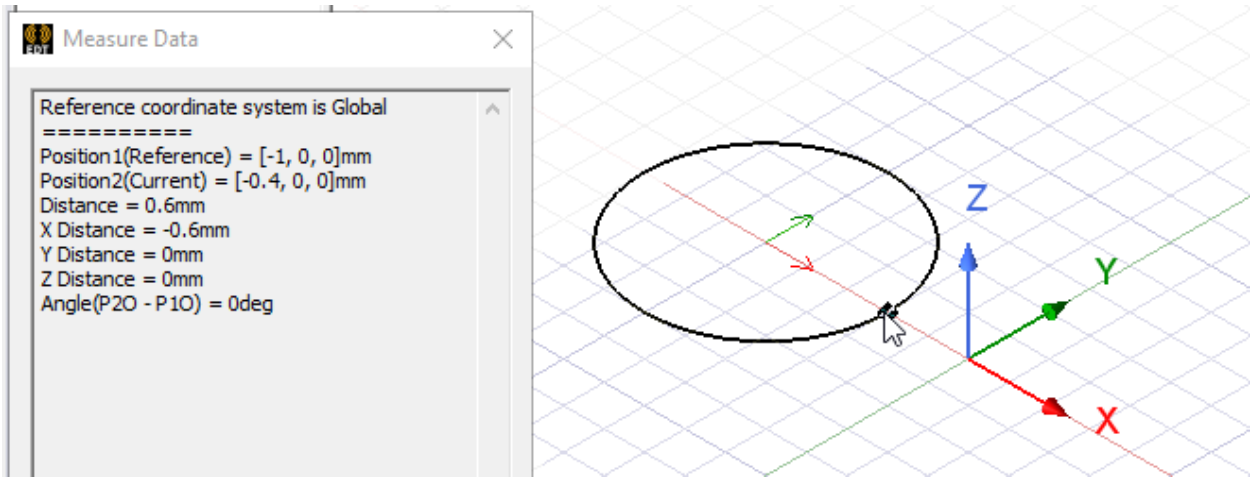


     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the ellipse in Cartesian, Cylindrical, or Spherical coordinates.



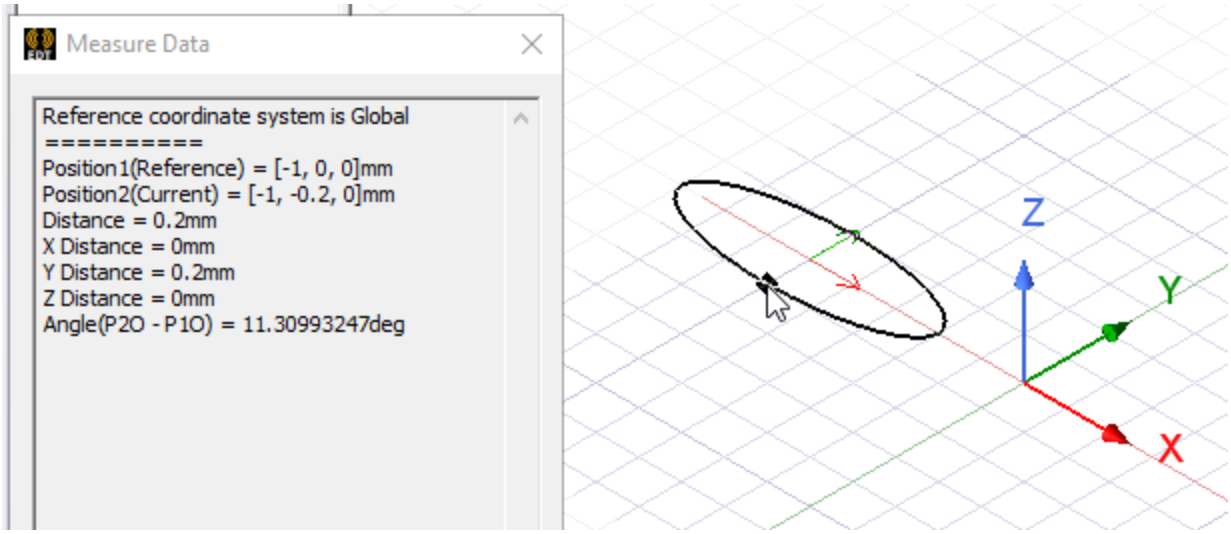
3. Specify the base radius of the ellipse. If the current drawing plane is xy, then x is the base radius direction. If the drawing plane is yz, then y is the base radius direction. If the drawing plane is xz, then z is the base radius direction. Select the point in one of the following ways:
   - Click the point. Ansys Electronics Desktop constrains mouse movement to the base radius direction.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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.

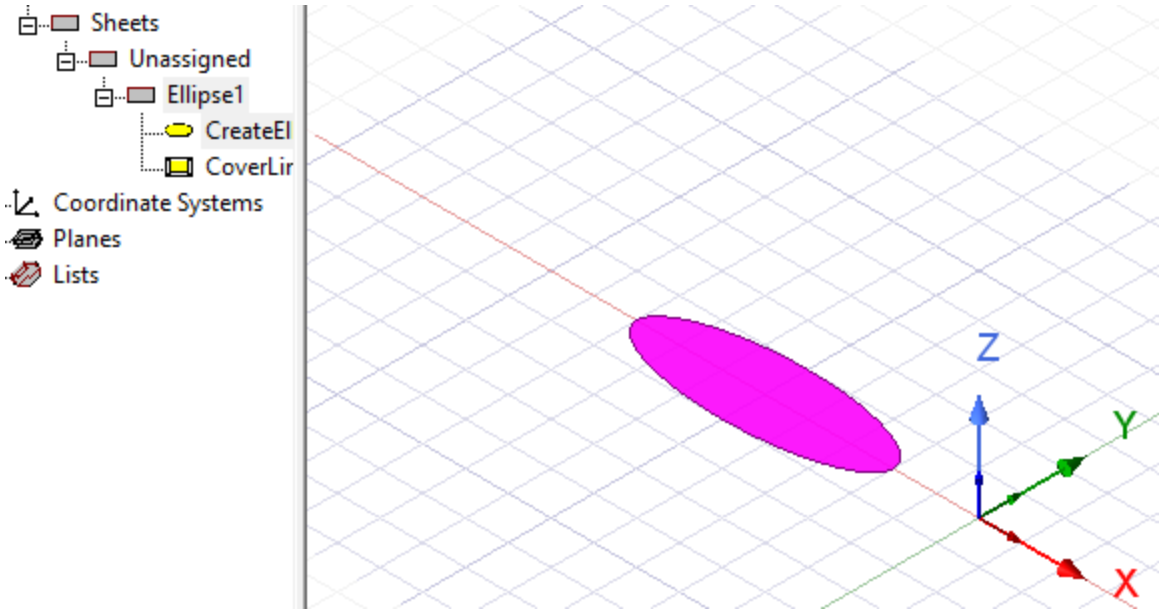
4. Specify the secondary radius of the ellipse. Select the point in one of the following ways:
   - Click the point. Ansys Electronics Desktop constrains mouse movement to a point on the plane orthogonal to the base radius direction.



- Type the coordinates of a point relative to the center point in the **dXm dY,** or **dZ** box.

Ansys Electromagnetics Suite 2026 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.

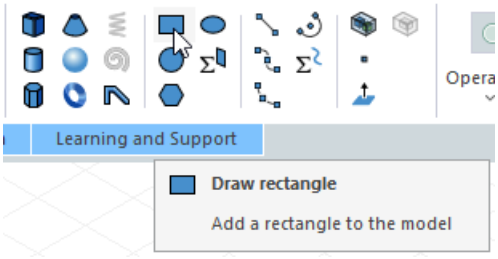
If the base radius is larger than the secondary radius, the ellipse's longer axis will lie along the default base radius direction. If the secondary radius is larger than the base radius, the ellipse's longer axis will lie perpendicular to the default base radius direction. To create an ellipse with an arbitrary orientation, rotate or move the ellipse after drawing it.

Drawing a Rectangle

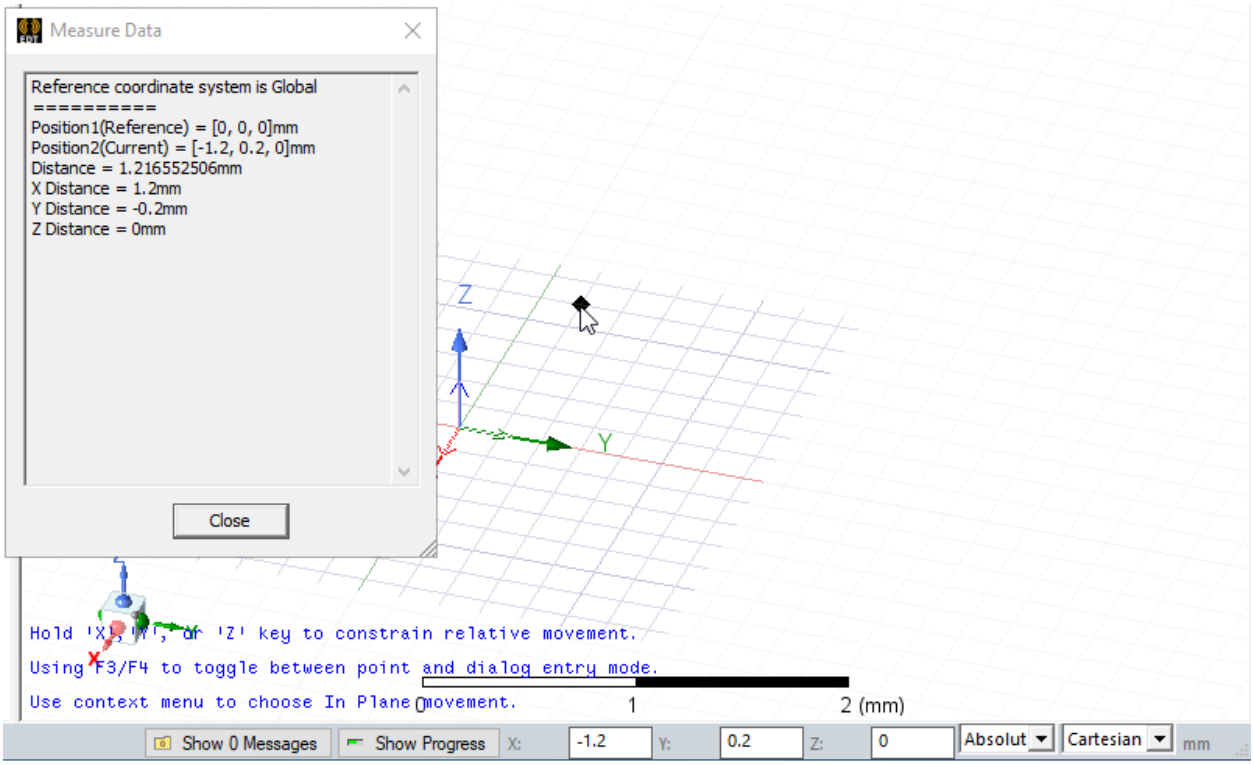
Draw a rectangle (or square) by selecting two diagonally opposite corners. Before you draw a rectangle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Rectangle**, or on the **Draw** ribbon tab, click the **Draw rectangle** icon:

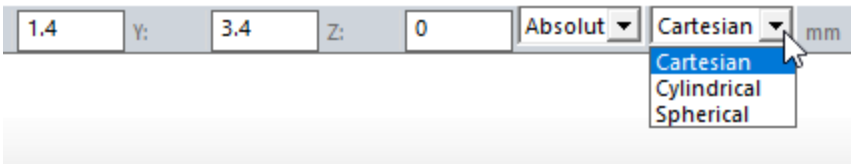
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the first diagonal corner in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.



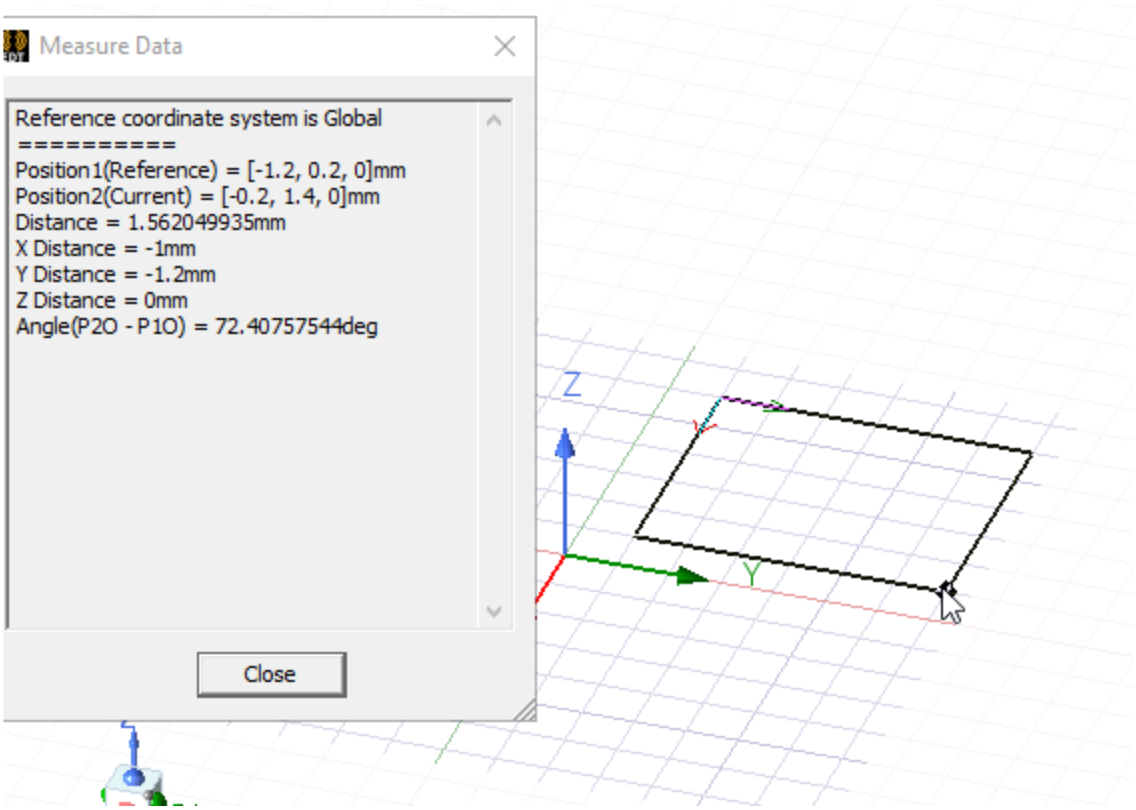
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cylinder in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2026 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 rectangle in one of the following ways:
   - Click the point.

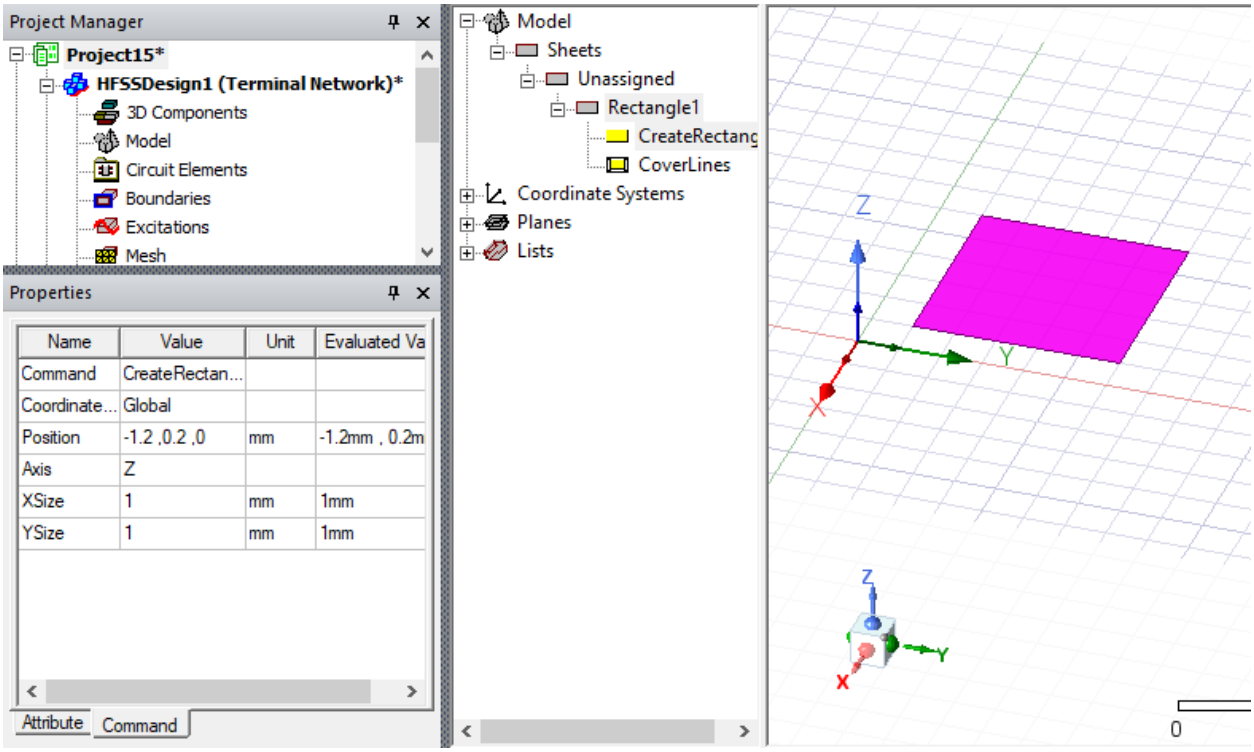


   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   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.

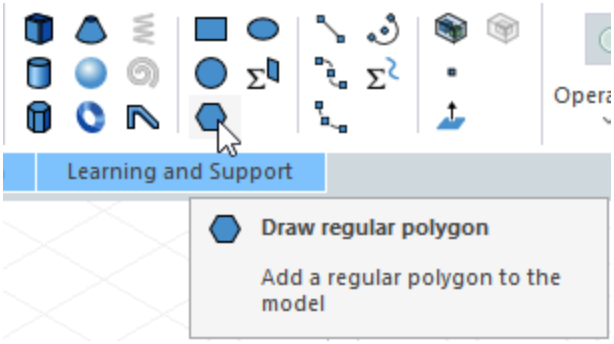
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing a Regular Polygon

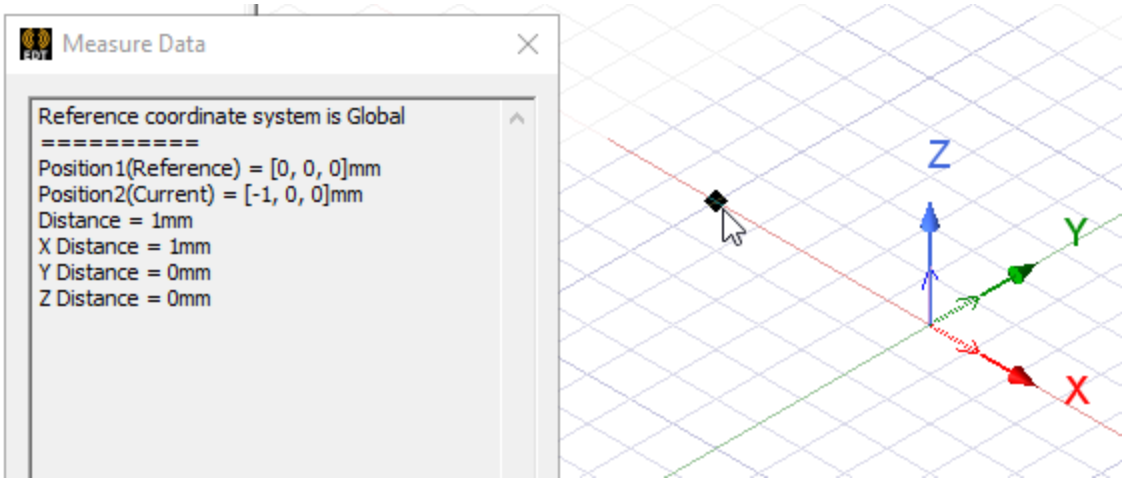
A regular polygon is a 2D object with three or more equal sides. Regular polygons are useful for drawing faceted 2D objects. Before you draw a regular polygon, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **Regular Polygon**, or on the **Draw** ribbon tab, click the **Draw regular polygon** icon:



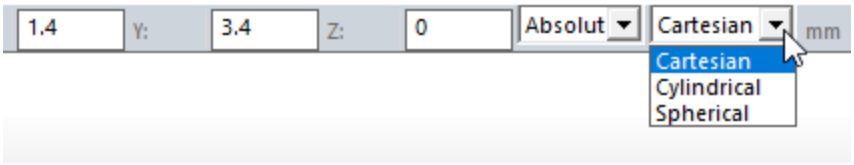
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

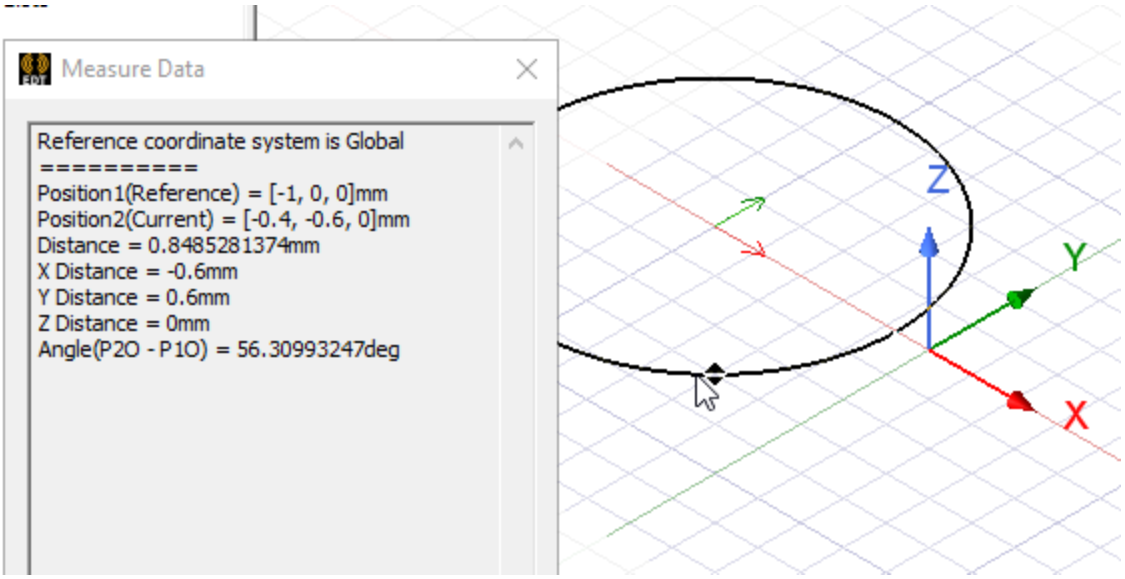
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



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:

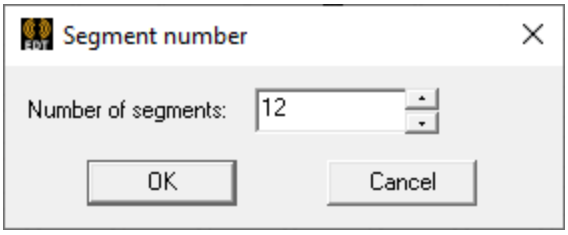
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



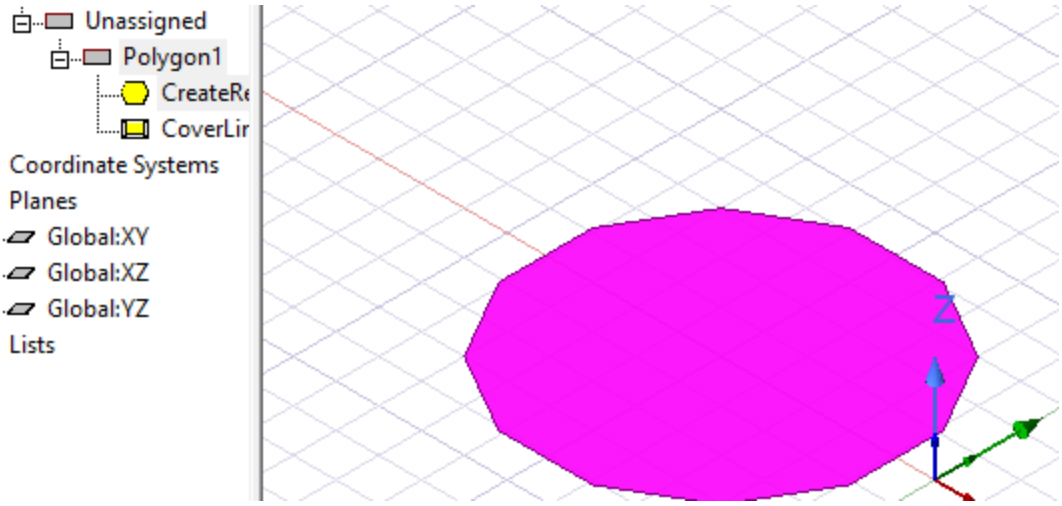
- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. In the **Segment number** dialog box, enter the **Number of segments** in the polygon, and then click **OK**.



Ansys Electromagnetics Suite 2026 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 box, click **OK**.

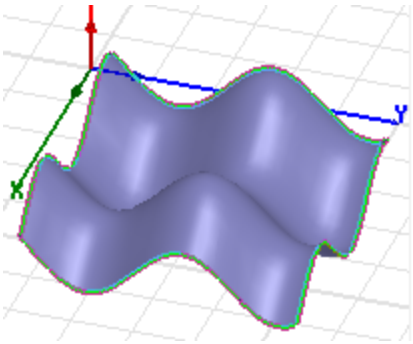
> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

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.

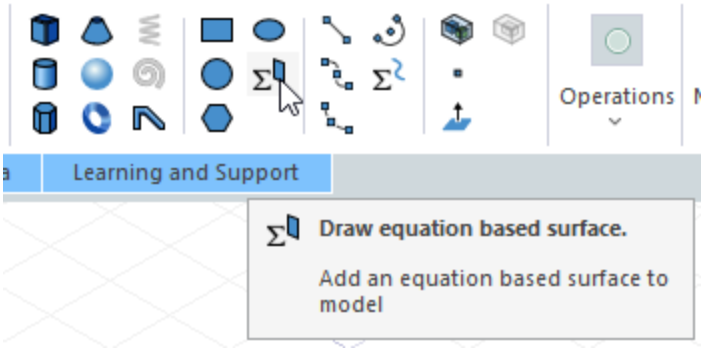
Drawing an Equation-Based Surface

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.

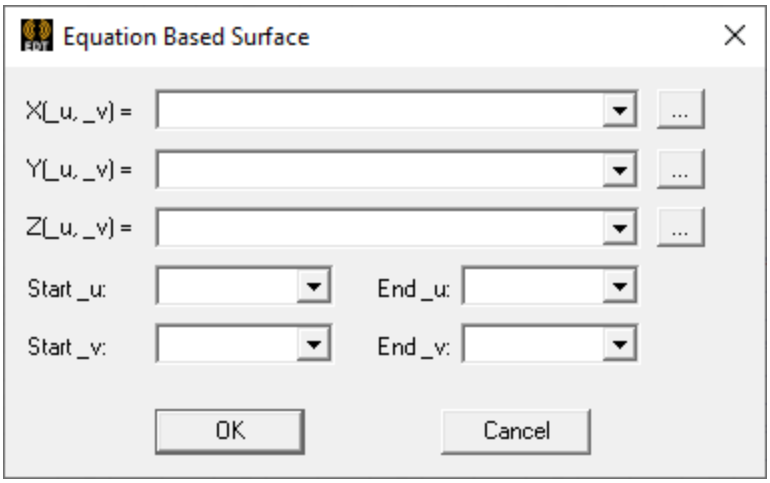
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



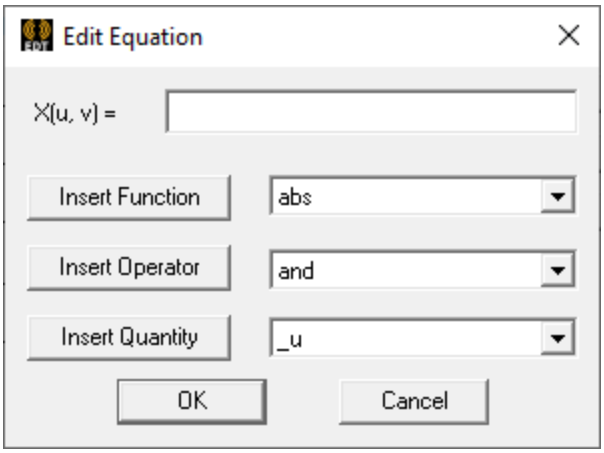
2. Type equations for **X(_u, _v)**, **Y(_u, _v)**, and **Z(_u, _v)**.

   You can also define an equation by doing the following:

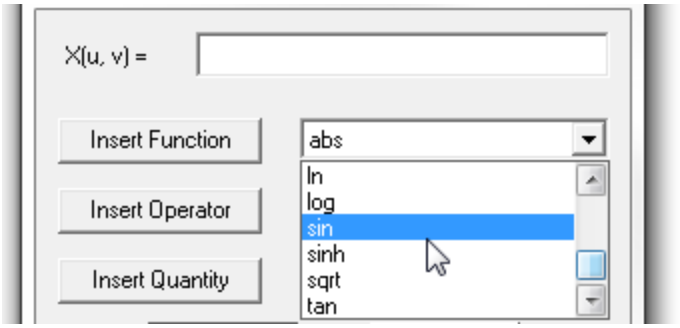
   a. Click the ellipsis button (**...**).

      The **Edit Equation** dialog box appears:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



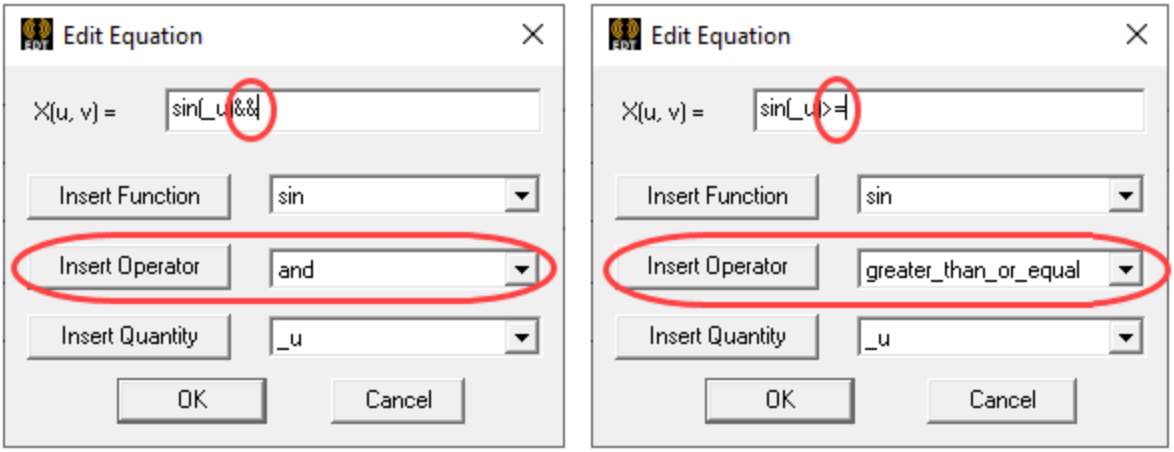
b. Do one or more of the following to define the equation:
   - Type a numerical value, variable, or expression directly in the text box
   - Build an equation for X(u, v), Y(u, v), or Z(u, v), by selecting the function, operator, and quantity to insert into the equation from the three drop-down menus:



     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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



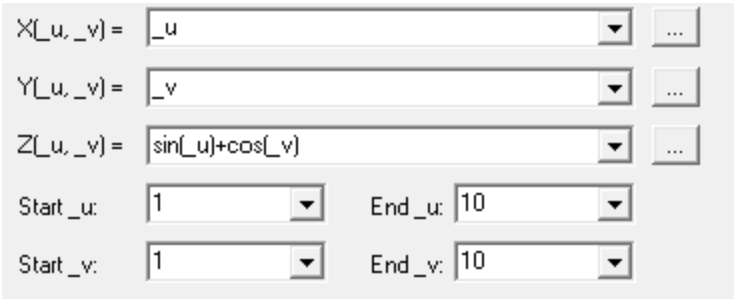
- You can combine the preceding two methods to build complex equations.



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.

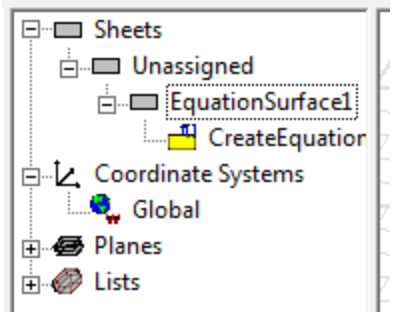
   Select end values for **End_u** and **End_v** fields.



4. Click **OK** on the **Properties** dialog box.

Ansys Electromagnetics Suite 2026 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.

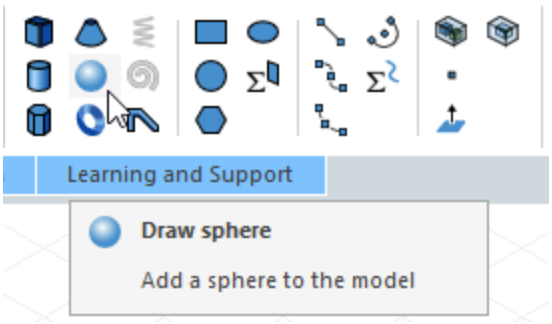


If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes.

Drawing a Sphere

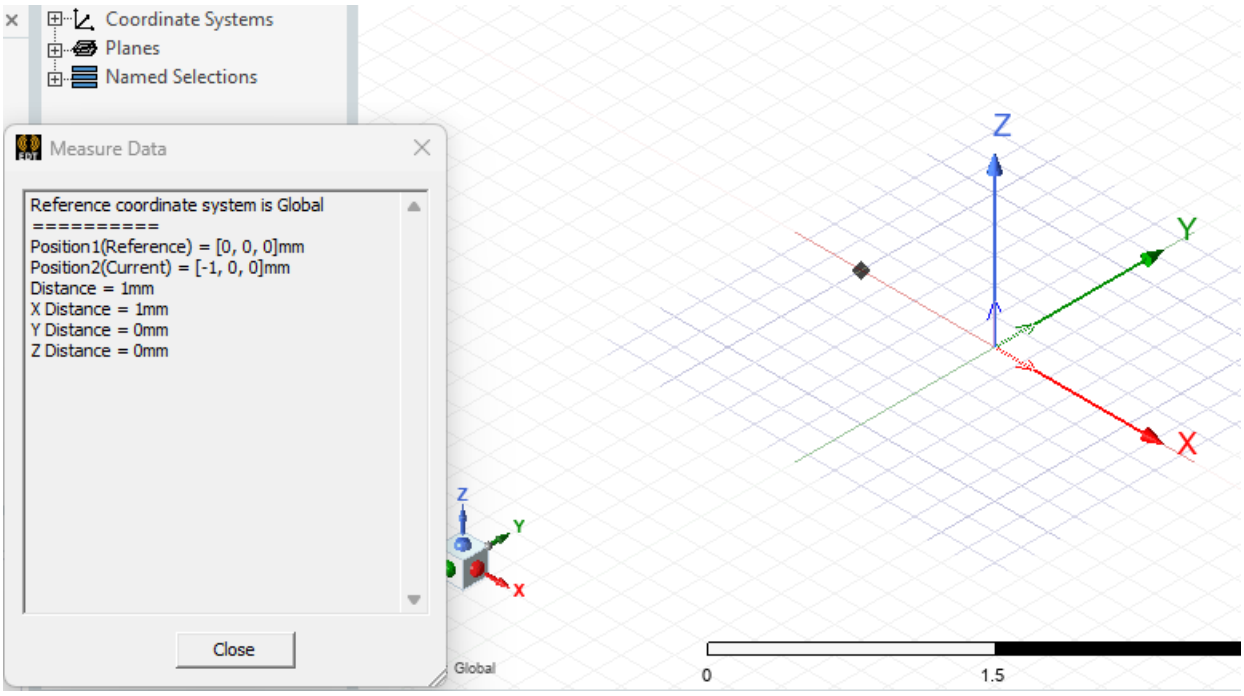
Draw a sphere, a 3D circle, by selecting a center point and a radius. Spheres are drawn as true surfaces in the modeler. Before you draw a sphere, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Sphere**, or on the **Draw** ribbon tab, click the **Draw sphere** icon:

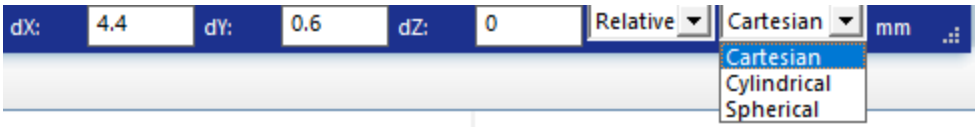


2. Select the center point of the sphere in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



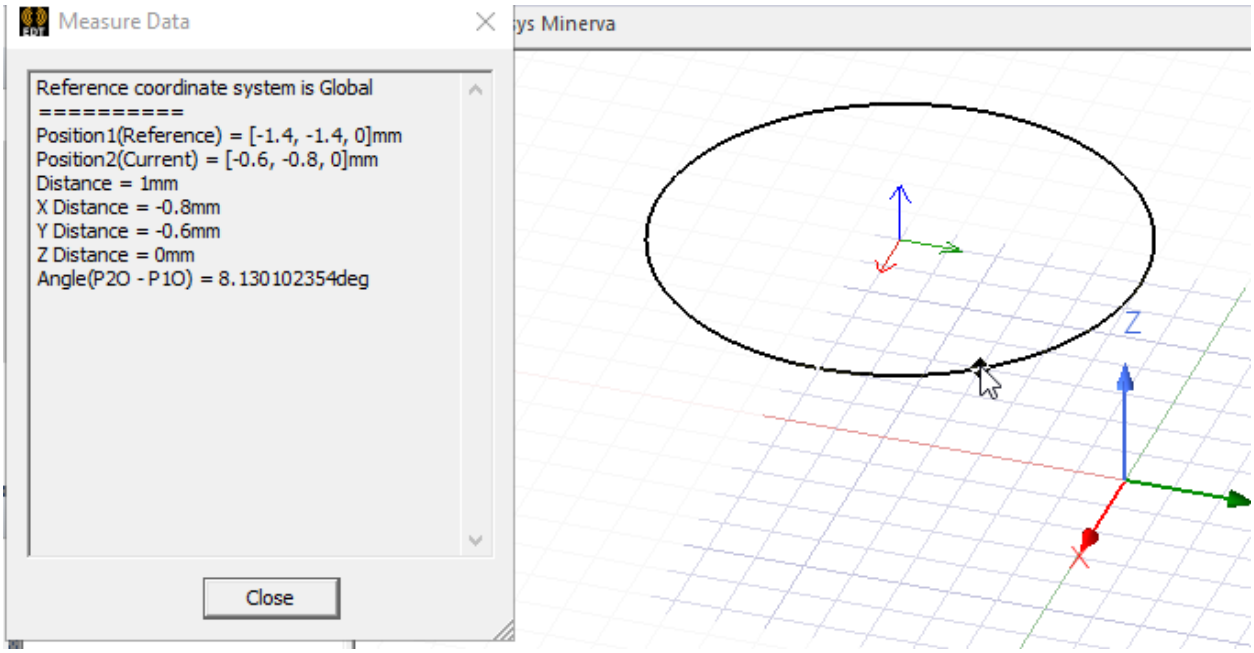
- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the radius by selecting a point on the sphere's circumference in one of the following ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

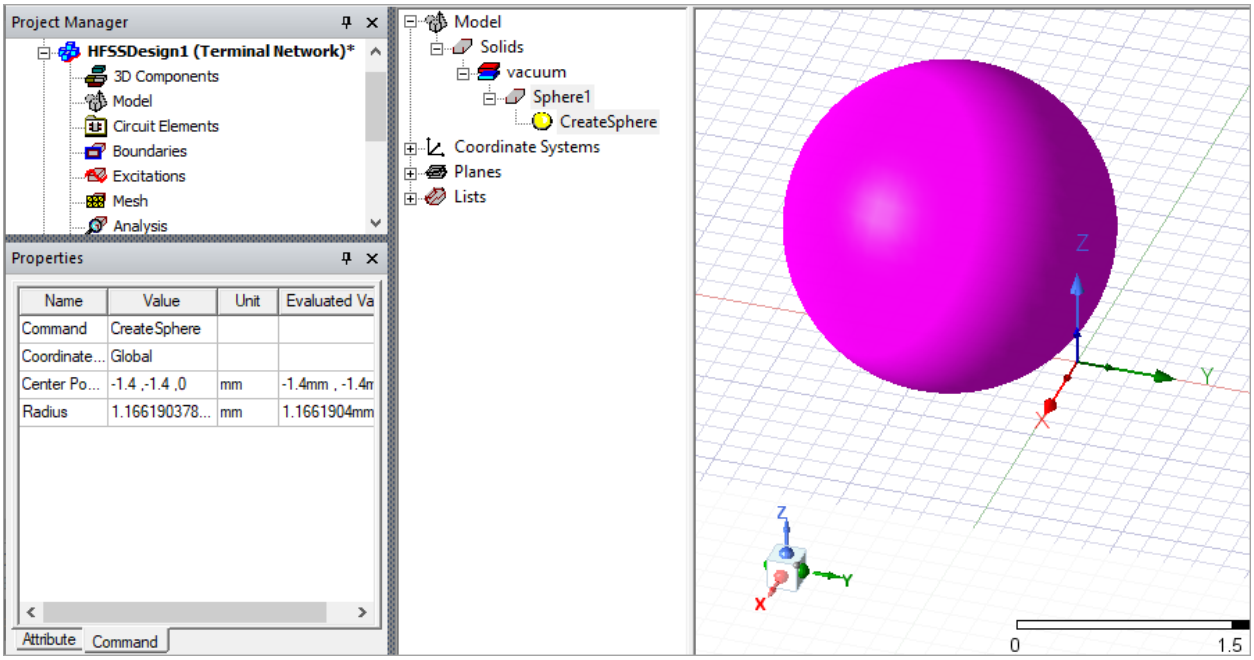
- Click the point



- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK**.

Related Topics

Modifying Objects

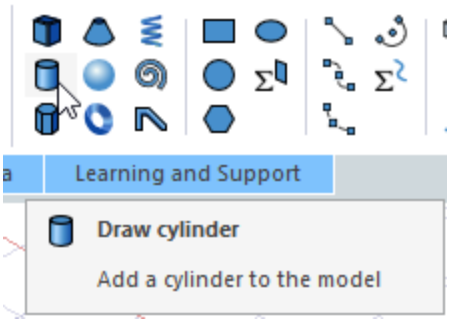
Modifying Surface Approximation Settings

Drawing a Cylinder

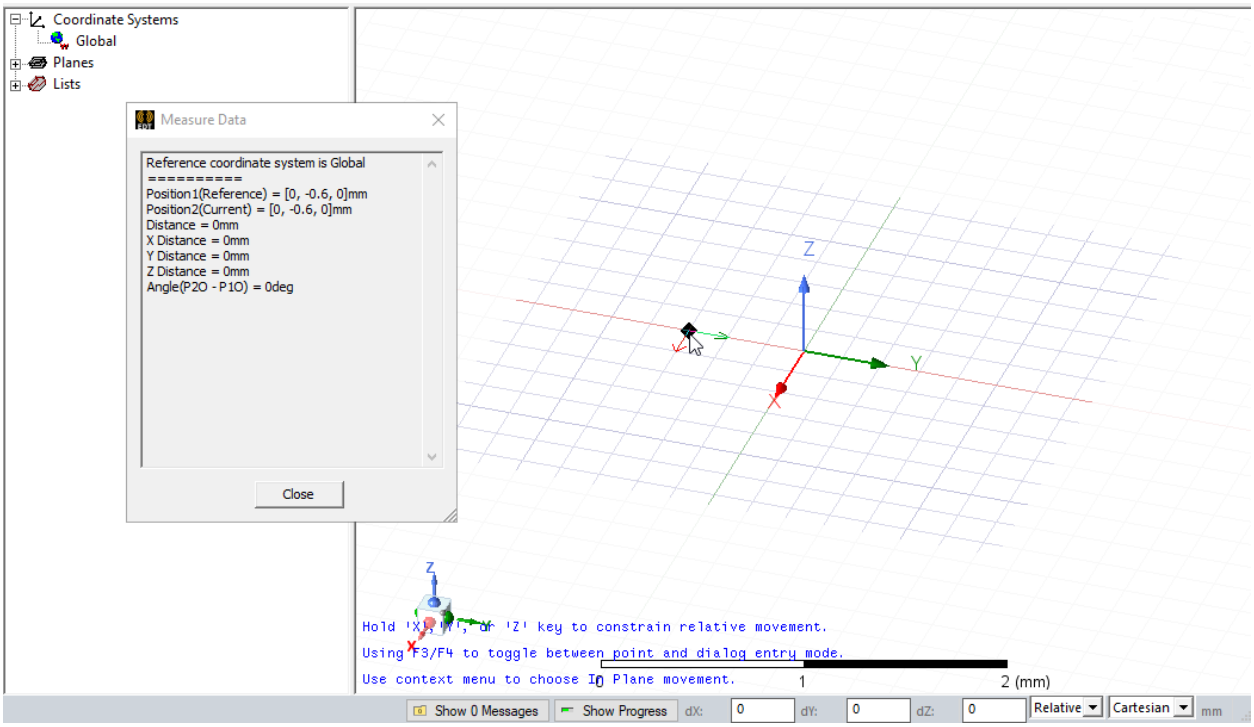
Draw a cylinder by selecting a center point, radius, and height. Cylinders are drawn as true surfaces in the modeler. Before you draw a cylinder, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Cylinder**, or on the **Draw** ribbon tab, click the **Draw cylinder** icon:

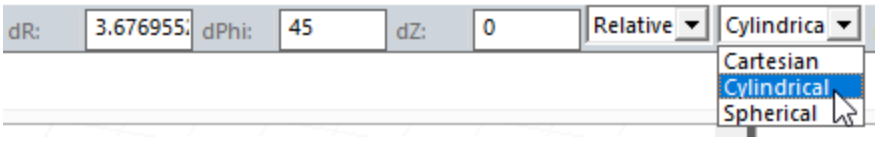
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the center point of the cylinder's base circle in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.

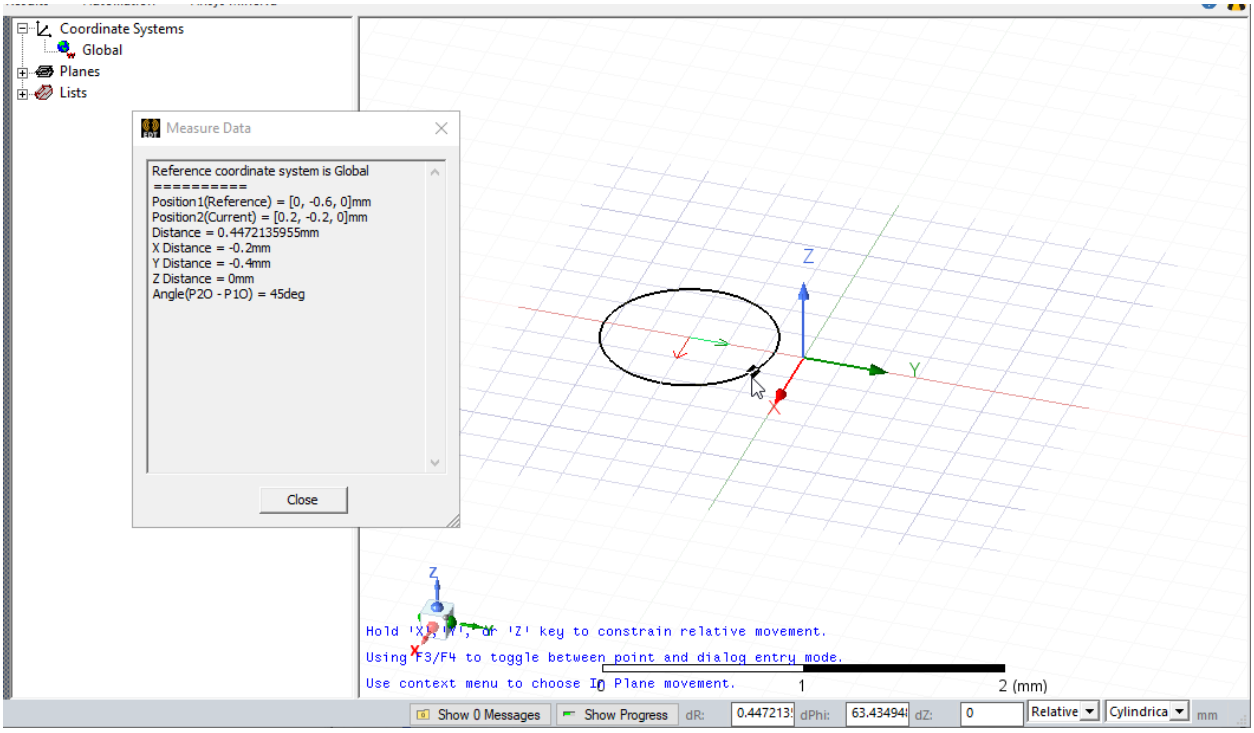


   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cylinder in Cartesian, Cylindrical, or Spherical coordinates.



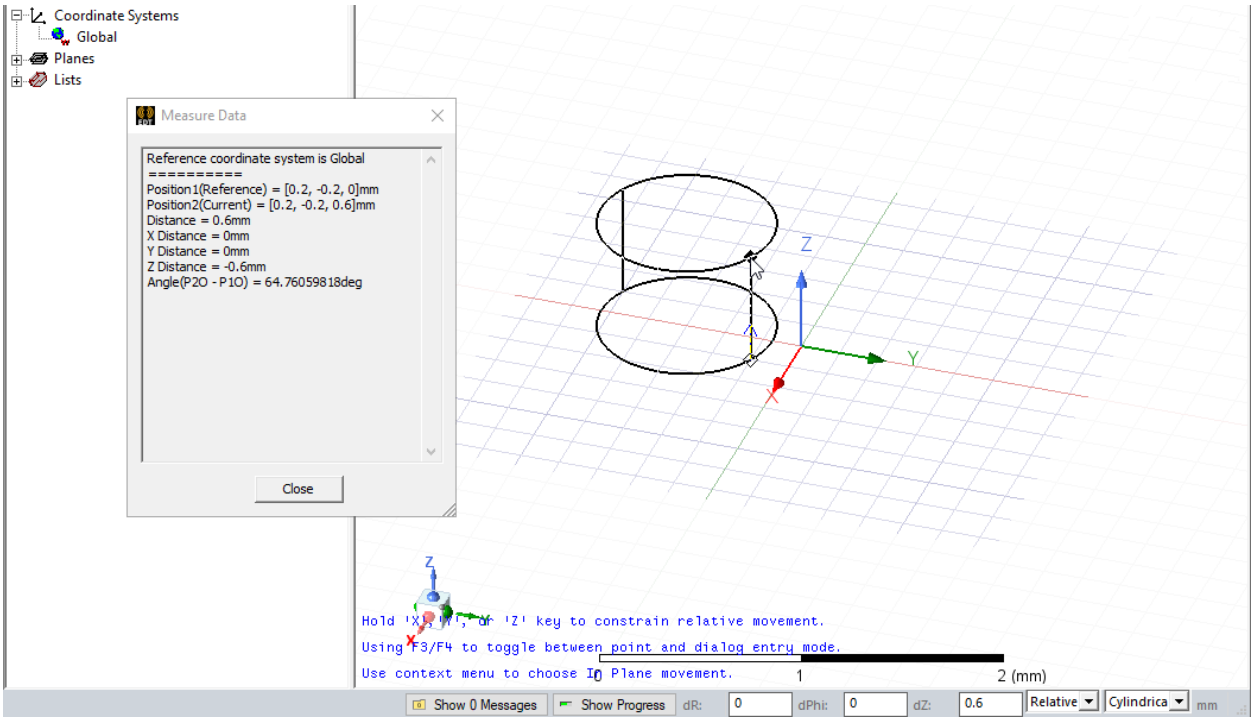
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



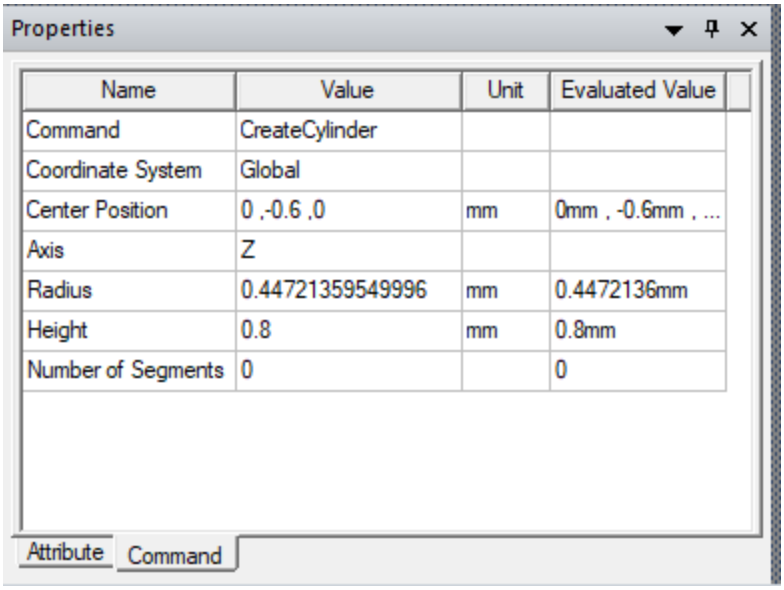
Note:

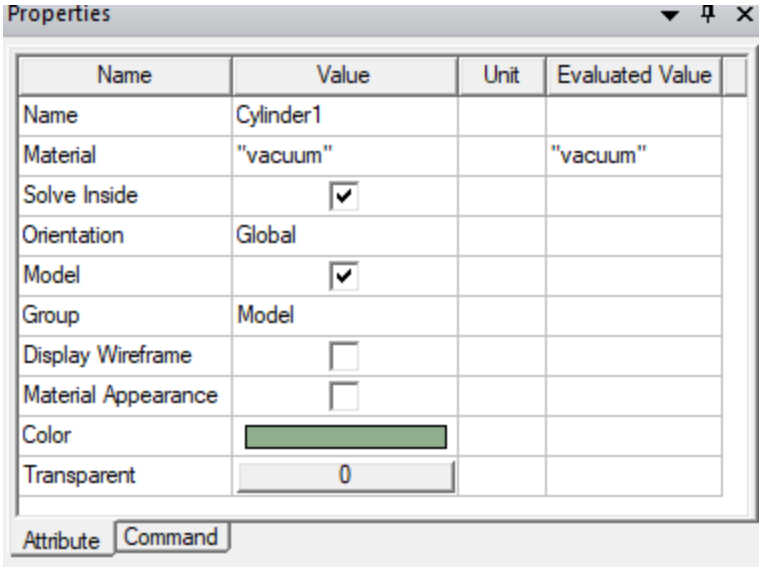
If you create a cylinder with a height of zero, Ansys Electronics Desktop draws a circular sheet object.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

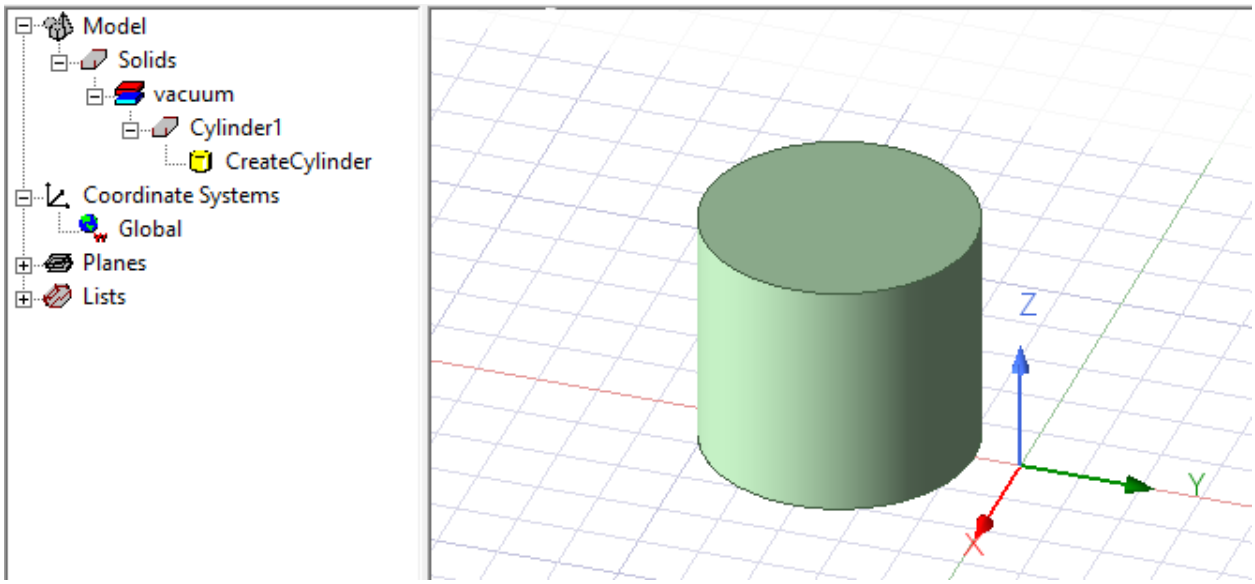
the docked Properties window.





5. If you used the Properties window, click **OK**. The cylinder appears in the Modeler window and is listed in the History tree.

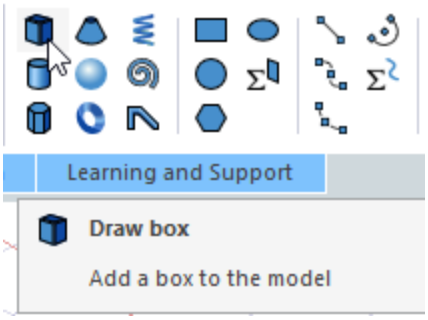
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing a Box

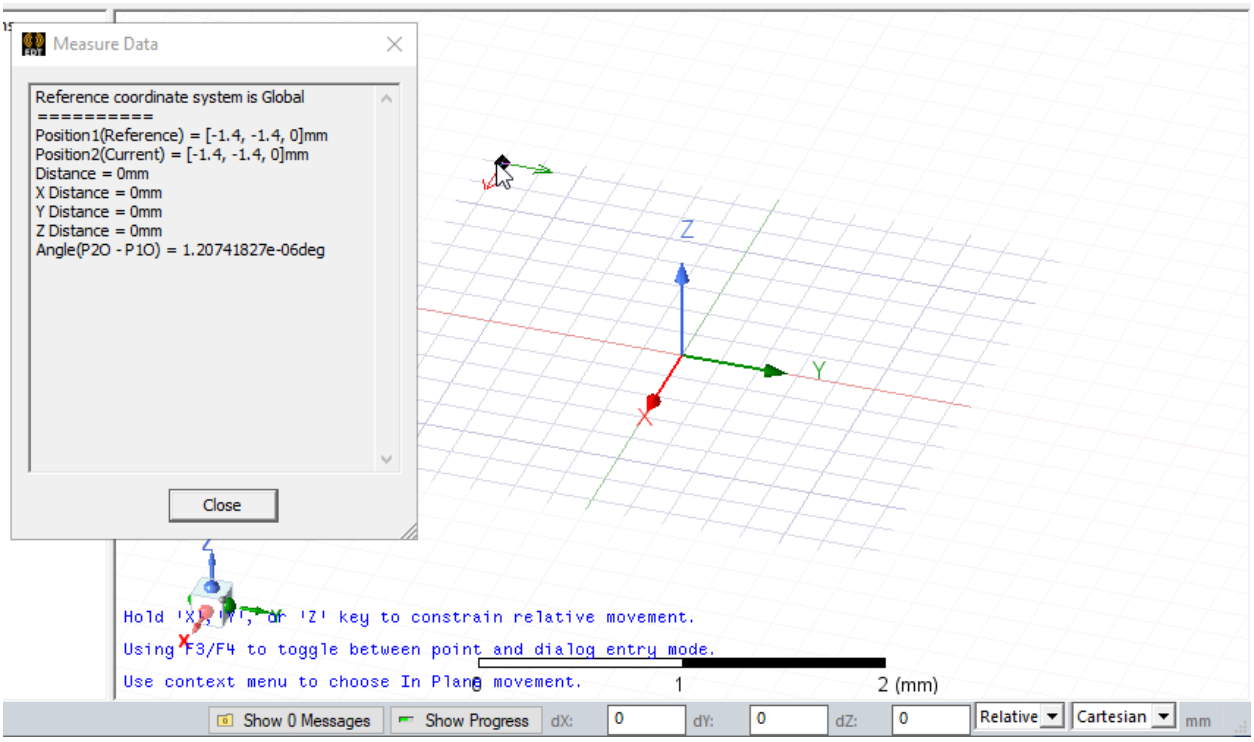
Draw a box by selecting two diagonally opposite corners of the base rectangle, then specifying the height. Before you draw a box, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Box**, or on the **Draw** tab of the ribbon, click the **Draw box** icon:

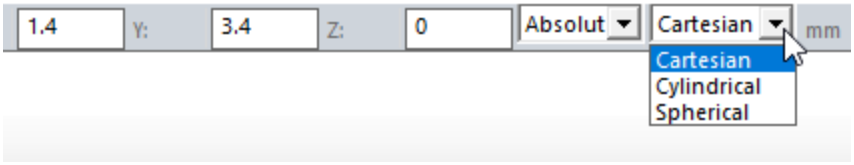


2. Select the first diagonal corner of the base rectangle in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



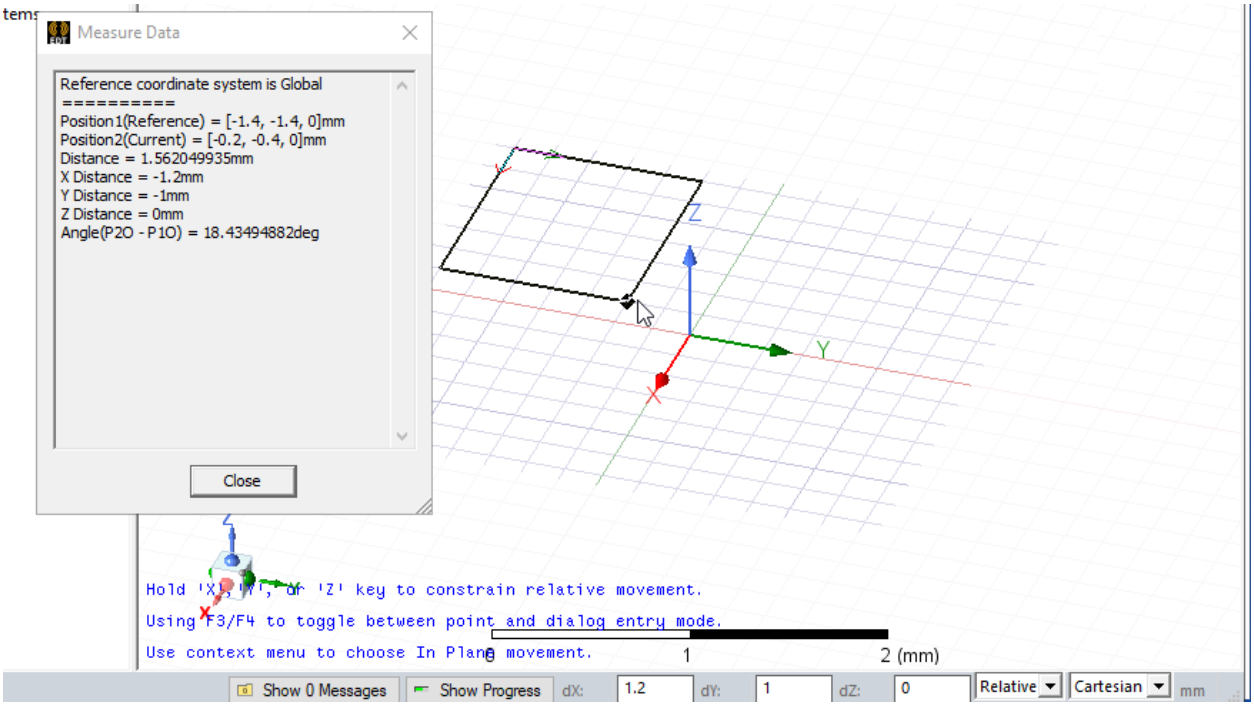
- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the box in Cartesian, Cylindrical, or Spherical coordinates.



To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

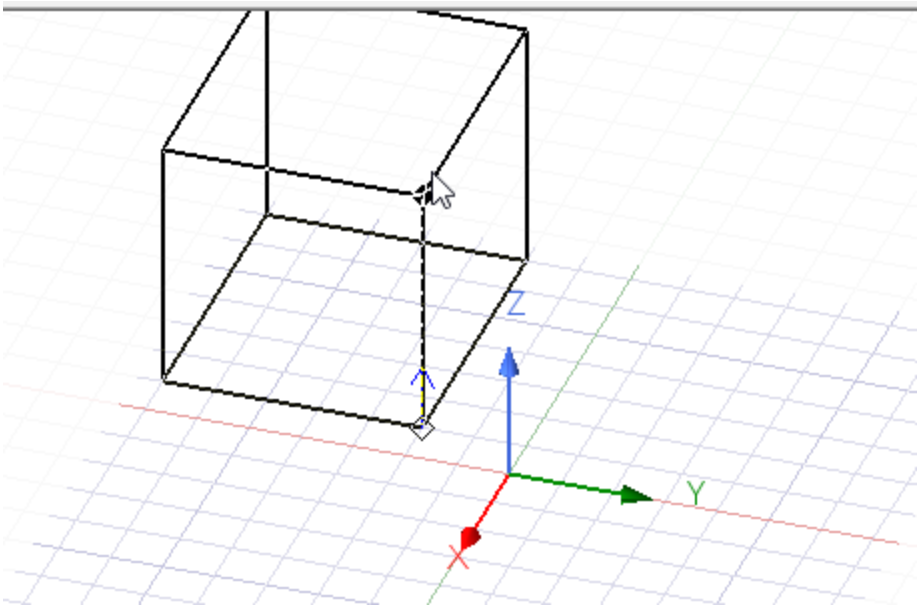
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the second corner of the 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.

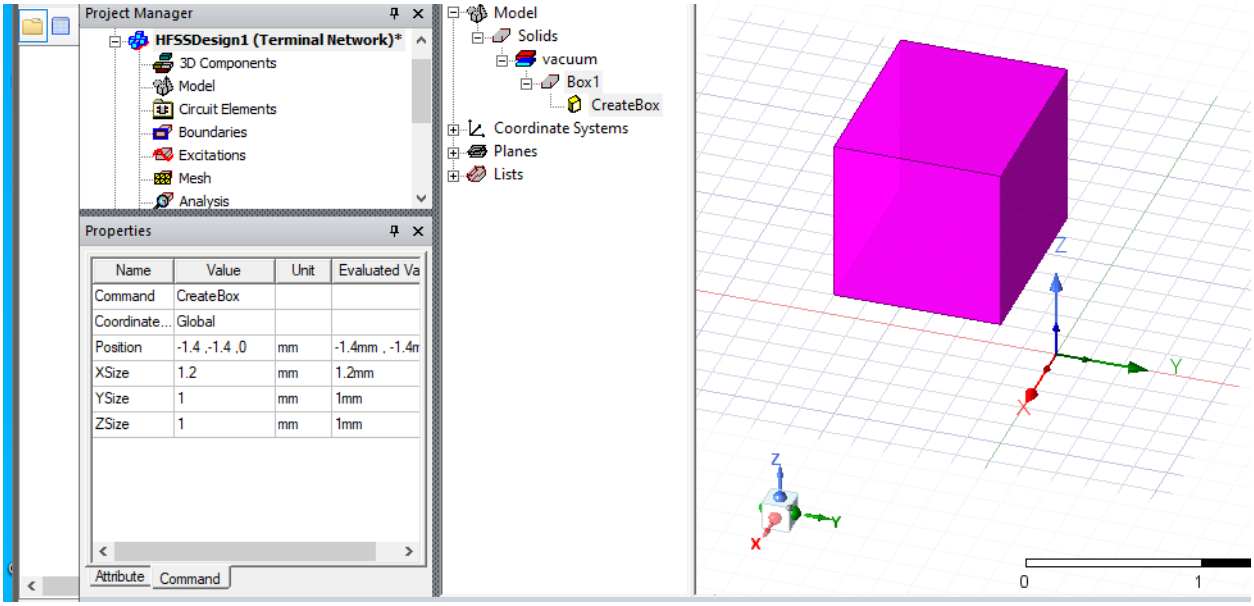
4. Specify the height of the box by selecting a point on the axis perpendicular to the base rectangle. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.



Ansys Electromagnetics Suite 2026 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.

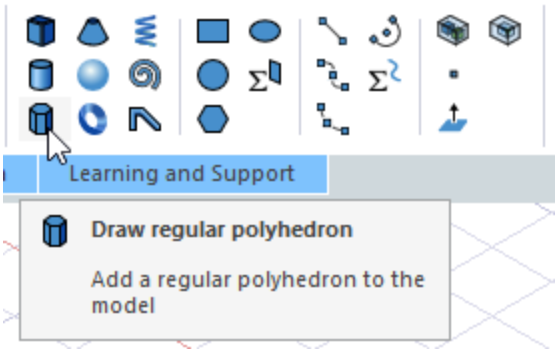
   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.



Drawing a Regular Polyhedron

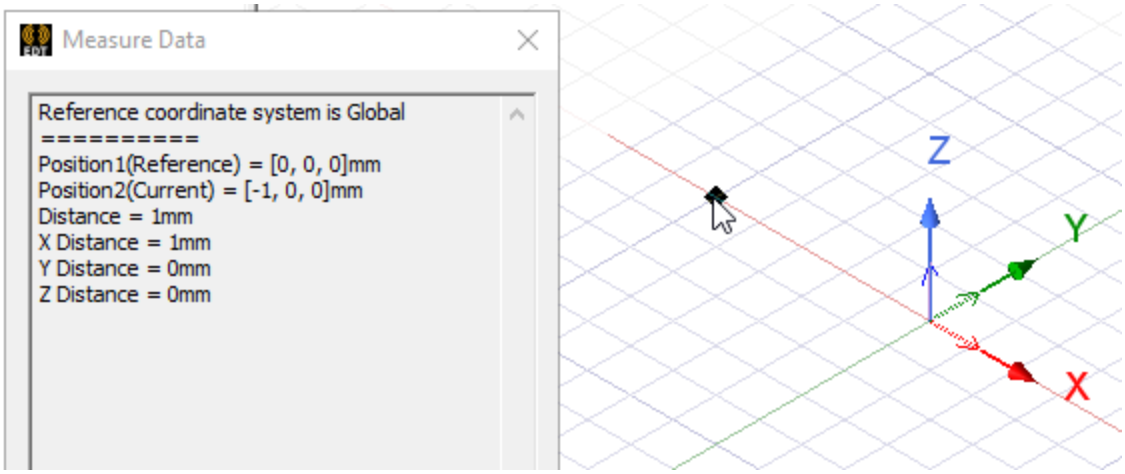
In the modeler, regular polyhedrons are 3D objects with regular polygon faces; each face has three or more equal sides. Regular polyhedrons are useful for drawing faceted 3D objects. Before you draw a regular polyhedron, you can specify the coordinate system, and you can set the drawing plane as Z, Y, 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:

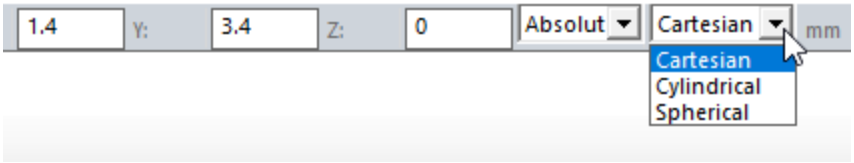


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

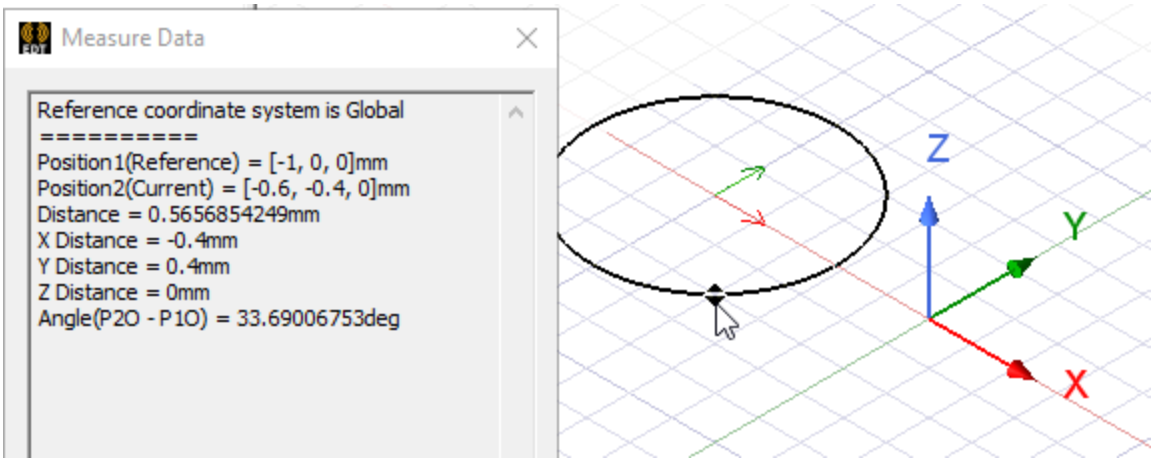
2. Select the center point of the polyhedron in one of the following ways:
   - Click the point.



   - 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.



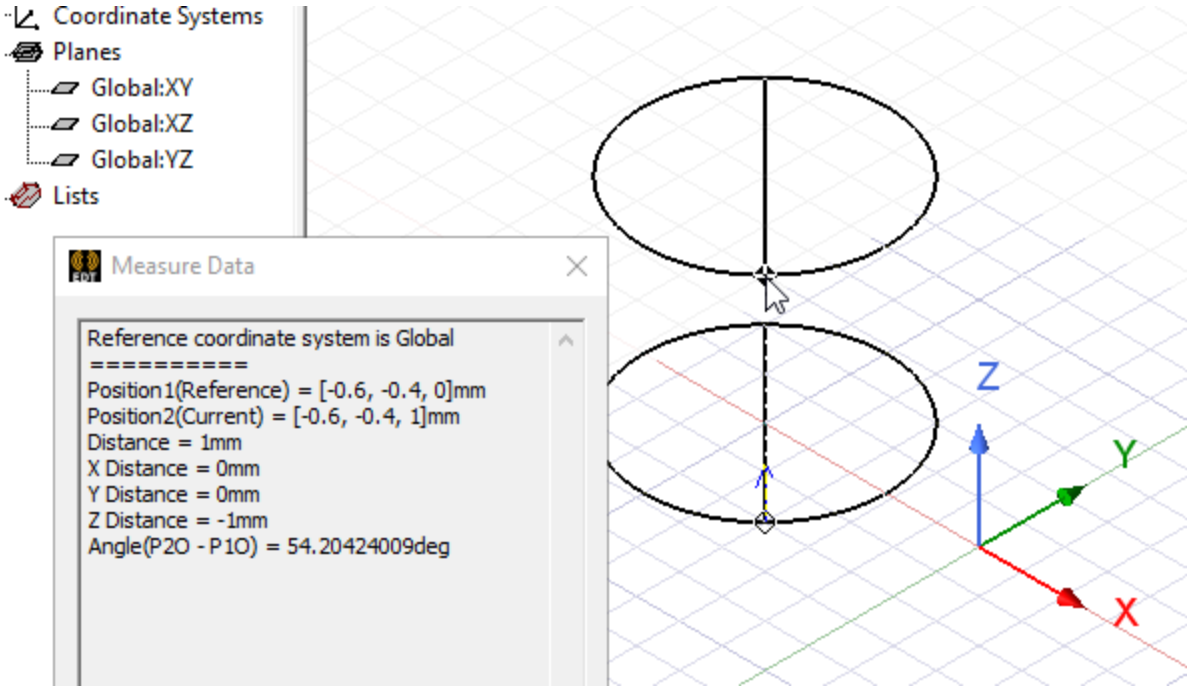
3. Select the radius of the polyhedron, the distance from the center point to one of the polyhedron's vertices, in one of the following ways:
   - Click the point.



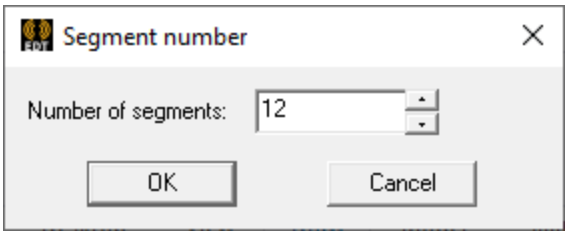
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

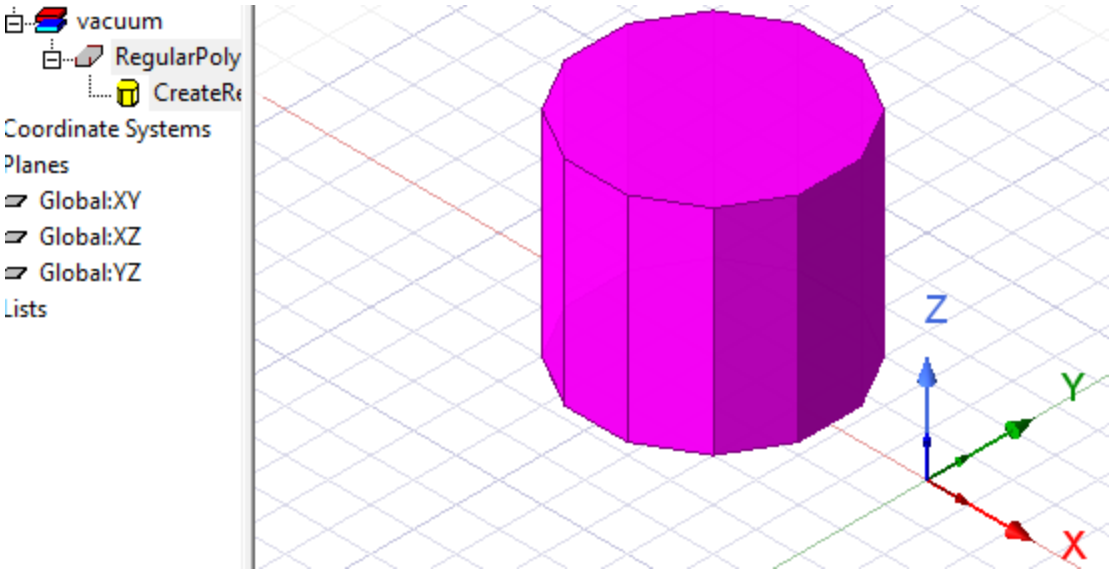


5. In the **Segment number** dialog box, enter the **Number of segments** in the polyhedron, and then click **OK**.



Ansys Electromagnetics Suite 2026 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.

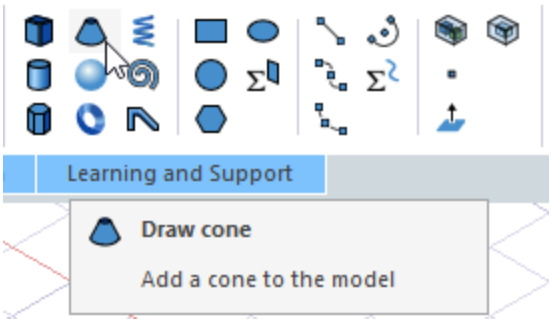
If the **Automatically cover closed polyline** option is selected in the **Modeler Options** dialog box, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Drawing a Cone

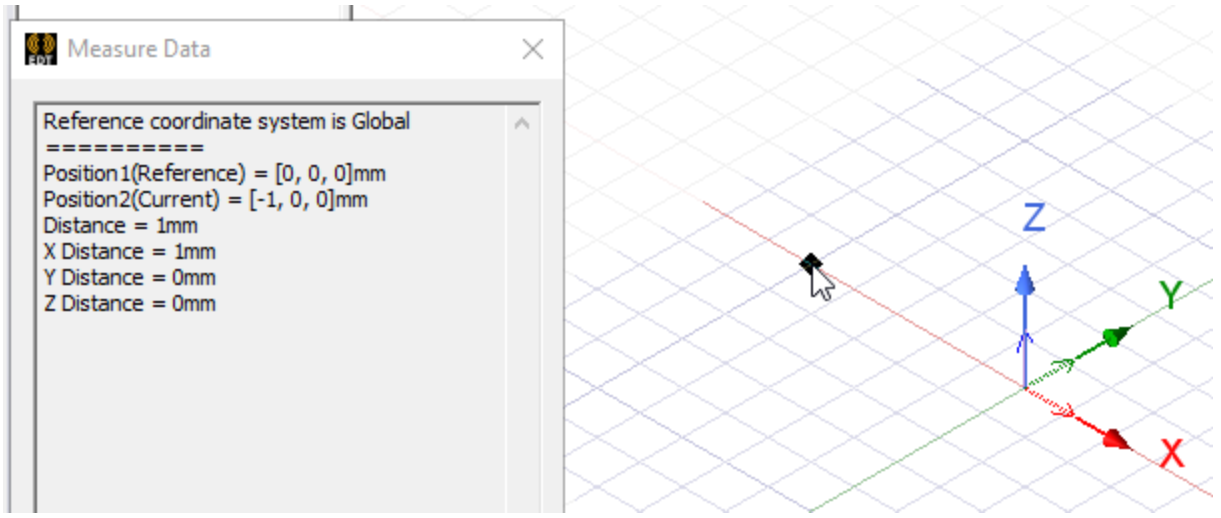
Draw a cone by selecting the center point and radius of the cone's base circle, then specifying the radius of the cone's top circle and the cone's height. Cones are drawn as true surfaces in the modeler. Before you draw a cone, you can specify the coordinate system, and you can set the drawing plane 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:

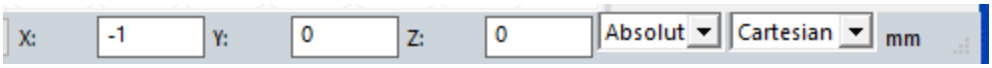
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



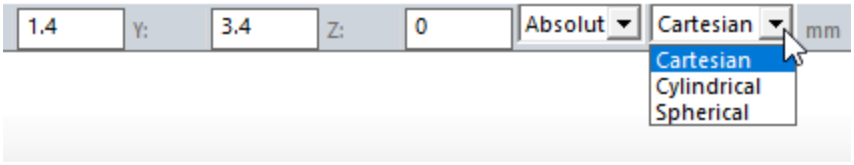
2. Select the center point of the cone's base circle in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.



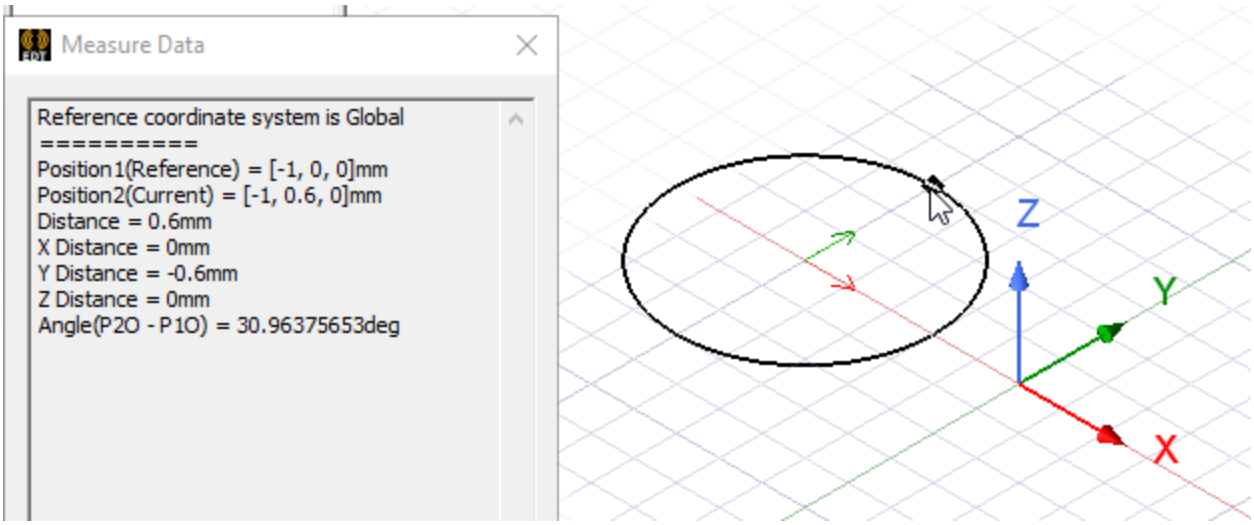
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cone in Cartesian, Cylindrical, or Spherical coordinates.



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:

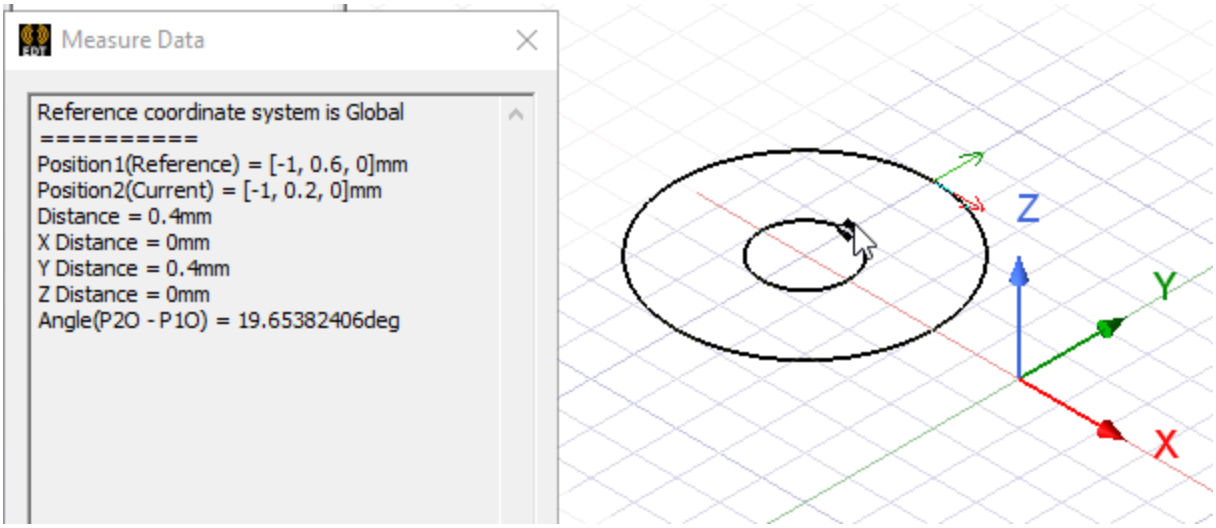
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



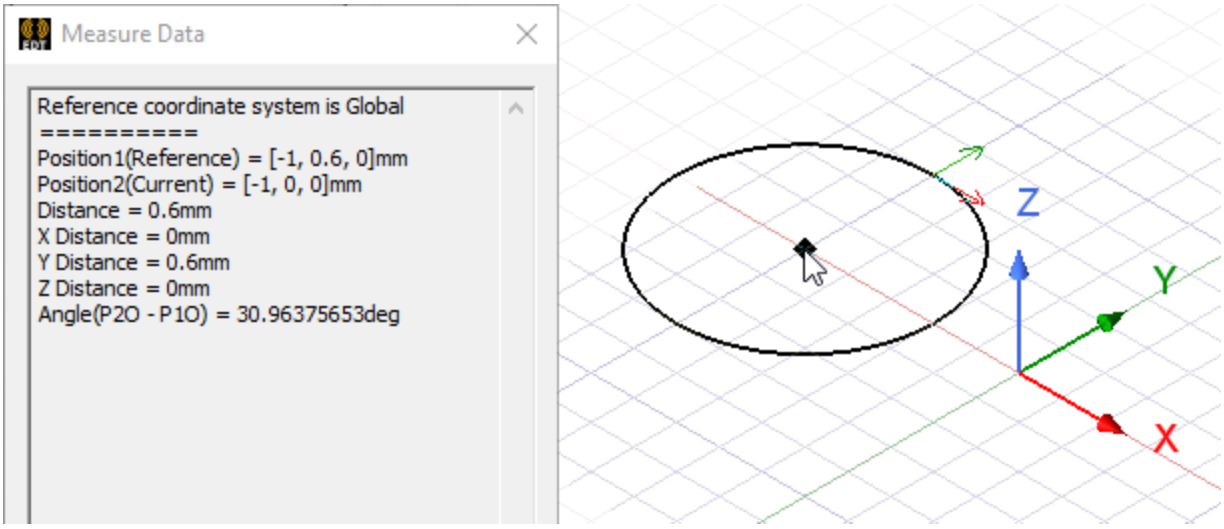
- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. 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.

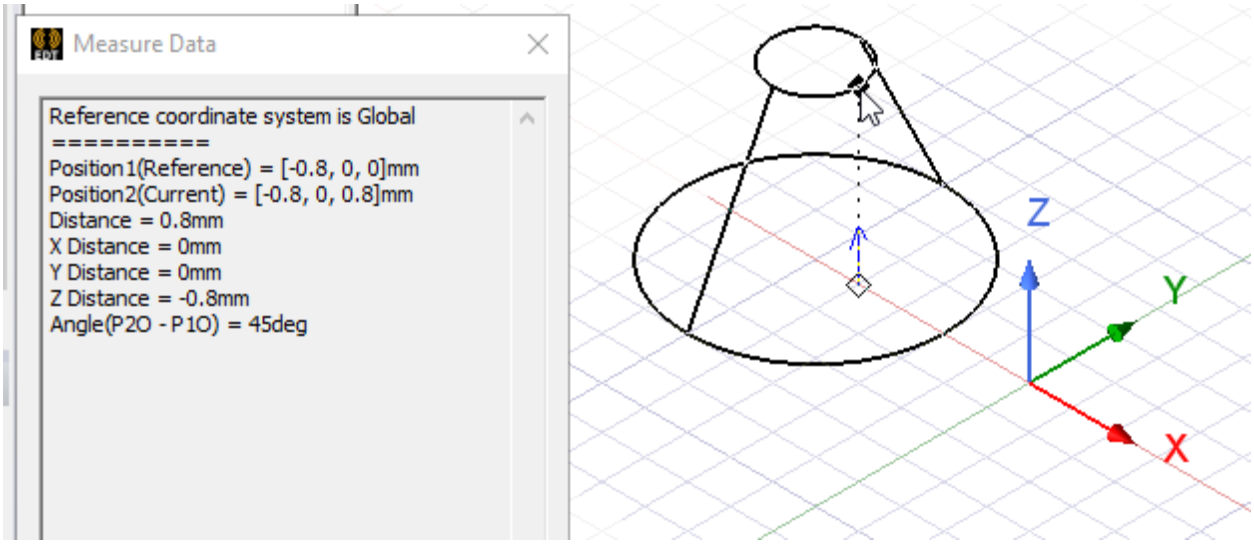


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create an apex, select the same center point as the cone's base circle.



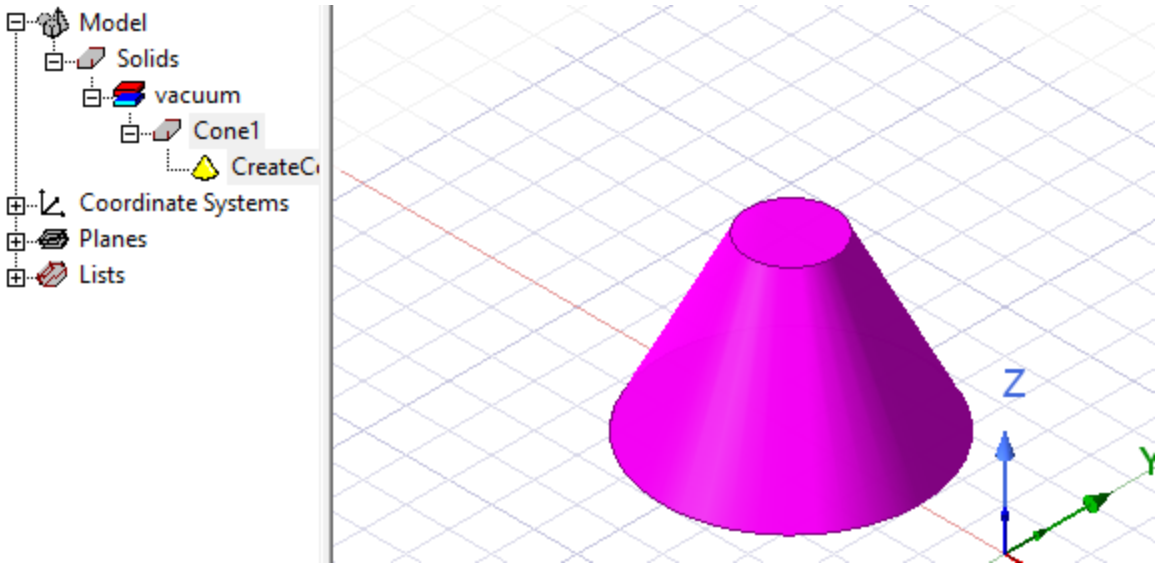
5. Specify the height of the cone by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.



6. Double click, or right click and select **Done** from the 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. The cone appears in the Modeler window

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

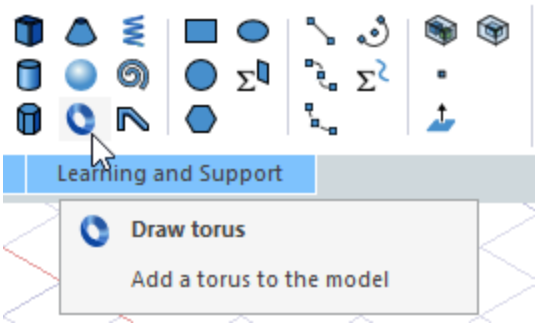
and in the History Tree.



Drawing a Torus

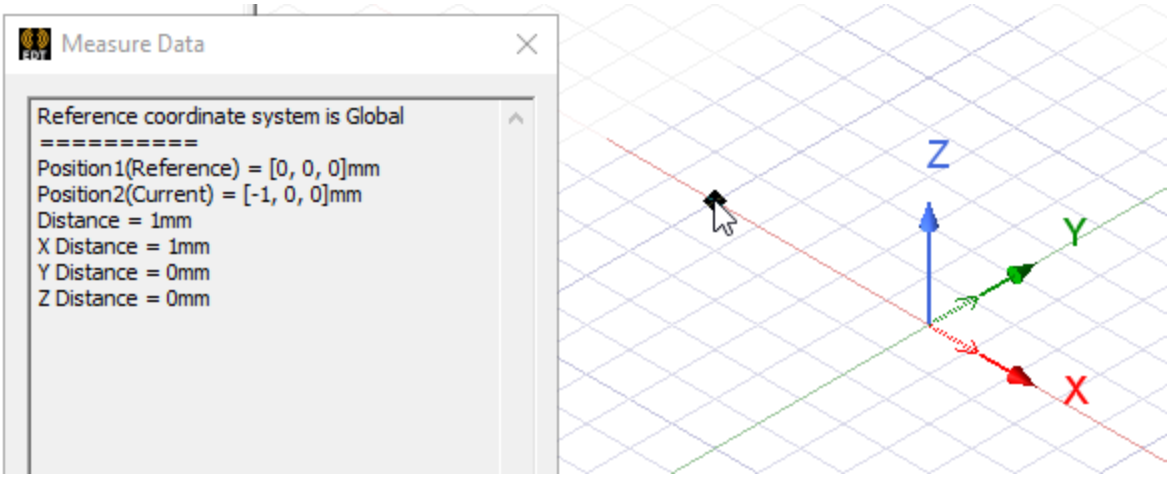
Draw a torus by selecting its center point, major radius, and minor radius. The modeler then sweeps a circle around a circular path. A torus is drawn as true surfaces in the modeler. Before you draw a torus, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

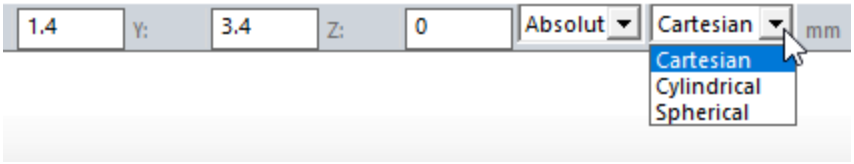
2. Select the center point of the torus in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

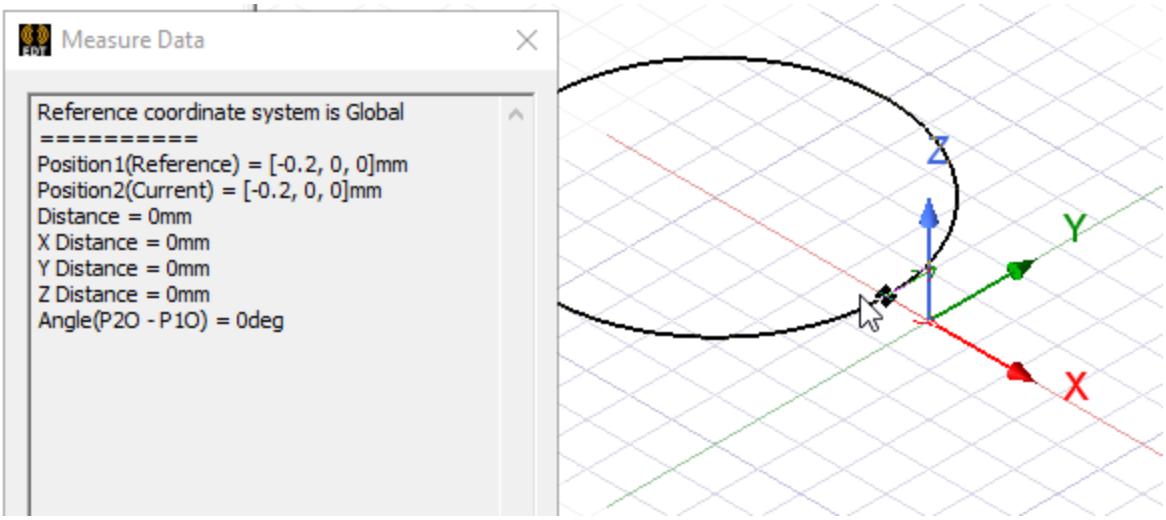


     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the torus in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2026 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.

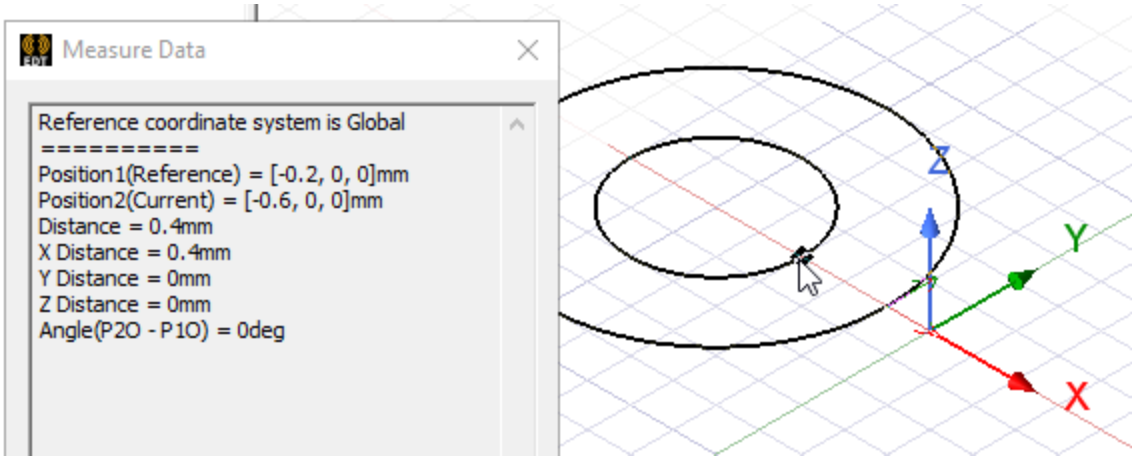


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The major radius determines the diameter of the torus.

4. Specify the minor radius by selecting a point relative to the major radius point.

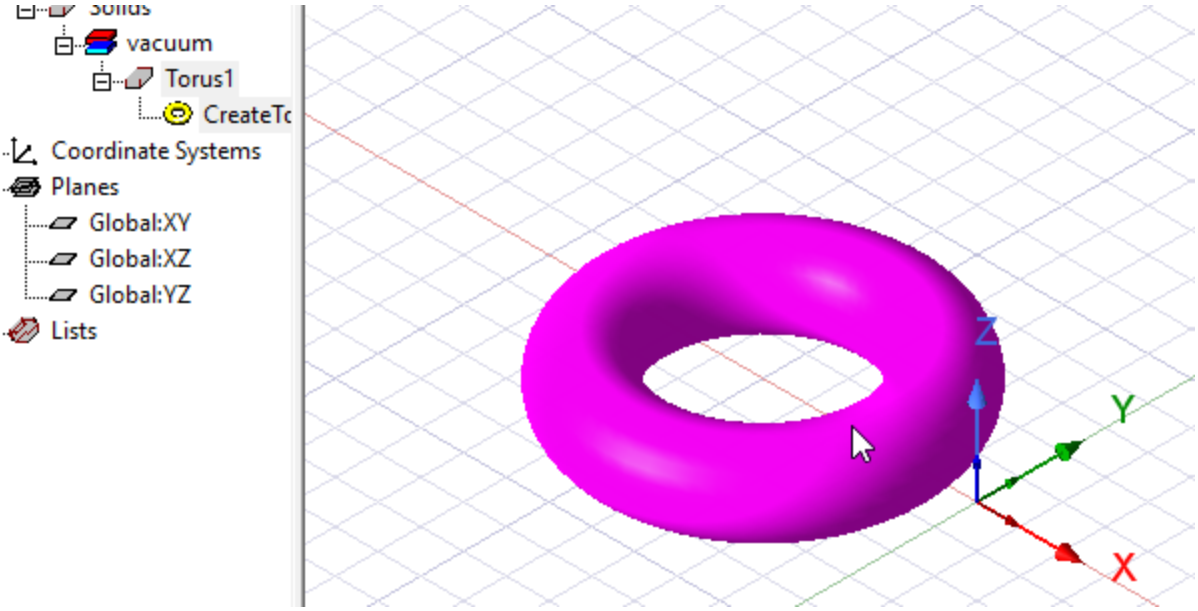
   The minor radius determines the diameter of the "donut hole".



   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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

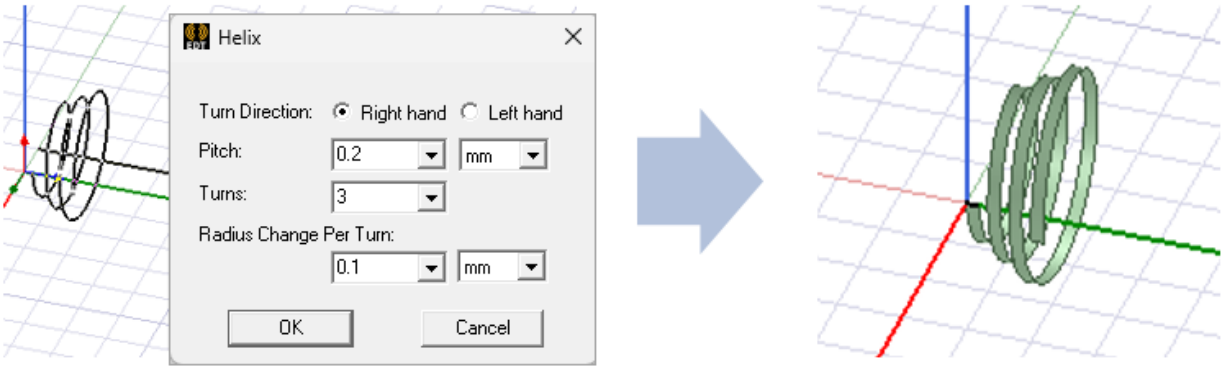
5. Double-click or right-click to open the shortcut menu and click **Done**. The torus appears in the Modeler window and in the History Tree.



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**.

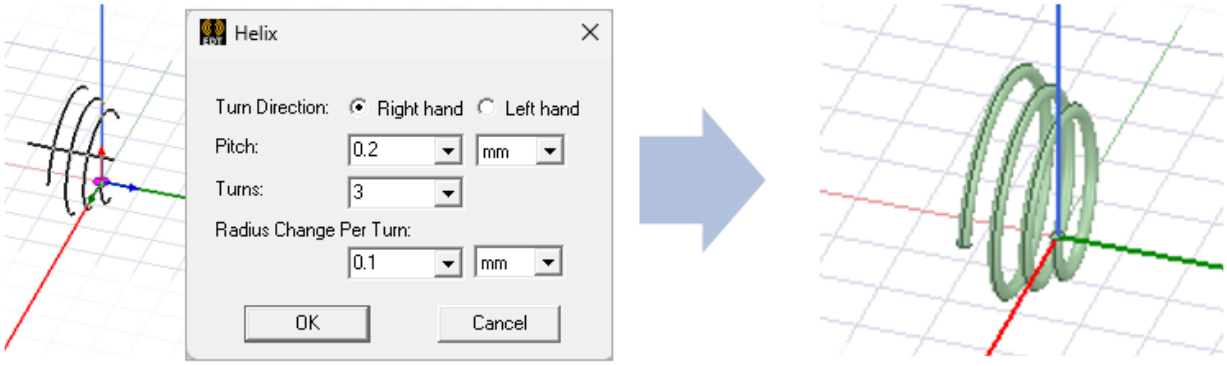
Drawing a Helix

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 create the vector, the **Helix** dialog box opens, where you then specify the turn direction, pitch, number of turns, and radius of the turn to define the helix.

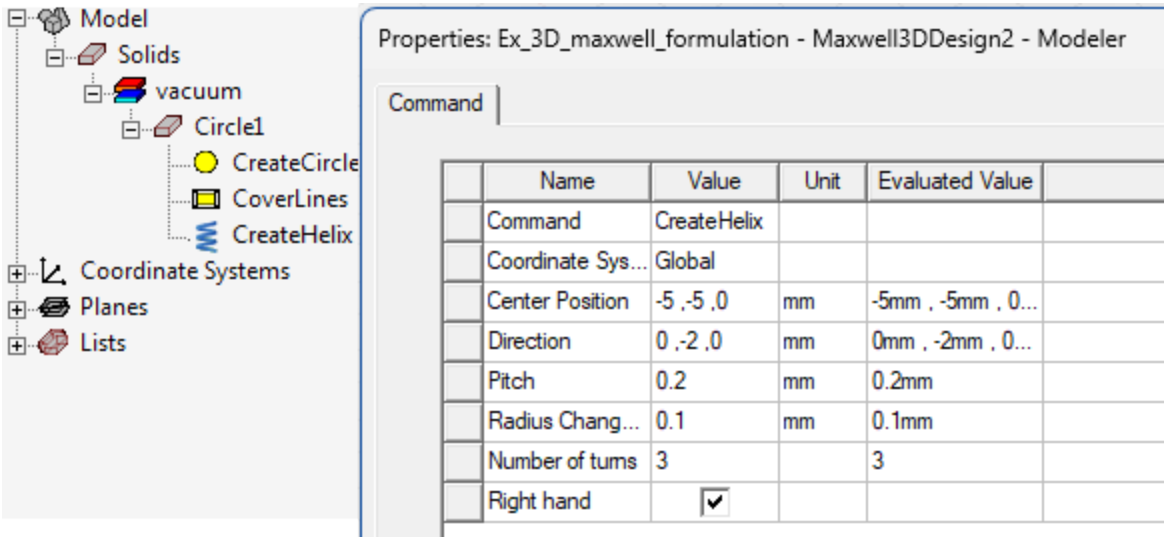


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sweeping a 2D object results in a hollow 3D object. Sweeping a 2D sheet object results in a 3D solid object.



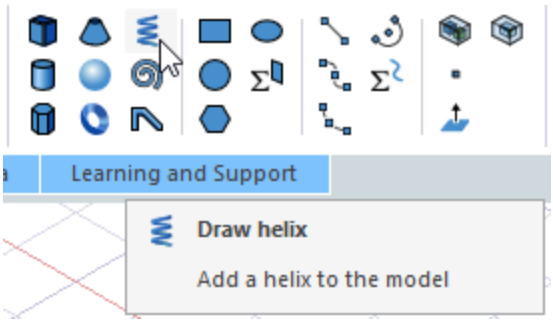
Once you have created a helix, you can changing it by edit the properties of the original object, and by editing the helix command properties for coordinate system, center position (which corresponds to the vector placement), direction, pitch, turns, radius change, and direction.



To create a helix:

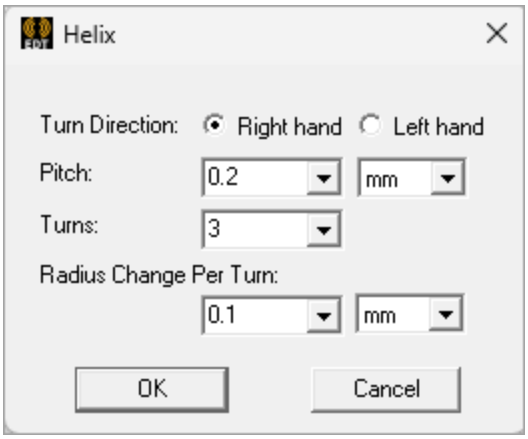
1. Create and select the line or 2D object you want to sweep to form a helix. Selecting a valid object enables the **Helix** icon on the **Draw** tab of the ribbon.
2. From the menu bar, click **Draw** > **Helix** or, on the **Draw** ribbon tab, click the **Draw helix** icon:

Ansys Electromagnetics Suite 2026 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. 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).
    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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Drawing a Segmented Helix with Polygon Cross-Section

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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

3. Click **OK**.

Drawing a Segmented Helix with Rectangular Cross-Section

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**.

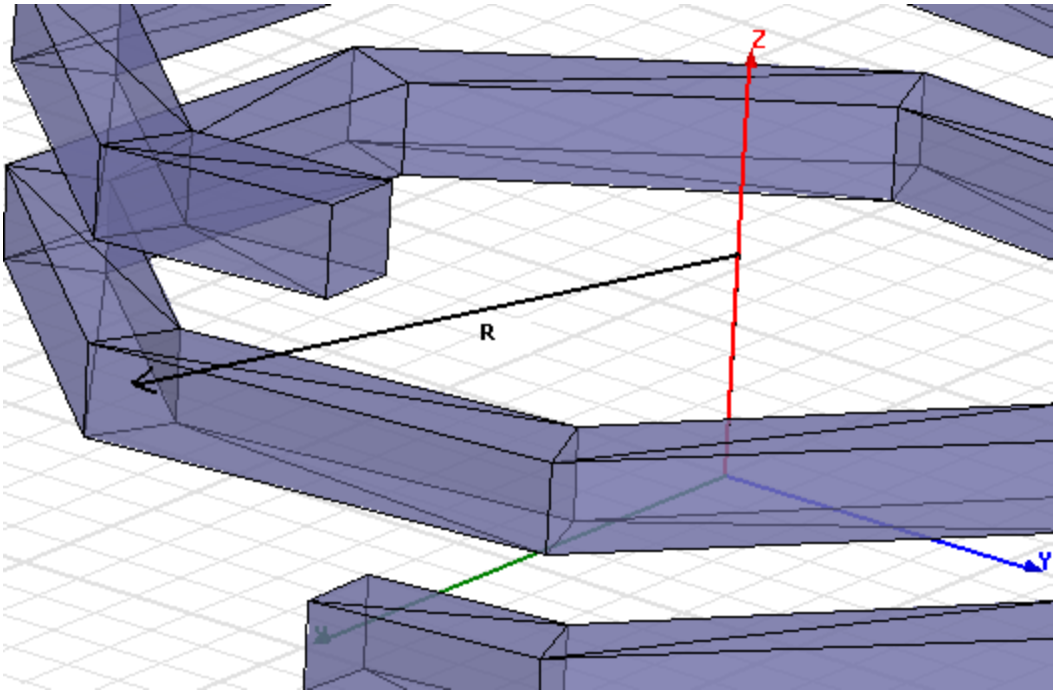
   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

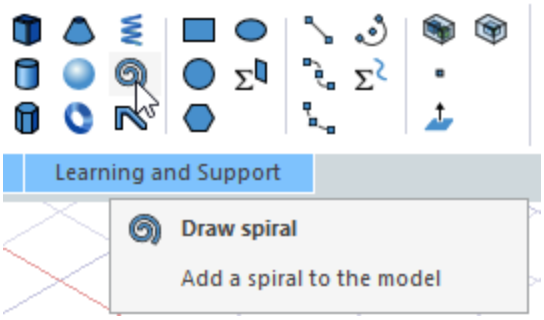
3. Click **OK**.



Drawing a Spiral

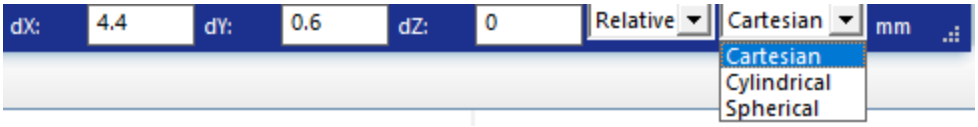
A spiral is a 2D or 3D spiral object created by sweeping an object around a vector. Sweeping a 1D object results in a 2D sheet object. Sweeping a 2D sheet object results in a 3D solid object. Before you draw a spiral, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or 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 2026 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.

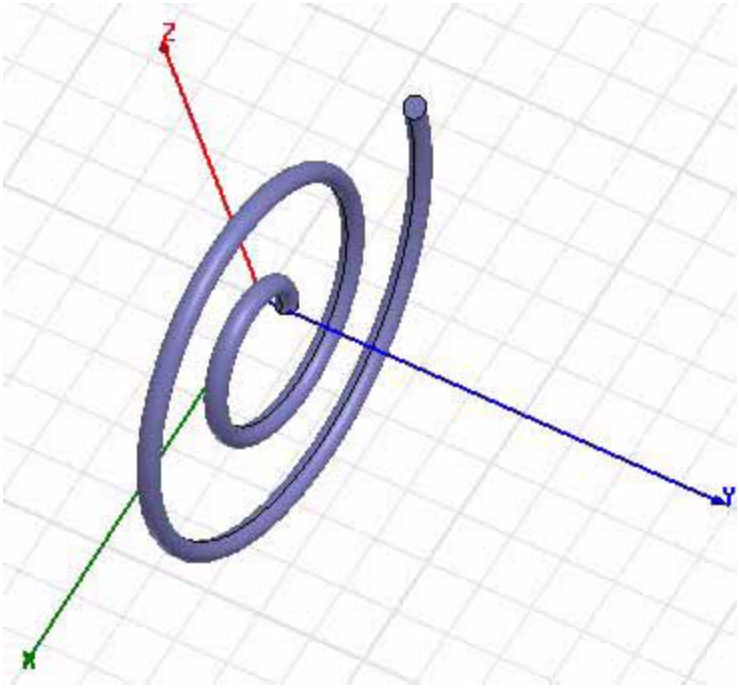
   The radius of the first turn is measured from the center point of the 1D or 2D object you are sweeping to the vector you drew.

6. Click a unit for the radius in the drop-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 2026 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.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

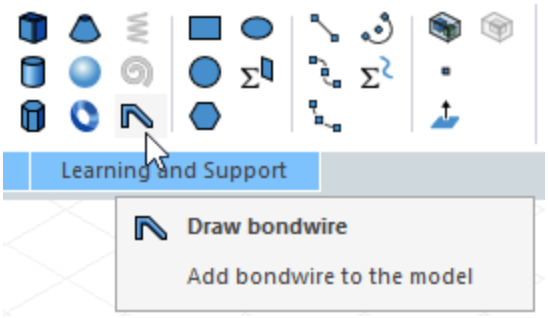
To see newly created DLLs, click **Draw** > **User Defined Primitive** > **Update Menu**.

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:



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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Type** selection changes the dialog box's 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.

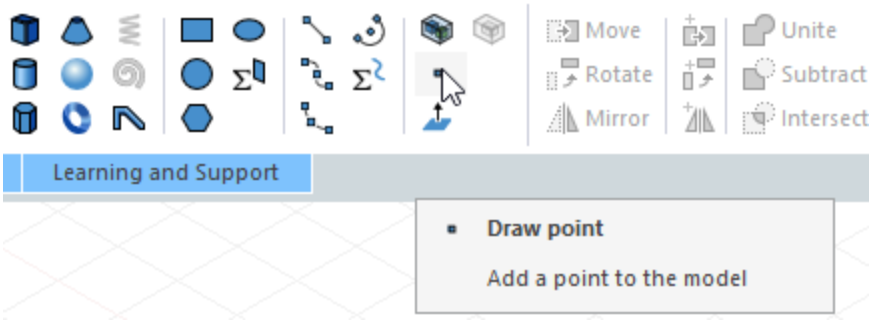
   Optionally, type a new value in the **h2** text box. Include the height's unit of length.

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**.

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.

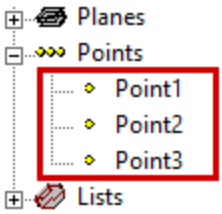
1. From the menu bar, click **Draw** > **Point** or, on the **Draw** ribbon tab, click the **Draw point** icon:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

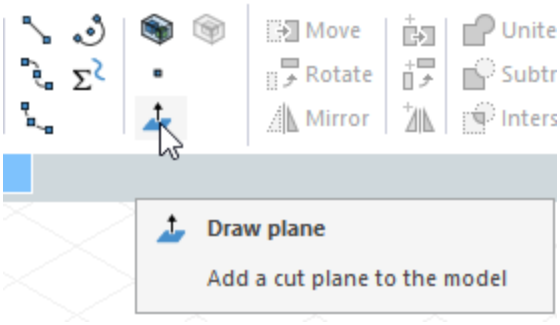


> **Note:**
>
> There is no visible indication of the point in the drawing area unless it is selected in the History Tree. Also, the point cannot be selected graphically once added to the model. It must be selected from the History Tree or from the **Geometry** window within the **Fields Calculator**.

Drawing a Plane

A plane object is a cut plane through the problem region. You can use plot fields 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 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:



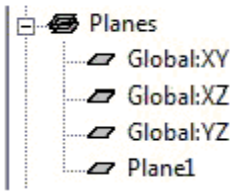
2. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   To delete the selected point and start over, press **Esc**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

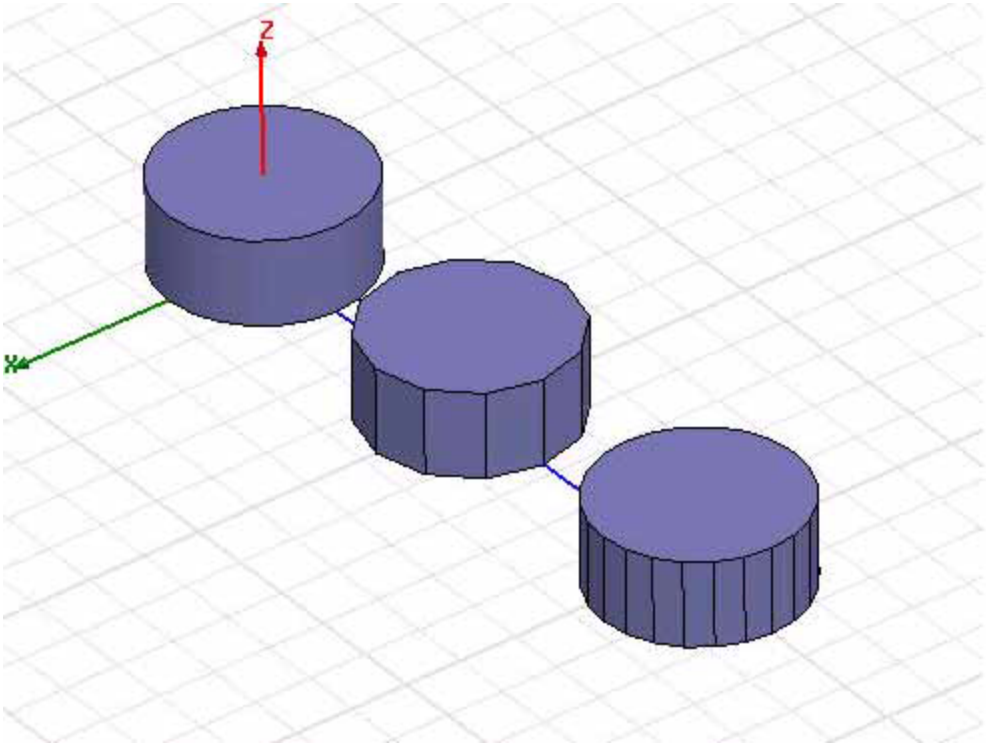


> **Note:**
>
> You only need to draw a plane that does not lie on a predefined xy, yz, and xz plane. Default planes are created on the xy, yz, and xz planes of the global coordinate system as well as any new coordinate system you create.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following model objects can be created as segmented structures:

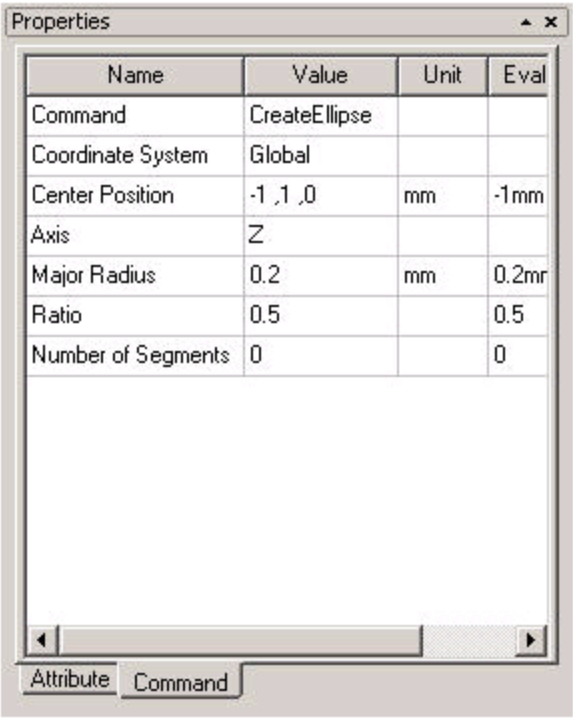
Circle, Ellipse, Cylinder	See: Segmented Objects
Polyline, Arc, Line Segment	See: Converting Polyline Segments

Segmented Objects

To create segmented circles, ellipses, and cylinders use the **Number of Segments** parameter on the Command Tab of the **Properties** dialog box 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 2026 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 2026 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.

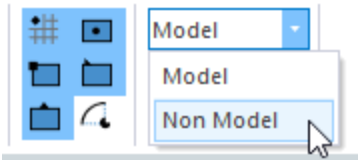
Modify an existing model object to be a non-model object.

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.

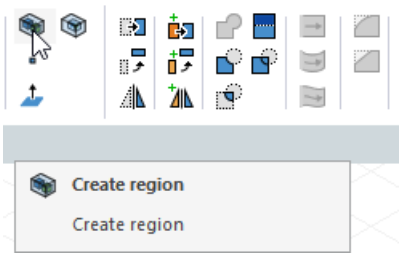
   The object will not be included in the solution process. If the object lies in the problem region, you can plot solution quantities on it.

Drawing a Region

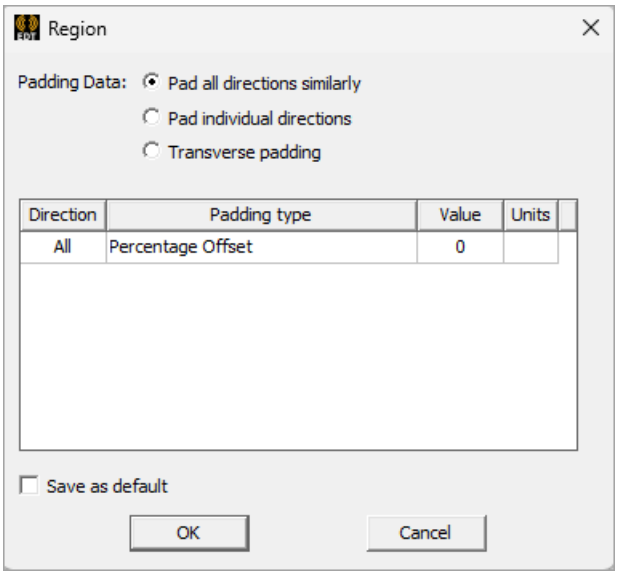
To draw a region encompassing the objects in the current project:

1. From the menu bar, click **Draw** > **Region** or, on the **Draw** ribbon tab, click the **Create region** icon:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

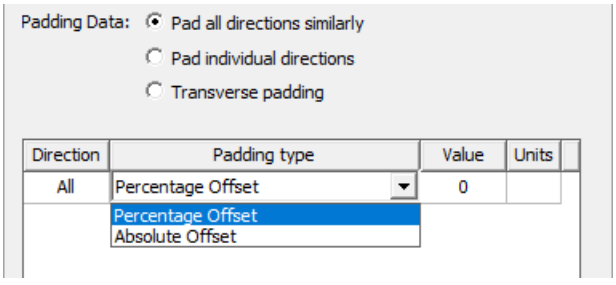


This displays the **Region** dialog box.



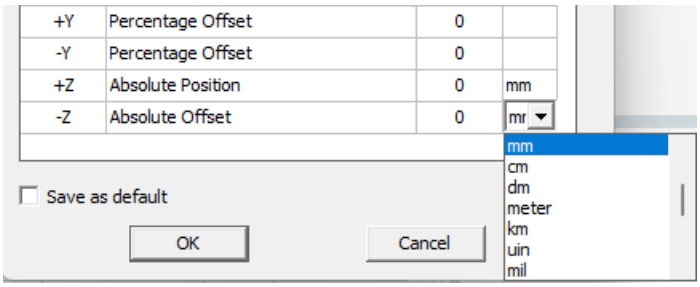
2. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   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.

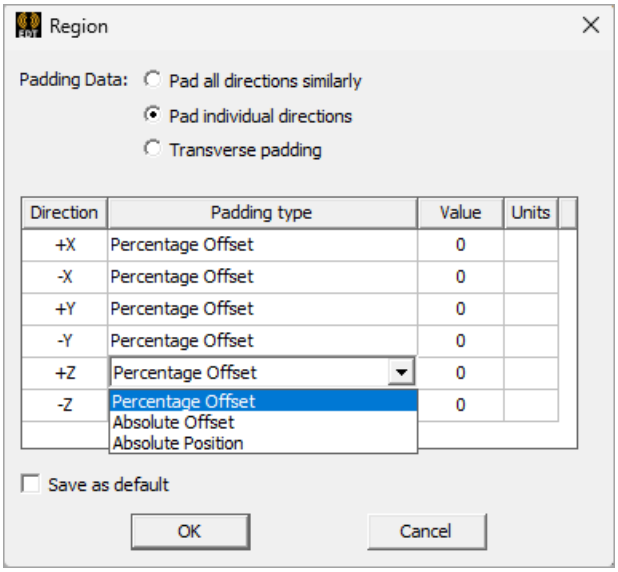


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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.

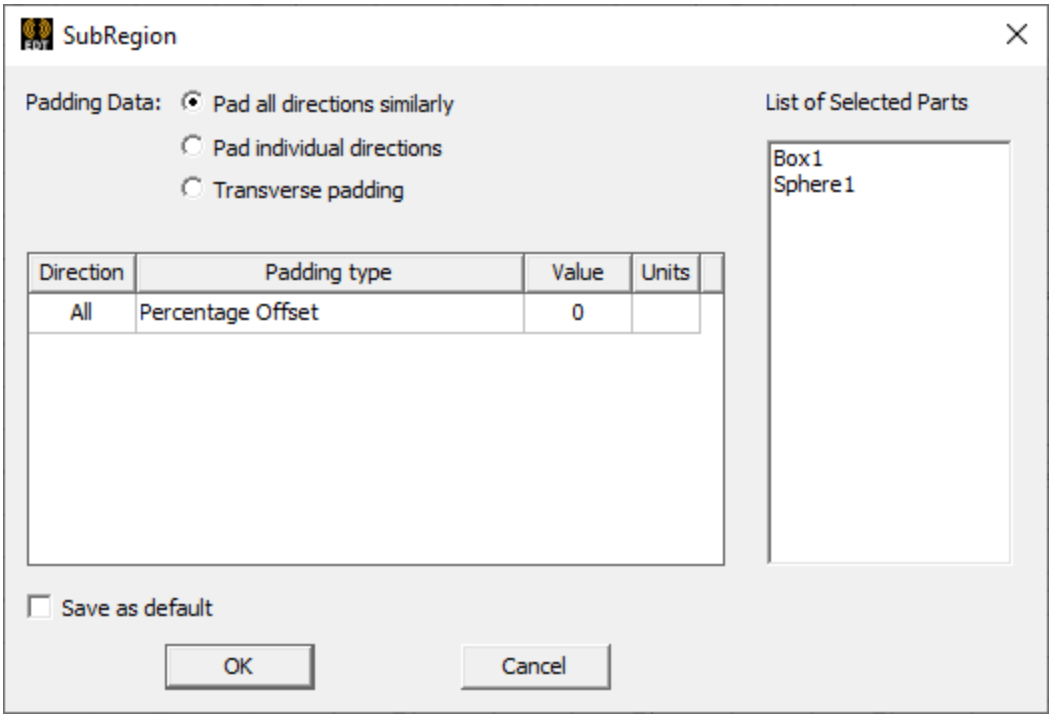
To draw a sub-region in the current project:

1. Select the parts of components that you want to include in a subregion.
2. From the menu bar, click **Draw** > **SubRegion**:



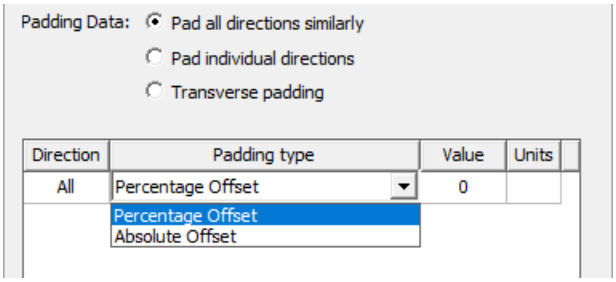
This displays the **SubRegion** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



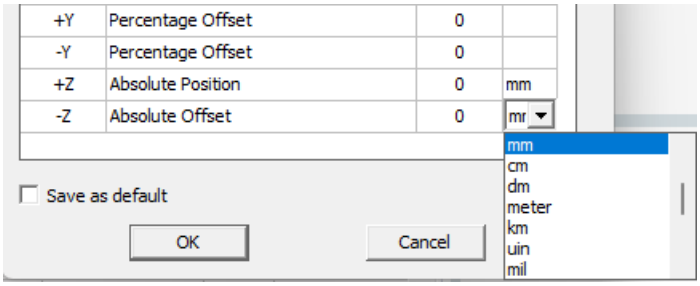
3. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   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.

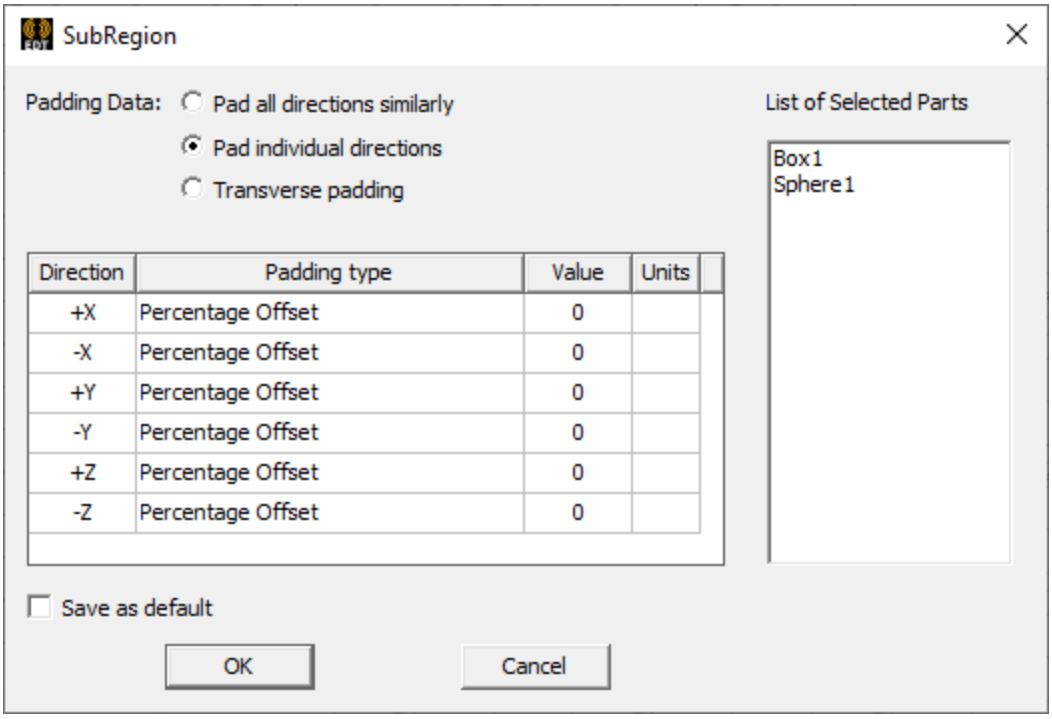


   If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

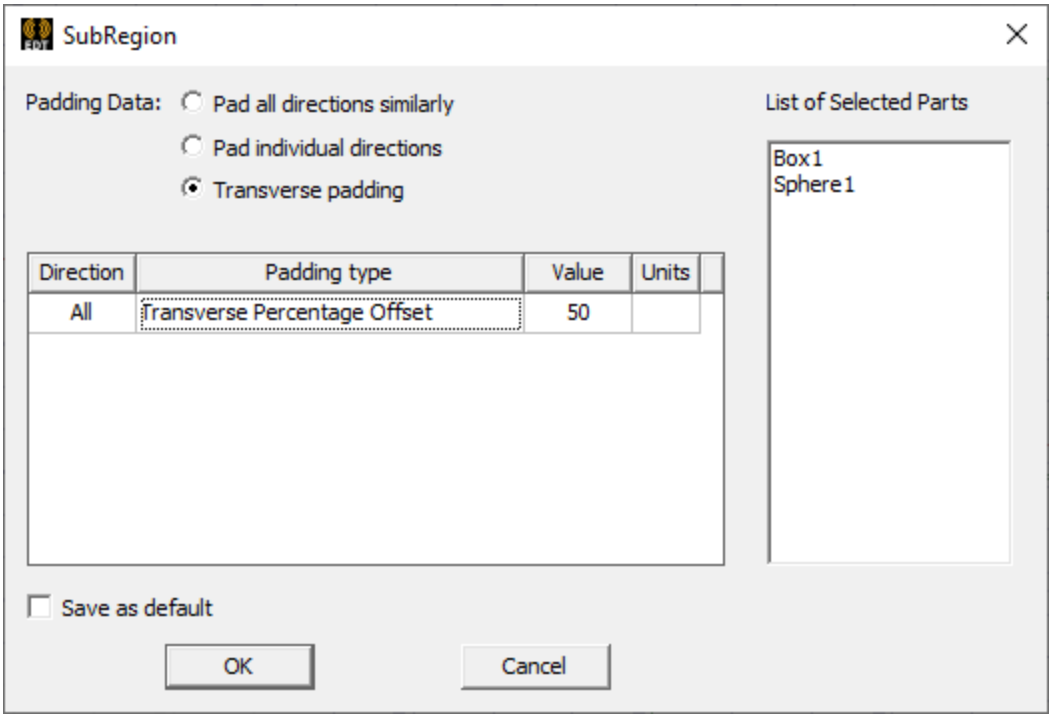


Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.



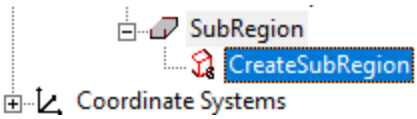
Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

Ansys Electromagnetics Suite 2026 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.

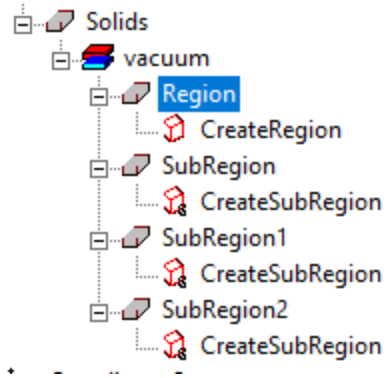
The sub-region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system.



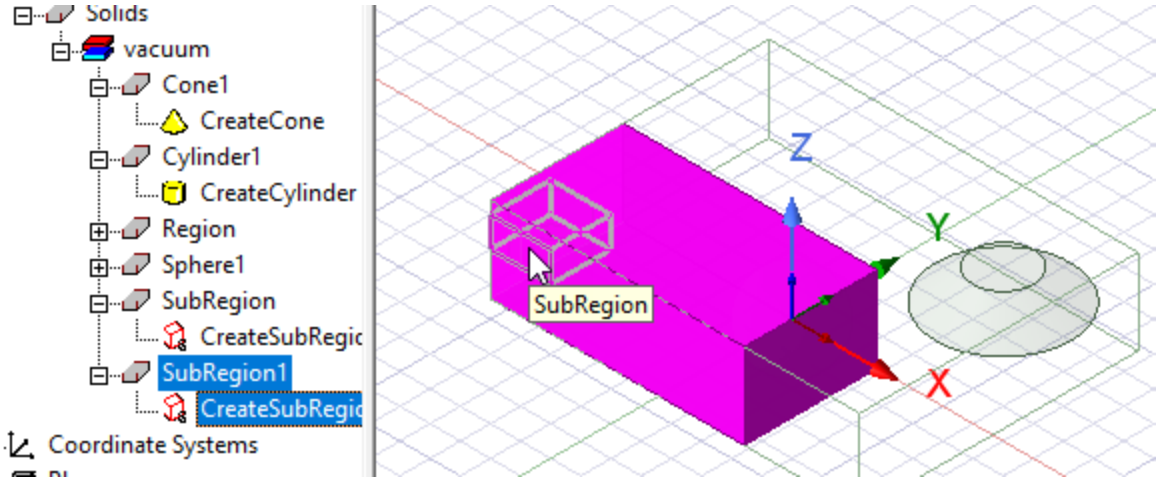
Additional SubRegions and Nested SubRegions.

If you create additional sub-regions, these appear in the History Tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



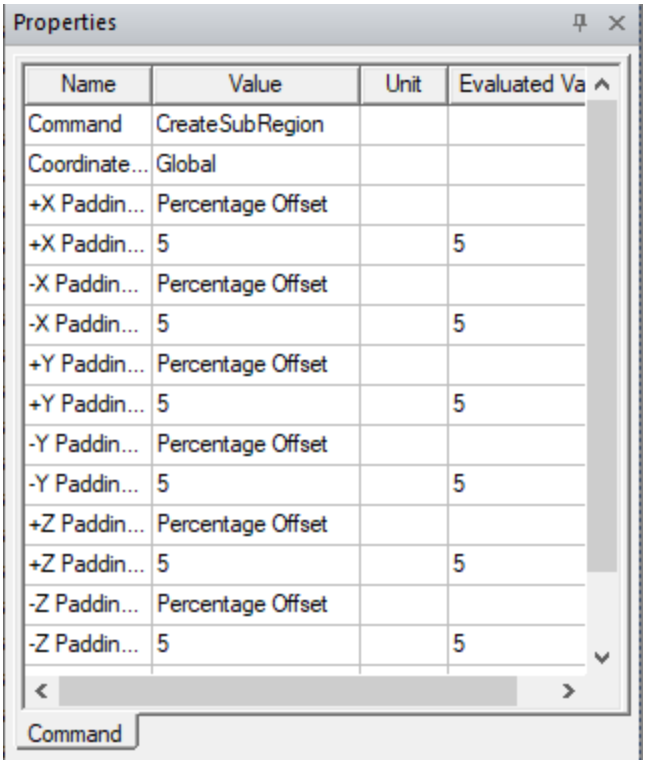
SubRegions can contain other SubRegions, that is, SubRegions can be nested.



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 2026 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.

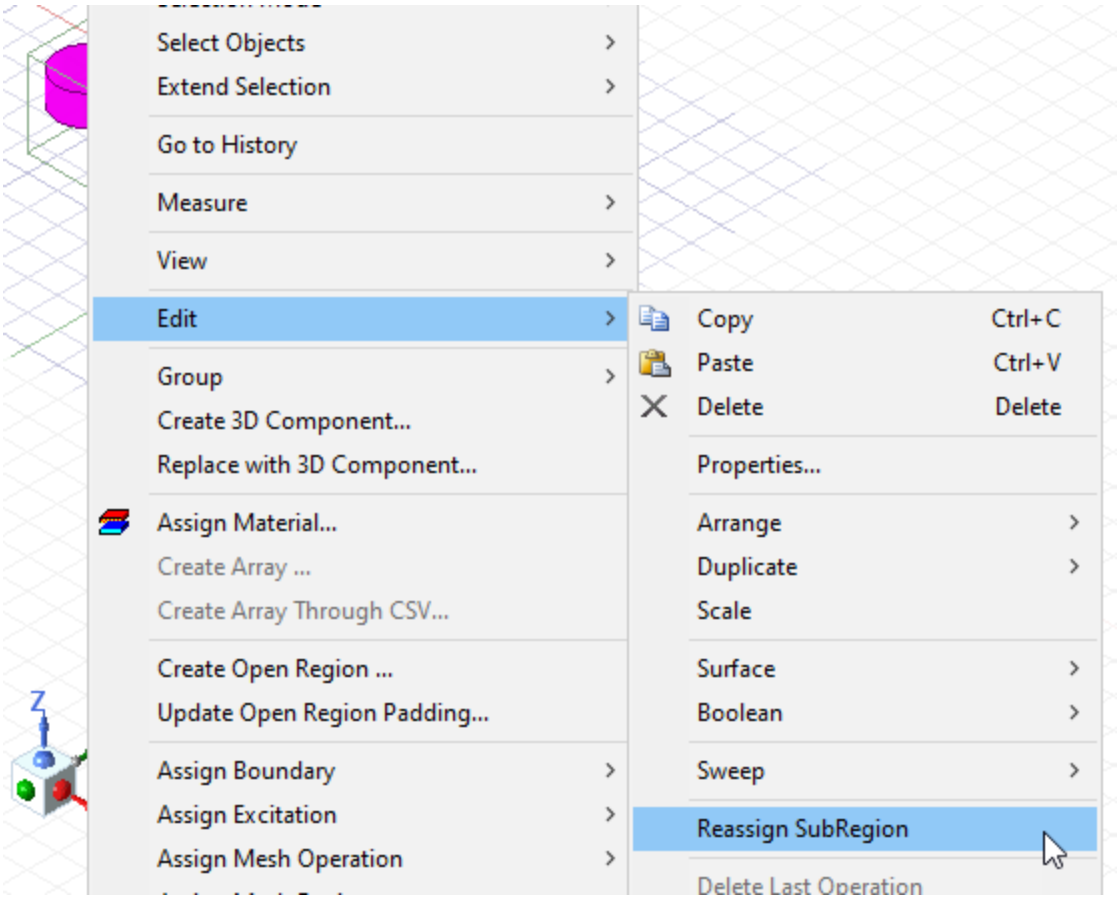
Reassigning a SubRegion

The Modeler allows you to reassign 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.

To reassign a sub-region in the current project:

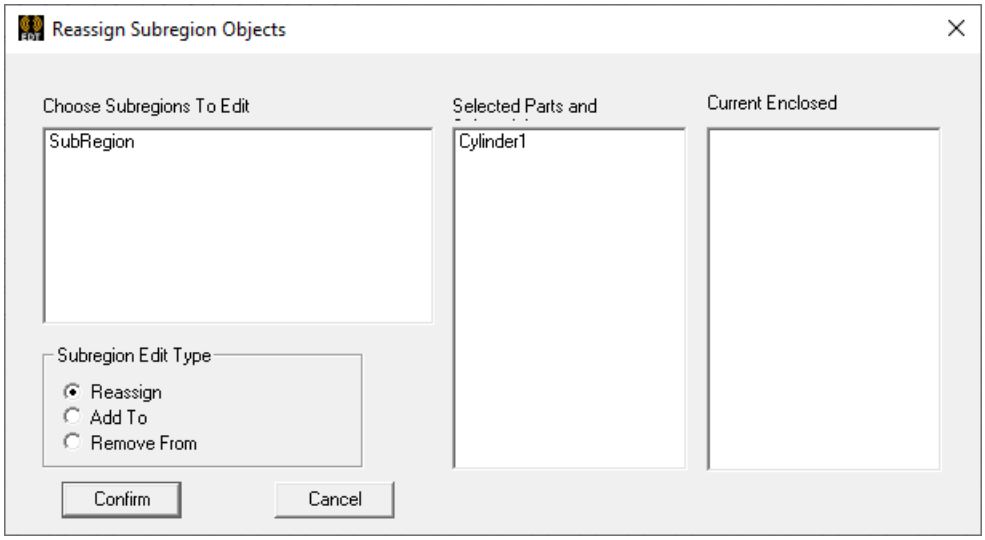
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the components or parts that you want to assign to a subregion.
2. Right-click in the Modeler window to invoke the shortcut menu and click **Edit** > **Reassign SubRegion**:

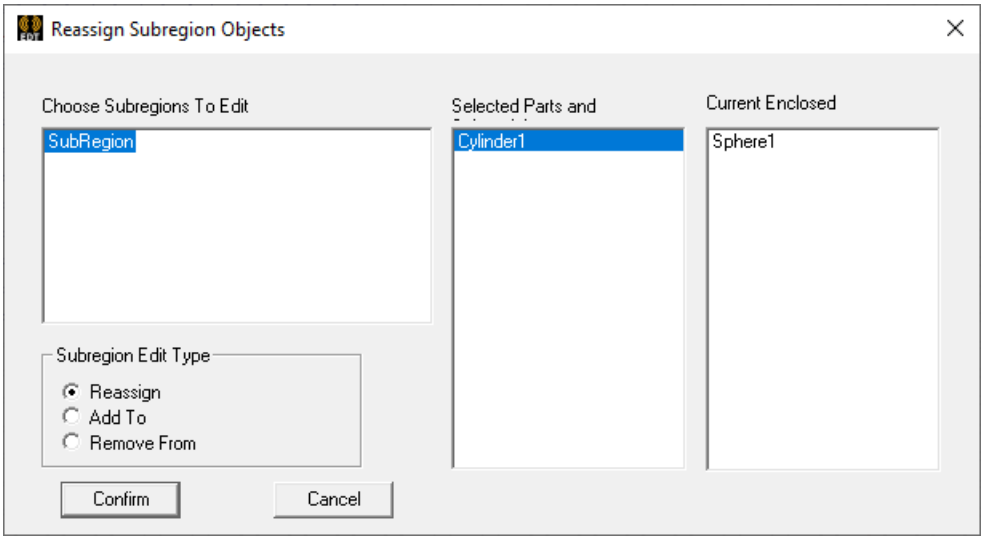


This displays the **SubRegion** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



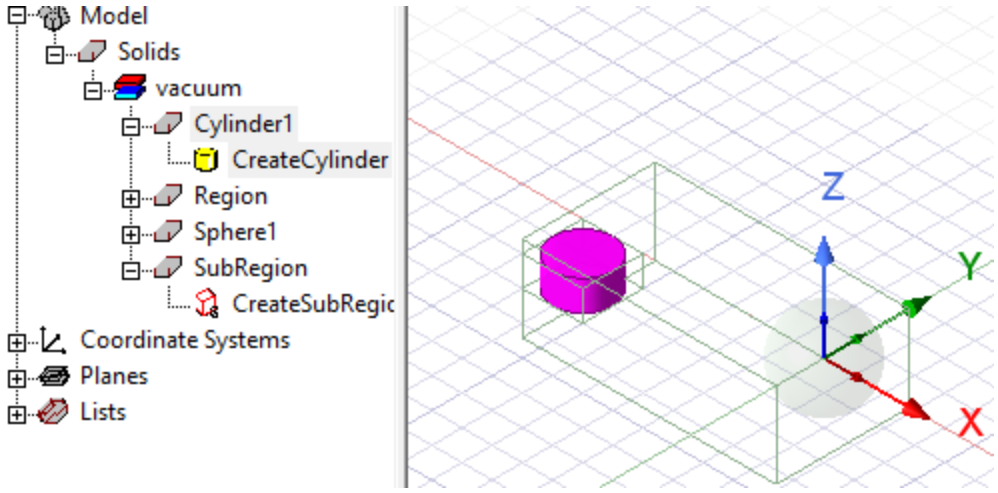
3. Select the SubRegion to Edit and the Parts to Reassign. If there are multiple SubRegions, these will be listed. When you have selected a SubRegion, the Currently Enclosed components are listed.



4. Specify whether to Reassign the selected parts to the SubRegion, or Add To the SubRegion, or Remove from the SubRegion.
5. Click **Confirm** to close the dialog complete the reassignments.

The sub-region is redrawn, selected, and displayed in the History tree. It is created using the current coordinate system.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Suppressing for Physics

In Icepak, you can suppress a selected geometry's or component's design data during a simulation. In design geometry's properties, the **Suppress For Physics** setting can be set to **True** or **False**. Select **True** to suppress the geometry or **False** to activate it for inclusion in the simulation. When an object is suppressed for a simulation, design data assigned to it are not factored into the analysis, including the following:

> **Note:**
>
> In order to suppress the design data, every piece of geometry linked to the assignment (e.g., boundary condition or a mesh region) must be suppressed. If you create one of the following design data assigned to fully suppressed objects, the design data is suppressed. If design data are assigned to partially suppressed objects (e.g., one suppressed object and two active), only the active objects are factored into the analysis.

- Boundary conditions

  > **Note:**
  >
  > Partial suppression is not supported for blowers, recirculating openings, and networks. All objects assigned to the boundary condition must be suppressed.

- Monitors
- Mesh regions
- Mesh operations

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

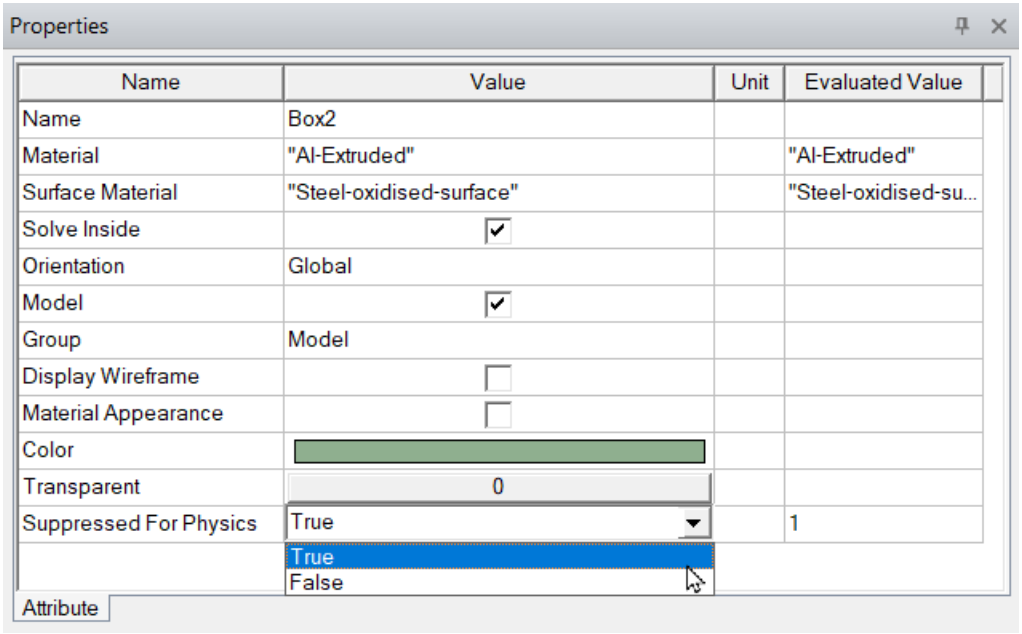
- Solar loading
- Initial temperature (transient designs)

You can parameterize an object's suppressed state to make objects active or inactive during parametric trials. To do so, click in the Value field and enter and define a variable (e.g., $sup).

> **Important:**
>
> To define the variable as True, enter 1. To define it as False, enter 0.

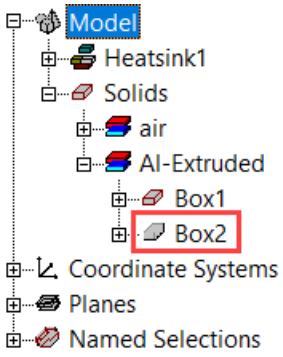
Perform the following steps to suppress an object's design data for a simulation.

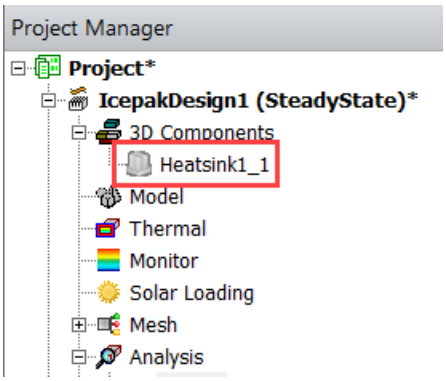
1. Do one of the following:
   a. Select geometry in the **3D Modeler** window.
   b. Select geometry object in the History tree.



2. In the **Properties** window, select **True** from the drop-down list next to **Suppressed For Physics**. Individual geometries that are suppressed appear inactive (gray) in the History tree. Suppressed components appear inactive under **Components** in the **Project Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Considerations for Suppressing Objects

The following should be considered when suppressing geometry or components.

- Suppressed objects and assigned design data are excluded from validation checks.
- Suppressing an object invalidates previously generated mesh, solution results, and post-processing objects.
- Component instances can be suppressed. Individual parts of inserted components must be suppressed by editing the component definition in its own design and then saved.
- Advanced mesh settings (e.g., maximum element sizes) are recomputed if a mesh region assigned to multiple objects has one or more objects suppressed.
- Automatic mesh settings are not recomputed if a mesh region assigned to multiple objects has one or more objects suppressed.
- For solution setups using mesh links or the **Continue/Start from a previously solved setup** option, if you suppress an object in the target design, the link is broken as the two designs are no longer geometrically equivalent.
- If a part is suppressed in the source design after its solve setup is linked to a target design, the source design's solution is invalidated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - You can then solve the source design without error.
    - When attempting to solve the target design, an error message is displayed stating that the source and target designs are not geometrically equivalent.
- If you suppress an object in target design, the solution is invalidated. Attempting to solve it again displays an error message stating the source and target designs are not geometrically equivalent.
- Design data for suppressed objects are ignored for designs with one- and two-way coupling.
- Data exported to Ansys Sherlock does not include data related to suppressed objects.

Snapping Geometry

In Icepak, you can use the Snap tool to quickly adjust the location and size of faces and objects, including the geometry of heat sink and fan native components. You can also adjust the location of a polyline's vertex to change its shape.

> **Note:**
>
> When snapping two pieces of geometry, their orientation can be set to different coordinate systems. For example, one piece of geometry can reside on the Global coordinate system, and another can reside on a relative coordinate system.

The following geometry types are supported:

- 2D Polylines
- 2D Rectangles
- 2D Circles
- 2D Polygons
- 3D Boxes
- 3D Cylinders

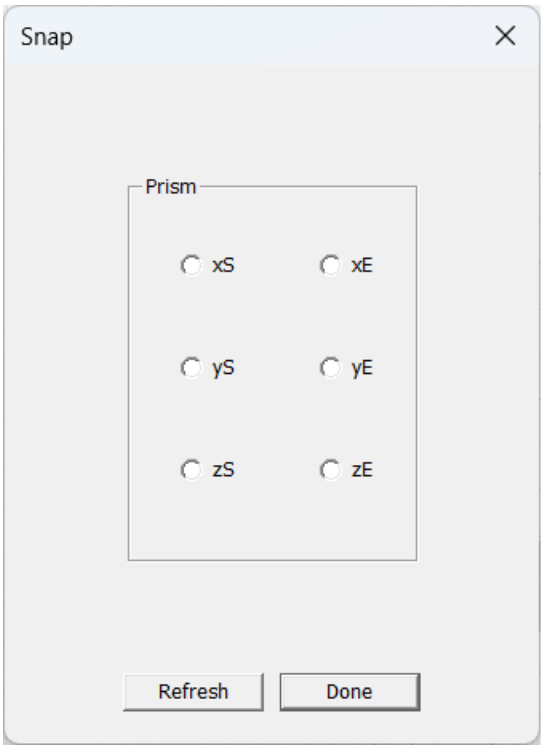
> **Note:**
>
> You can use the Snap tool on imported geometry. The History tree is automatically updated to include the edits to size and location.

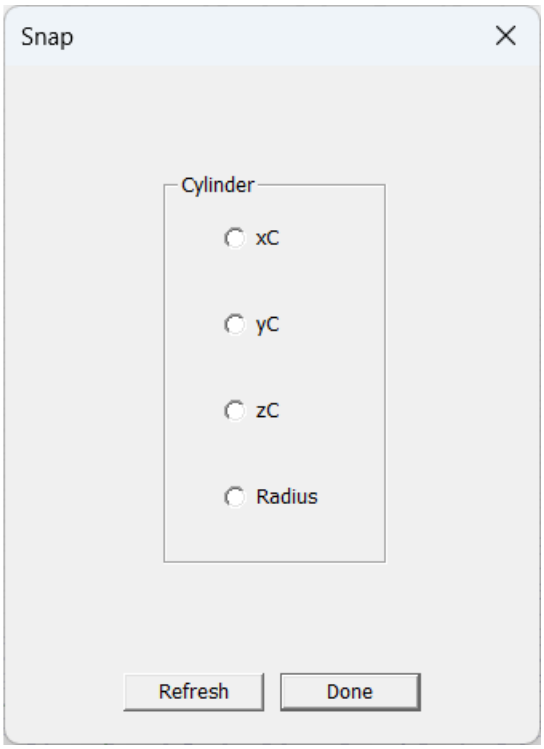
When snapping 3D objects or 2D sheets, the **Snap** dialog box allows you to select the coordinate direction in which you want to move the geometry. The coordinate directions available depends on what type of geometry is selected. The following images display options snapping a prism and a cylinder.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The **Snap** dialog box does not appear when snapping the face of a 3D object.





You can snap multiple objects by pressing **Ctrl** and selecting them in the **3D Modeler** window or History tree. After snapping an object, you can select another object and click **Refresh** to update the **Snap** dialog and continue snapping geometry. For cylinders, select Radius to adjust the size of the radius based on the selected point.

Note:

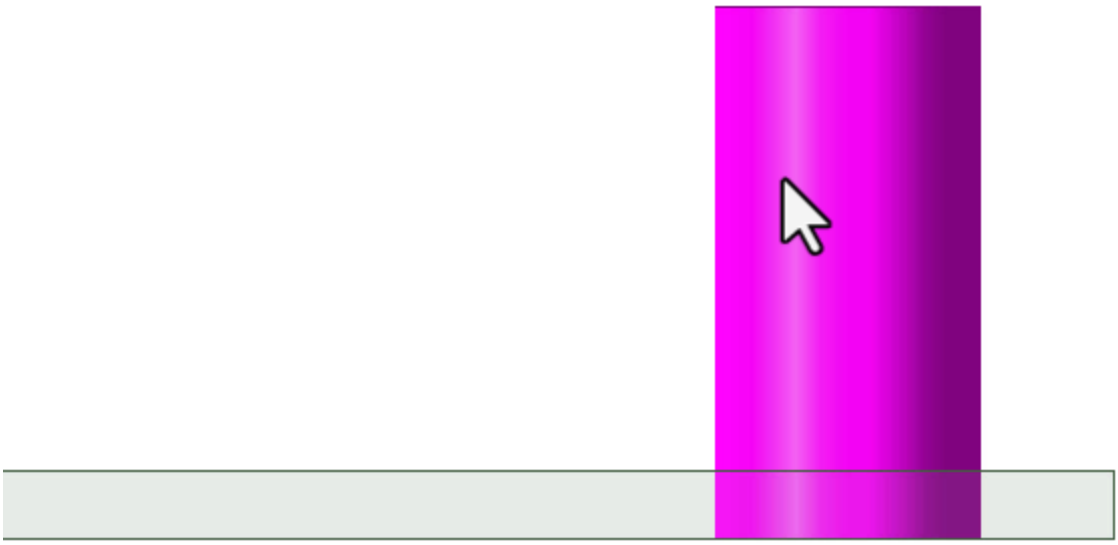
After snapping geometry, you can manually edit the size or location of a geometry by selecting the geometry in the **3D Modeler** window or History tree and editing it in the **Properties** window.

Snapping 3D and 2D Geometry

The following instructions detail how to snap 3D and 2D geometry.

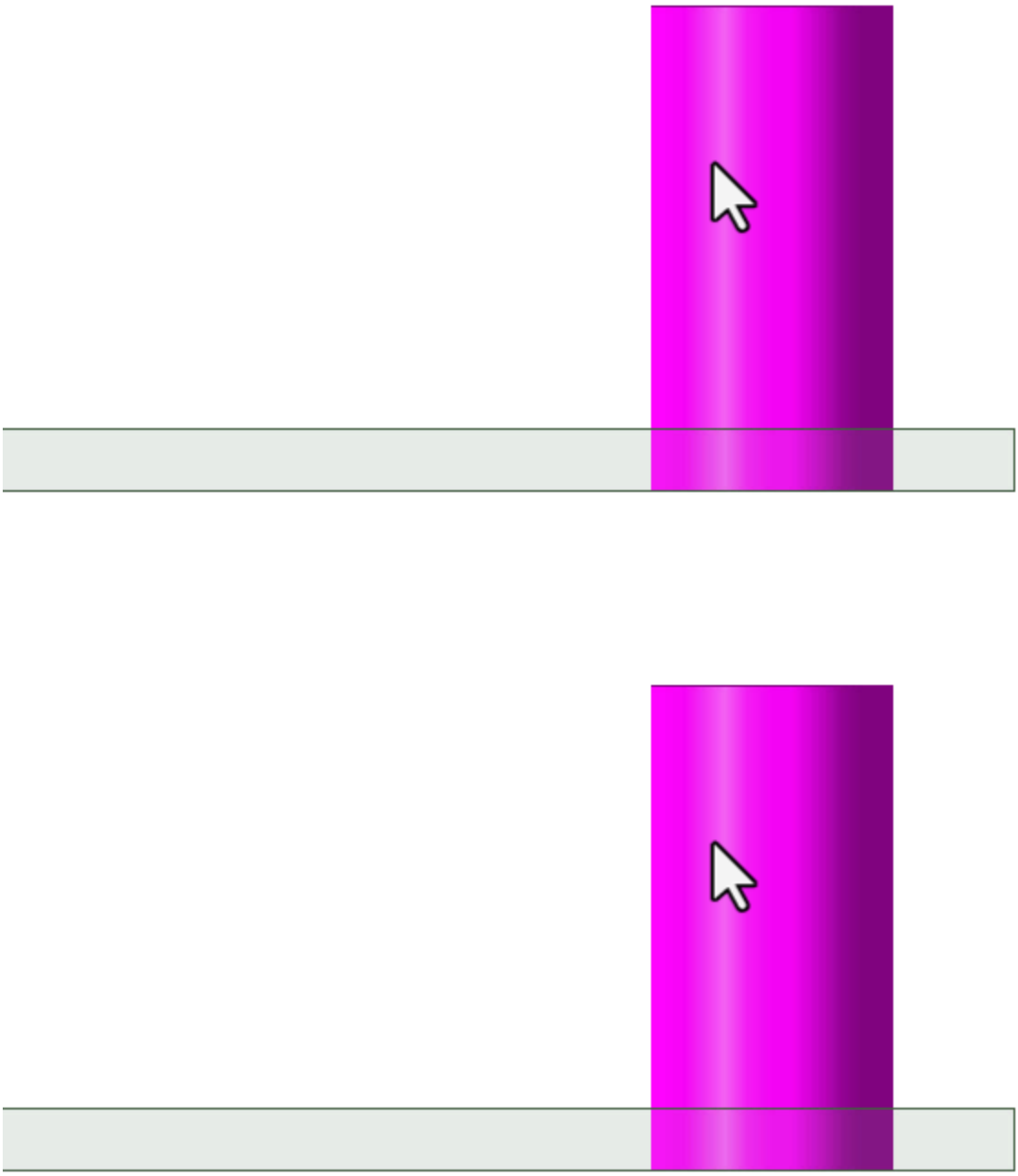
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Press O to enter object selection mode.
2. Do one of the following:
    a. Select an object in the **3D Modeler** window.
    b. Select an object in the History tree.



3. Do one of the following:
    a. Right-click in the **3D Modeler** window and select **Snap**.
    b. From the **Icepak** menu, select **Snap**.
4. In the **Snap** dialog box, select the coordinate direction in which you want to move the geometry.
5. In the **3D modeler** window, select the point to which you want the geometry to move. The location is updated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

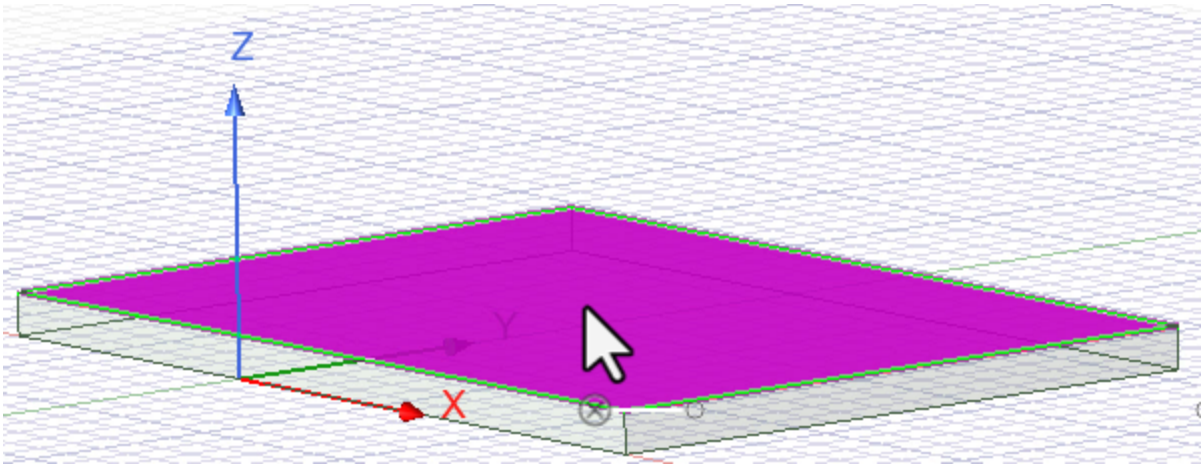


Snapping Faces

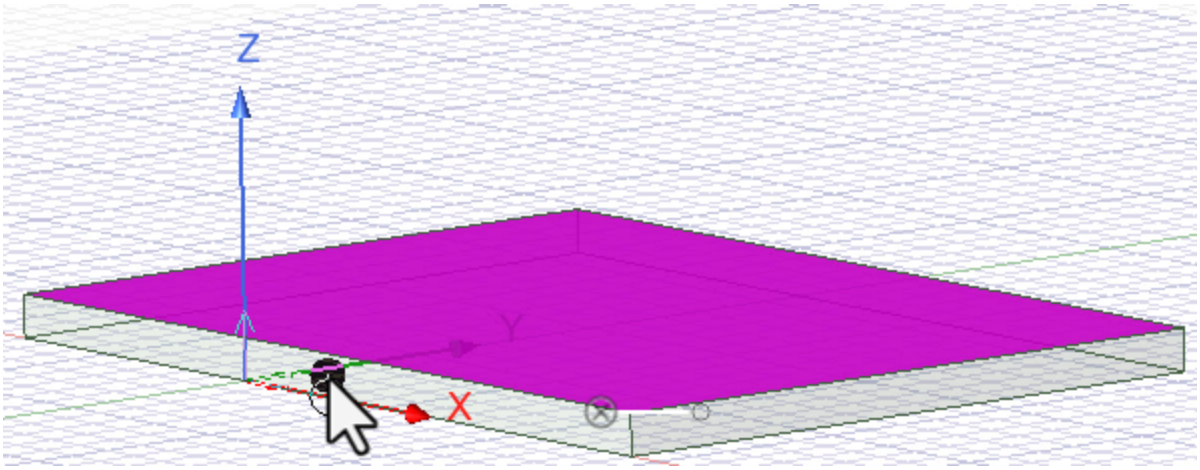
The following instructions detail how to snap a face of 3D geometry while in face selection mode.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

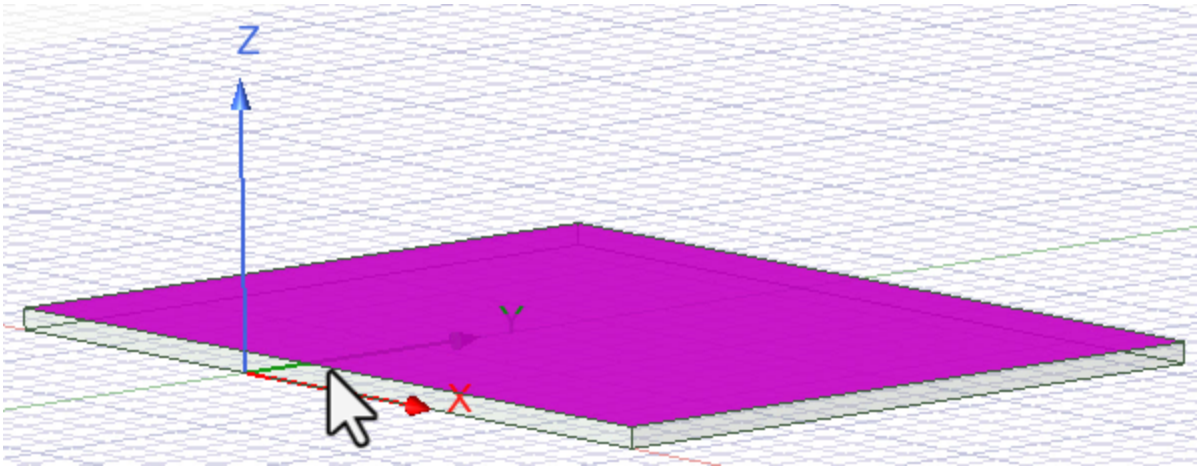
1. Press F to enter face selection mode.
2. Do one of the following:
   a. Select a face or sheet in the **3D Modeler** window.
   b. Select a face or sheet object in the History tree.



3. Do one of the following:
   a. Right-click in the **3D Modeler** window and select **Snap**.
   b. From the **Icepak** menu, select **Snap**.
4. In the **3D modeler** window, select the point to which you want the face to move. The location is updated.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

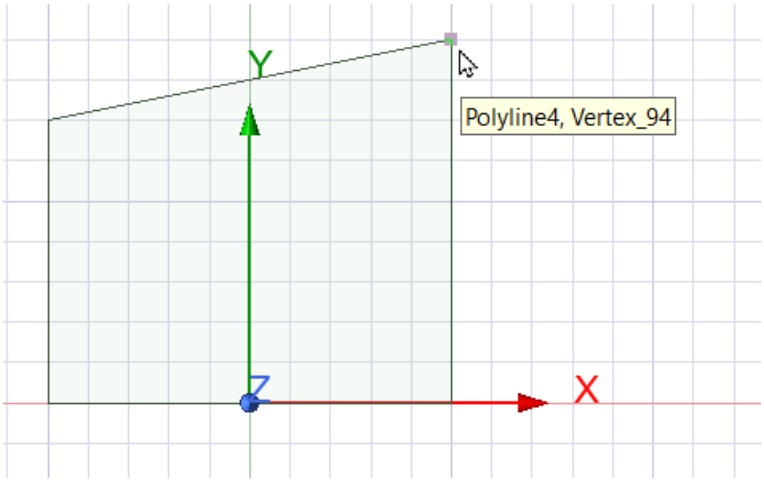


Snapping a Polyline Vertex

The following instructions detail how to snap a vertex of a polyline while in vertex selection mode.

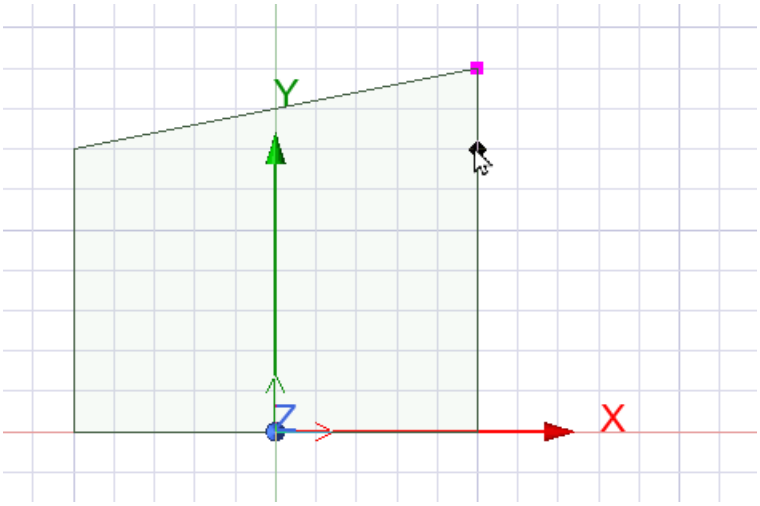
> **Note:**
>
> You can snap multiple vertices from different polylines. Snapping is supported for polylines that are closed and have a CoverLines operation.

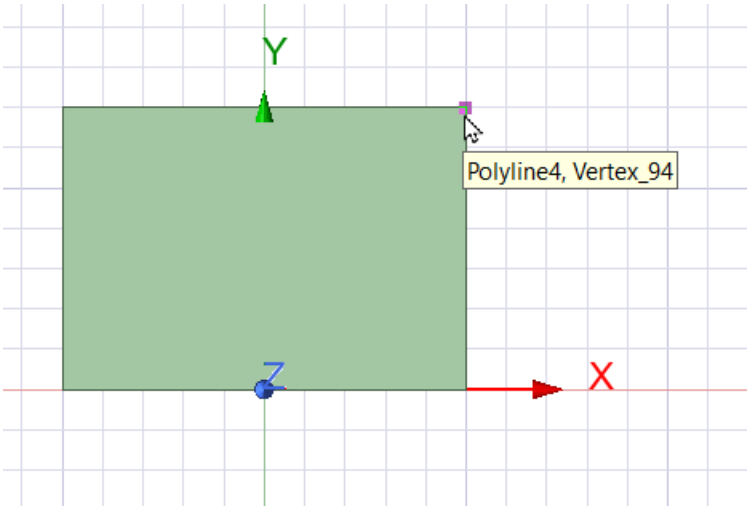
1. Press V to enter vertex selection mode.
2. Select a vertex in the **3D Modeler** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Do one of the following:
    a. Right-click in the **3D Modeler** window and select **Snap**.
    b. From the **Icepak** menu, select **Snap**.
4. In the **3D modeler** window, select the point to which you want the vertex to move. The location is updated.

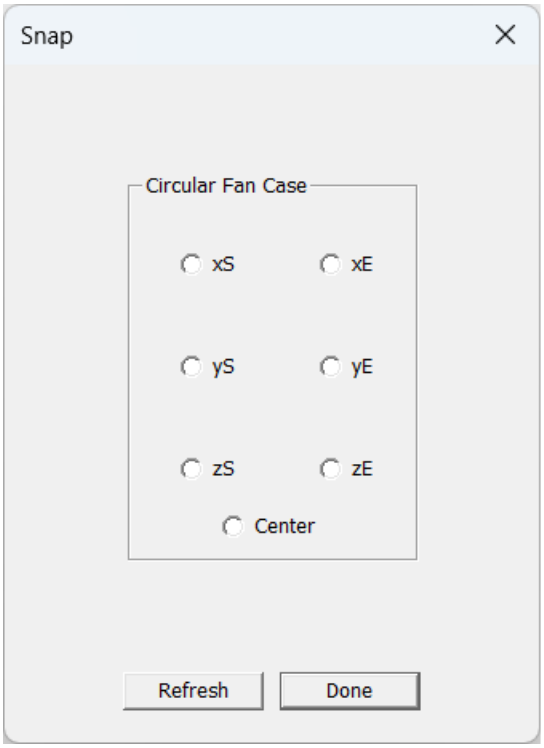


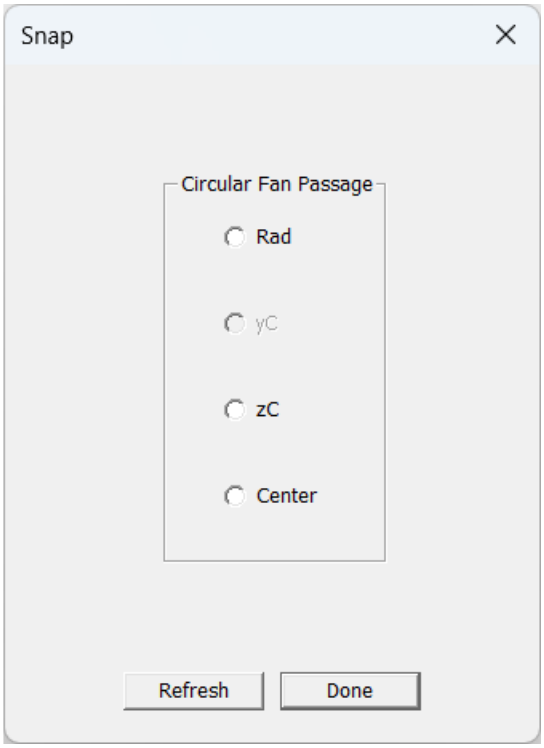


Snapping Fan Components

You can use the snap tool to modify the pieces that compose fan native components: cases (3D fans only), hubs, and passages. The coordinate directions available depends on the fan shape. The following images display options snapping circular and rectangular fans.

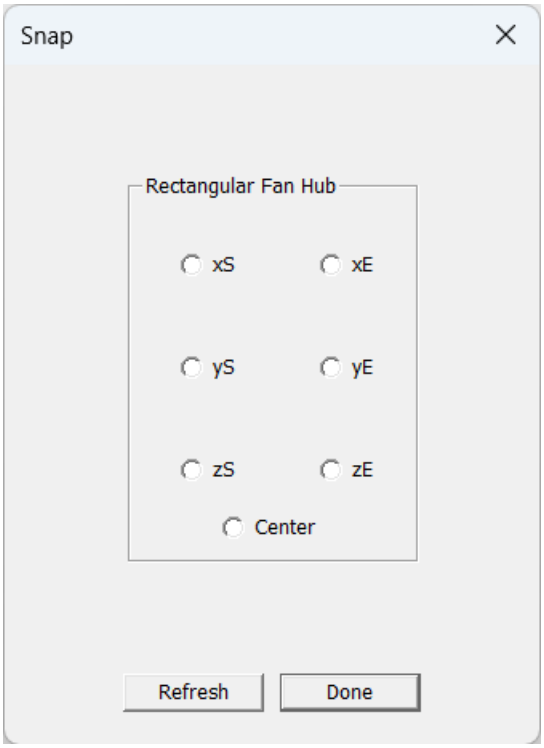
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





A circular fan case is rectangular geometry. It can be snapped in any coordinate direction. A circular fan's passage and hub are circular geometry. Select **Rad** for the snap tool to resize the geometry based on its radius. The other coordinate direction depends on which plane the fan resides. Select this option to move the fan in that direction. Select **Center** to place the center of the fan to a selected point along the existing plane.

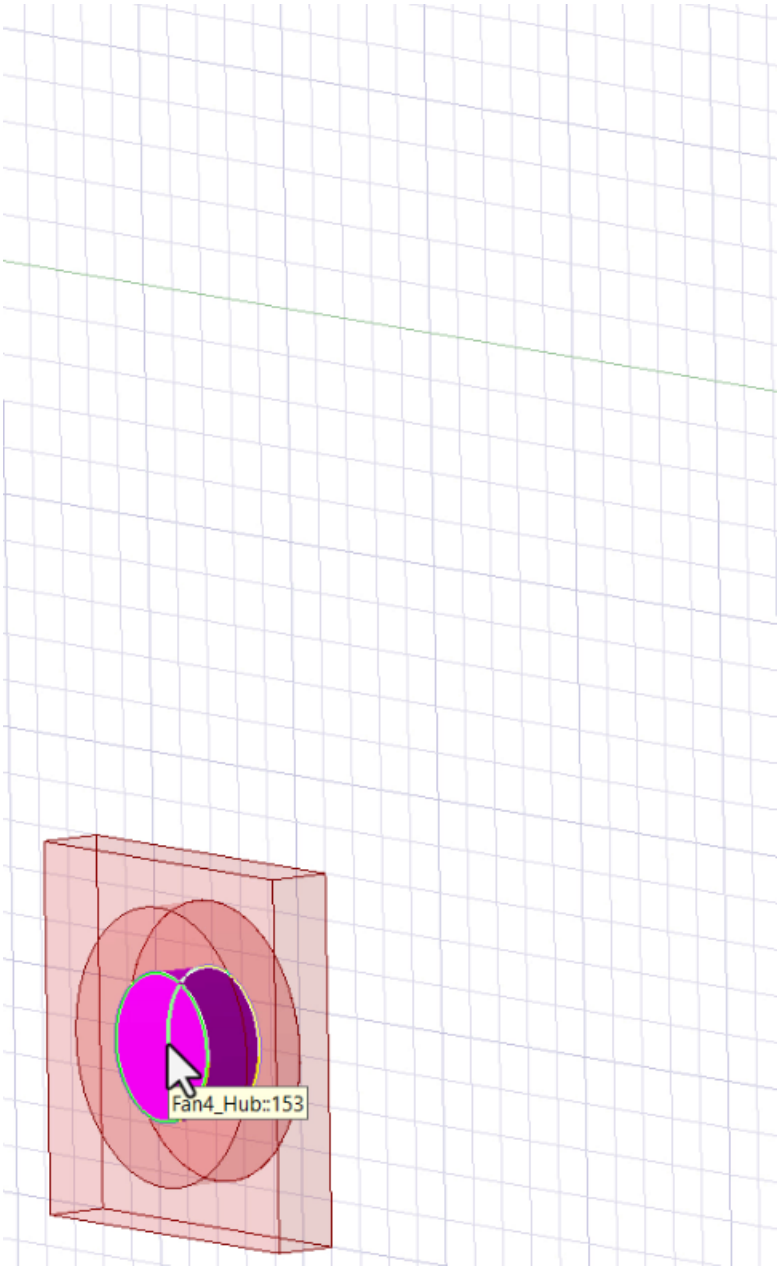
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A rectangular fan's case, passage, and hub are rectangular and can be snapped in any coordinate direction.

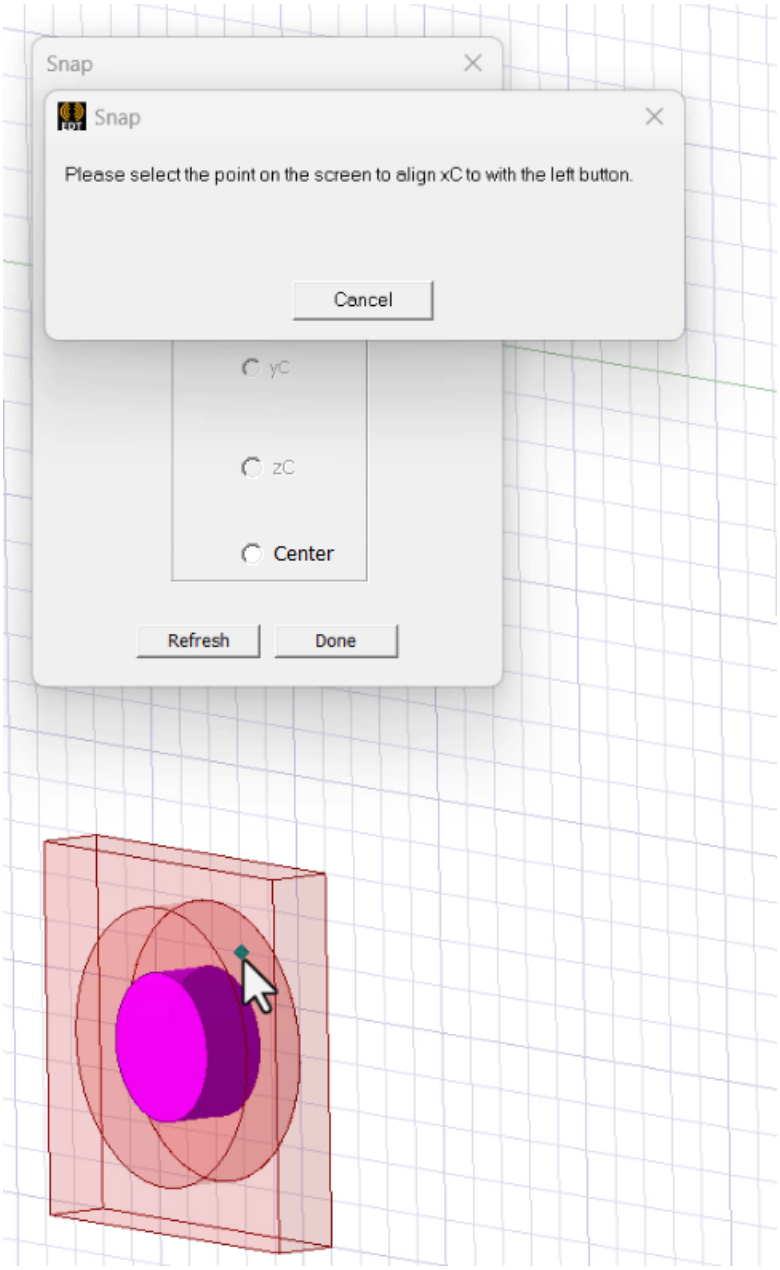
1. Press O to enter object selection mode.
2. Do one of the following:
    a. Select a fan component (case, passage, or hub) in the **3D Modeler** window.
    b. Select a fan component (case, passage, or hub) in the History tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

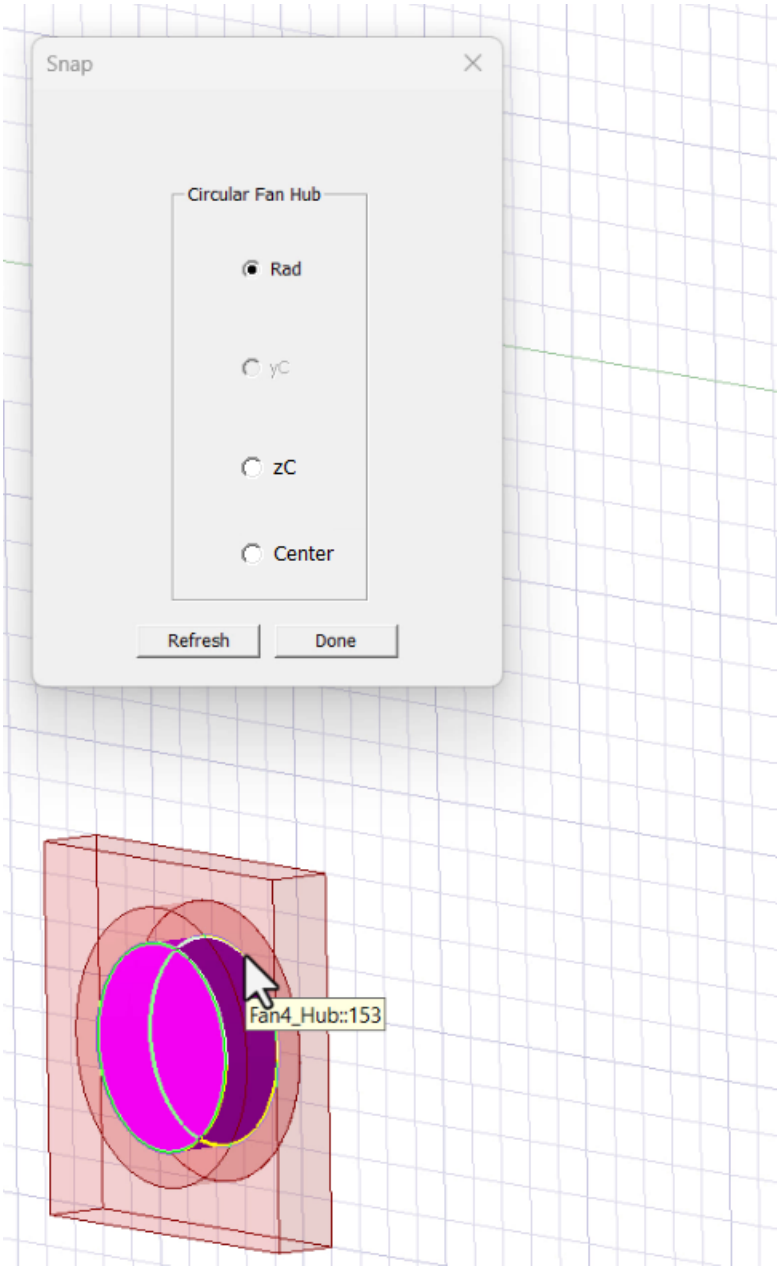


3. Do one of the following:
    a. Right-click in the **3D Modeler** window and select **Snap**.
    b. From the **Icepak** menu, select **Snap**.
4. In the **Snap** dialog box, select the coordinate direction in which you want to move the geometry. For circular geometry, select **Rad** to resize the fan's passage or hub radius.
5. In the **3D modeler** window, select the point to which you want the geometry to move. The geometry is updated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



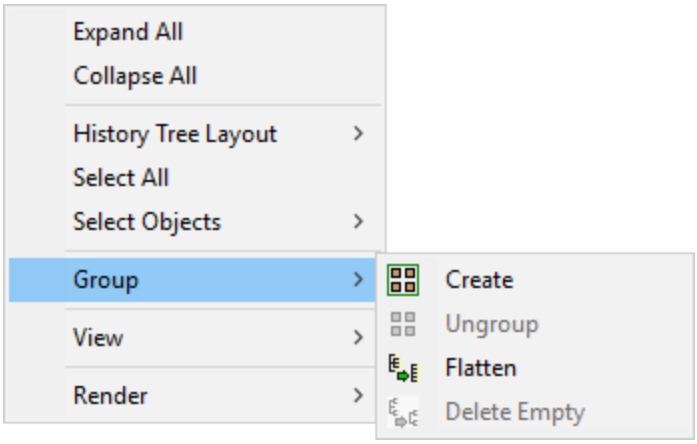
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Group Commands for Modeler Objects

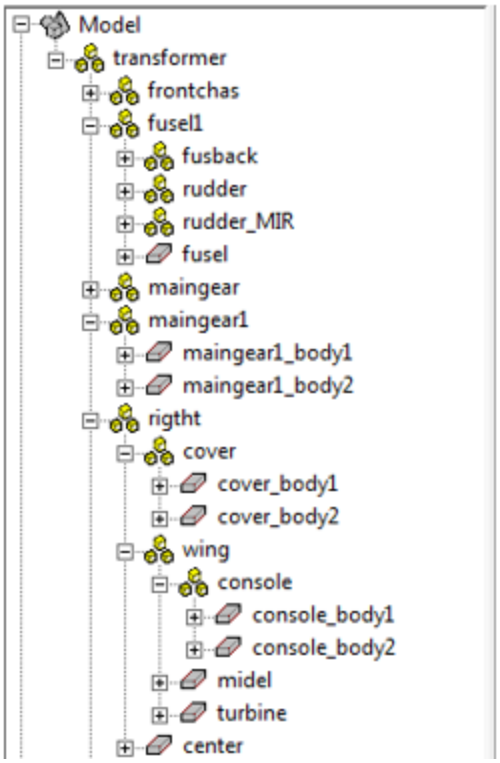
The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups.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

Ansys Electromagnetics Suite 2026 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-groups 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 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

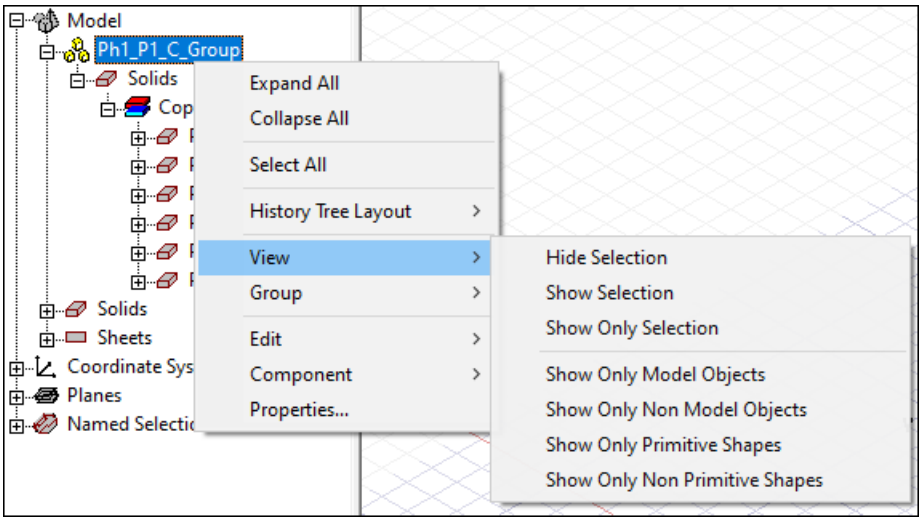
Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren 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

View Options

For groups, you can hide or show the entire group. You can also opt to show only particular types of objects or shapes in that group. The right-click view options are shown below:



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 a 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

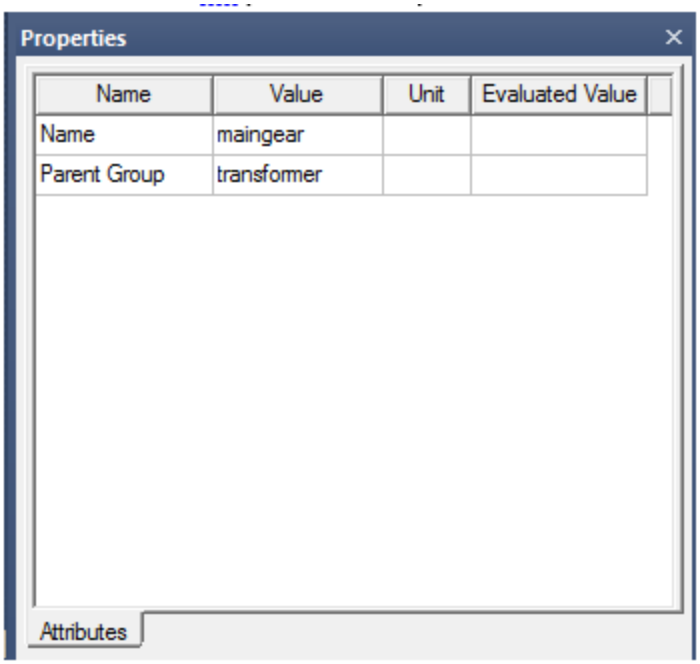
Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate are available for groups. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See 3D Modeler Options: Group Options.

If the group has hierarchy, you can also flatten it.

Group Properties

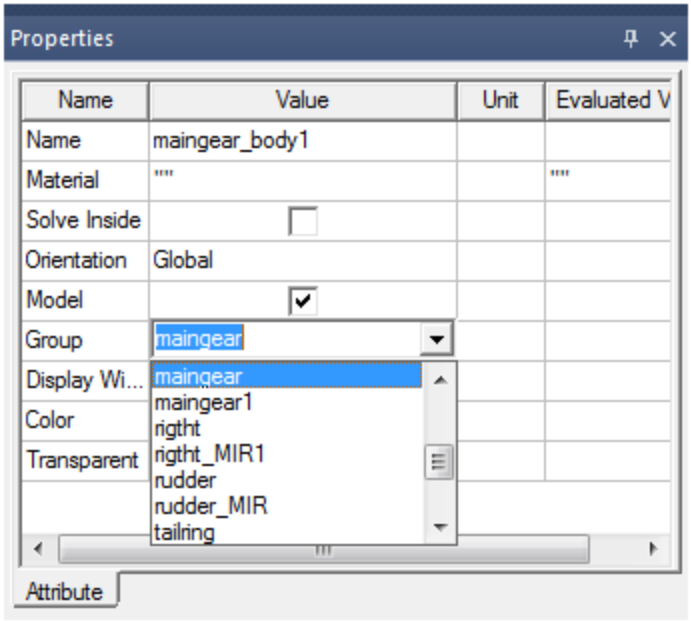
A Group’s Property window is shown when you select a group in the history tree. The Group’s properties include Name and parent Group.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prioritizing Objects

You are able to prioritize the objects to ensure that the model is meshed appropriately. The mesh will consider object priority when an object in the model intersects or is contained within another object in the model. The object with a higher priority (higher number) will retain the mesh where it intersects with an object with a lower priority.

The **Object Priority** dialog box contains two tabs, one displaying the priority list of 3D objects and the other displaying the priority list of 2D objects. The priority of 3D and 2D objects are independent of each other, though by default 2D objects that touch or are embedded inside 3D objects have a higher priority than those 3D objects. Icepak 3D native components that contain both 3D and 2D geometry appear on both tabs. The dialog box has three columns displaying the **Priority** level, **Selections**, and **Type**. **Selections** is a list of objects or component instances, and the **Type** is either **Object** or **Component** based on the selections.

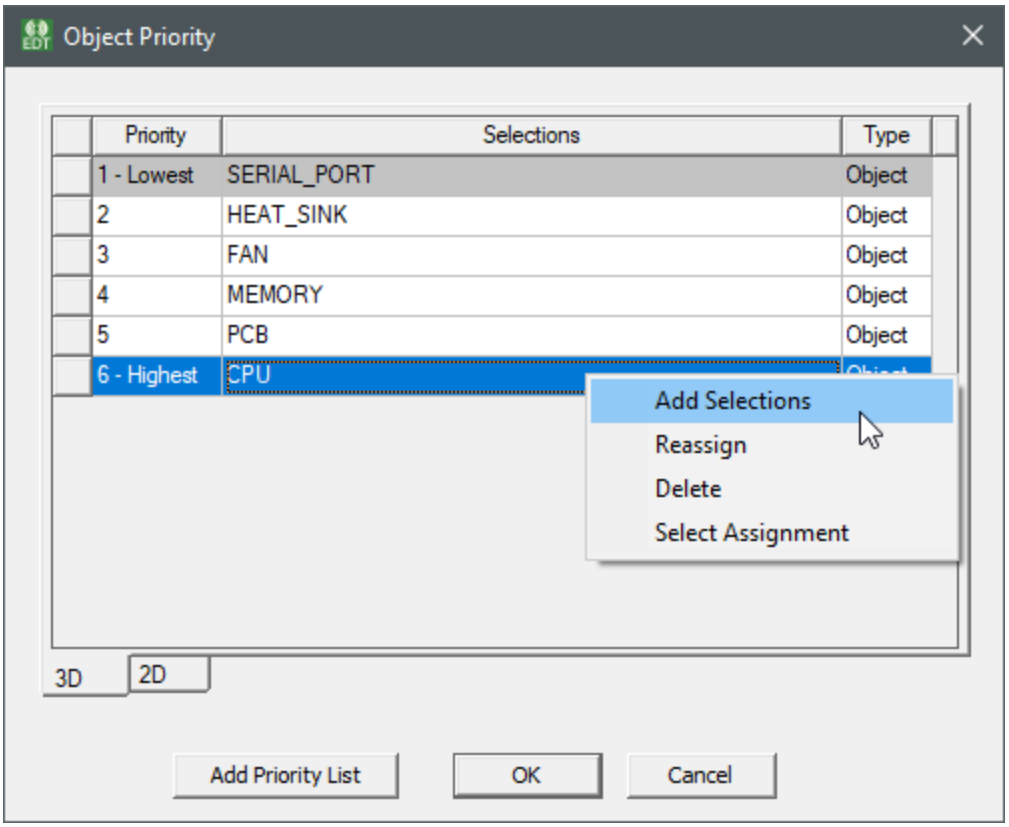
> **Note:**
>
> By default, the **Object Priority** dialog box displays all objects at the same priority level when first accessing it. Follow the procedure below to reprioritize objects in the model.
>
> Non-model objects are not displayed in the **Object Priority** dialog box. When a model object is changed to a non-model, the object priority is retained if you changed it back to a model object.

To edit object priority:

1. Click **Modeler**>**Object Priority**. The **Object Priority** dialog opens on the 3D tab. The lowest priority assignment appears at the top of the list.

   > **Note:**
   >
   > Alternatively, you can access the **Object Priority** dialog box by right-clicking **Model** in the **Project Manager** and selecting **Object Priority**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. As needed, select objects in the **3D Modeler** window or history tree and click **Add Priority List** to create a new level of priority to the list. The newly created priority level is the highest.
3. Drag the row(s) you want to change to the desired order of priority.

> **Note:**
>
> You can right-click in a selection and access the following commands from the menu:
>
> - **Add Selections**: Adds the object selected in the **3D Modeler** window or history tree to the selected priority level.
> - **Reassign**: Reassigns the selected object in the **3D Modeler** window or history tree to the selected priority level and moves the existing object(s) as the lowest priority level.
> - **Delete**: Deletes the priority level and moves the existing object(s) as the lowest priority level.
> - **Select Assignment**: Selects the object(s) in the select priority level in the **3D Modeler** window and history tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Repeat steps 2 and 3 on the **2D** tab.

> **Note:**
>
> By default, an object contained within another object has a higher priority than the container object. If you specify a higher priority for a container object, a validation error occurs.

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.

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, with results for all model analysis, including diagnostic information relating to mesh issues.

2. Choose the objects that have "Invalid Entities Found" and **Perform** > **Heal Objects**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **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. This enables the Check Level menu. The setting can be **Basic**, **Strict**, or **Comprehensive:**
   - **Basic** – Only severe problems that are expected to cause missing facets on faces and/or preclude any kind of modeling will be regarded as errors. Errors include corrupt data structures or identifiers, missing geometry for faces, edges, or vertices in the model.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Basic checks do not indicate a completely valid model for geometric modeling, but rather the model is in a reasonable state to pursue meshing of the model. Users should carefully review mesh on parts where warnings were detected with the Basic level.

- **Strict** – Treats almost all geometry/topology issues as errors. In addition to the Basic errors, it will find faces that have serious topological or geometric errors at the face, edge, or vertex level that do not facet.
- **Comprehensive**: In addition to the **Basic** and **Strict** checks, it also looks for face-face intersections.

> **Note:** Customers who need to perform geometric modeling on imported objects or to run geometric parametric studies involving such objects are urged to use the **Strict** level and resolve all errors to make sure that geometric operators do not fail unexpectedly.

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** in this dialog box 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

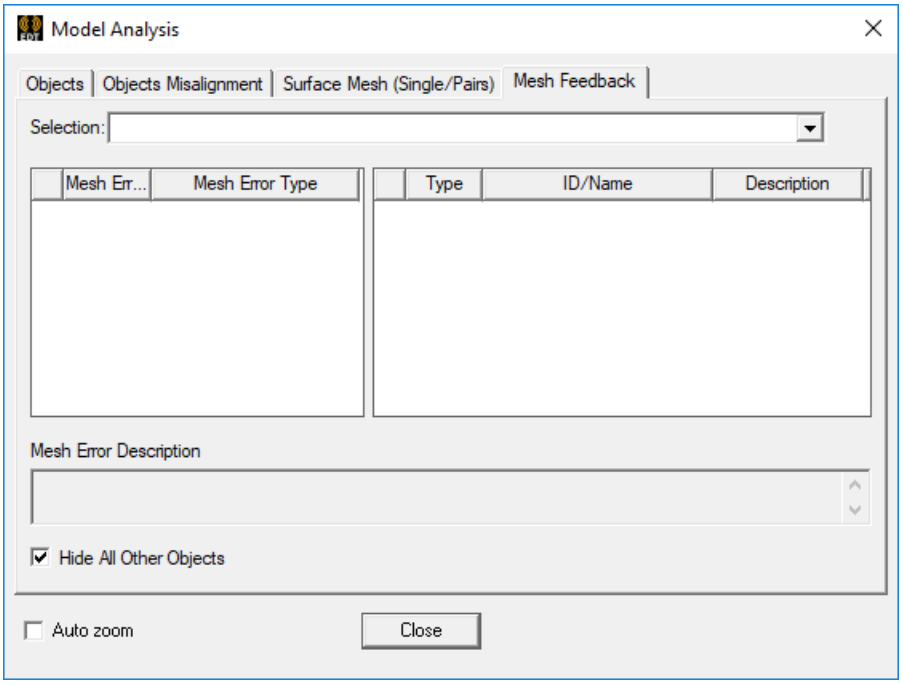
Model Analysis Dialog Box

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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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 that 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 that 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.
   - **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 box.
   - **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 box 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

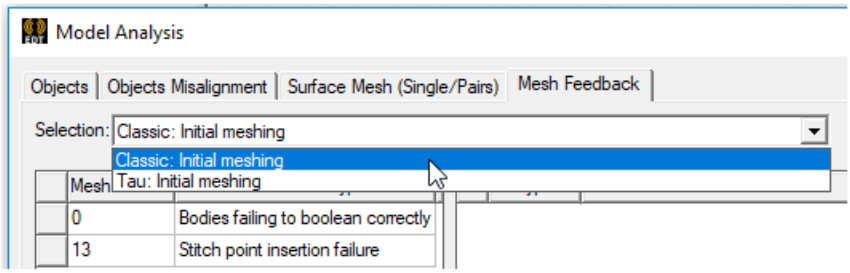
2. The results table contains the following information:
   - **Object** – column listing object name or a pair of object names.
   - **Last Analysis Status** – column stating the meshing status of the object or object pair.
     - Mesh Success
     - Mesh Failure
   - **Error Type** – this column gives 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 list lets you select which to view:

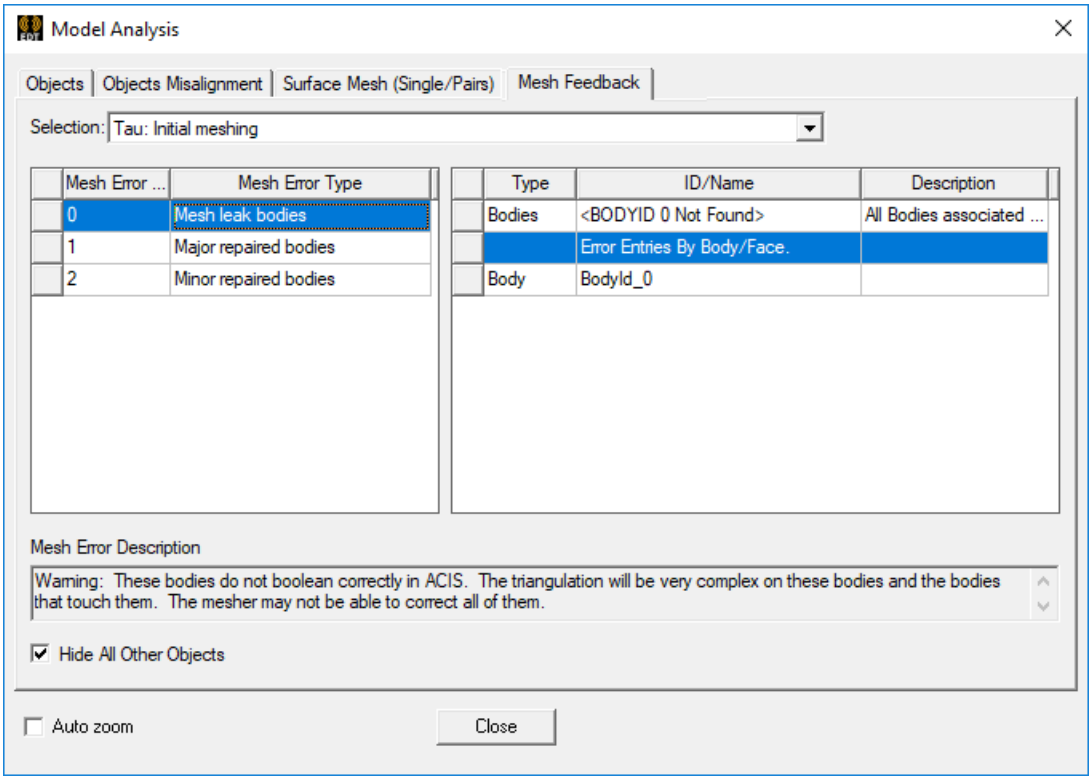


The left-side table has columns for Mesh Error Serial ID and Error Type.

- **Mesh Error Serial ID** – the 0 item is the body with which the subsequently listed entries are associated.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Error Type** – this column gives the category of error that caused the mesh failure, for example, Non-Manifold Edge, or Point Insertion Failure.



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.

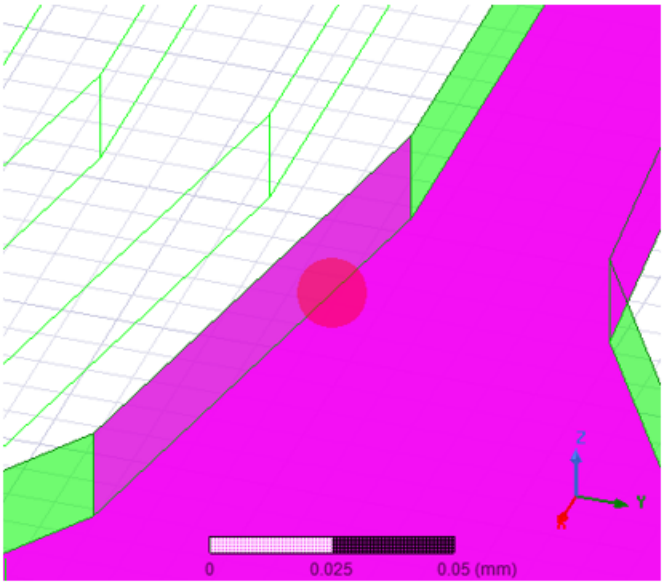
Display options include

- **Hide All Other Objects** – hides objects in the Modeler window not associated with the selected Bodies, Faces, or Triangles.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.



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

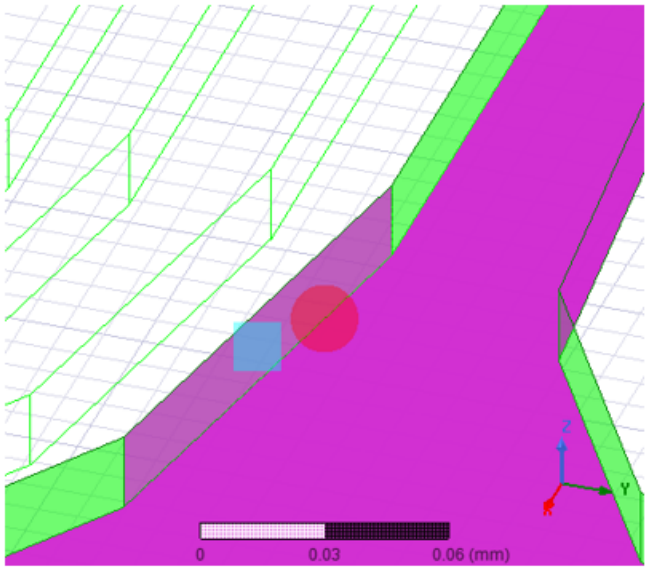
As shown above the mesh errors are marked by ‘Red Circles’. The following table describes all such markers:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Mesh errors
	Mesh warnings
	Mesher information
	Selected Faces

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.

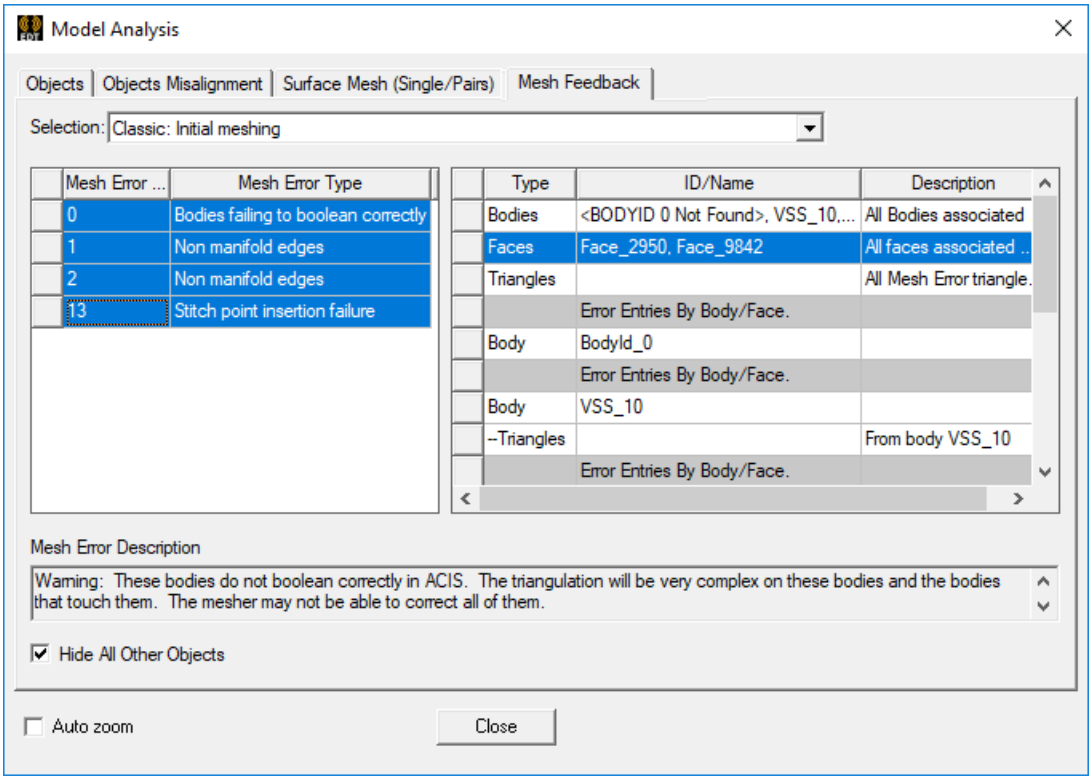
Face Selection

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

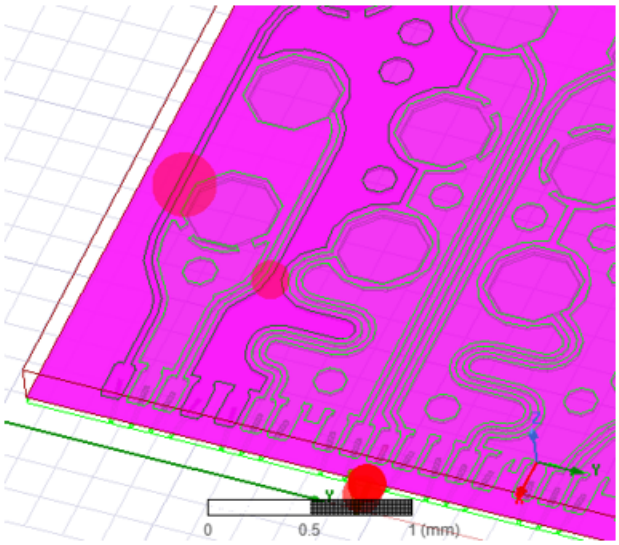
When the all faces row or individual face rows are selected, the Modeler selects the face(s). This enables the face related commands in the Modeler window. Also when you select faces this way, it makes the corresponding bodies of the selected faces visible, even when 'Hide all other objects' is enabled. 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



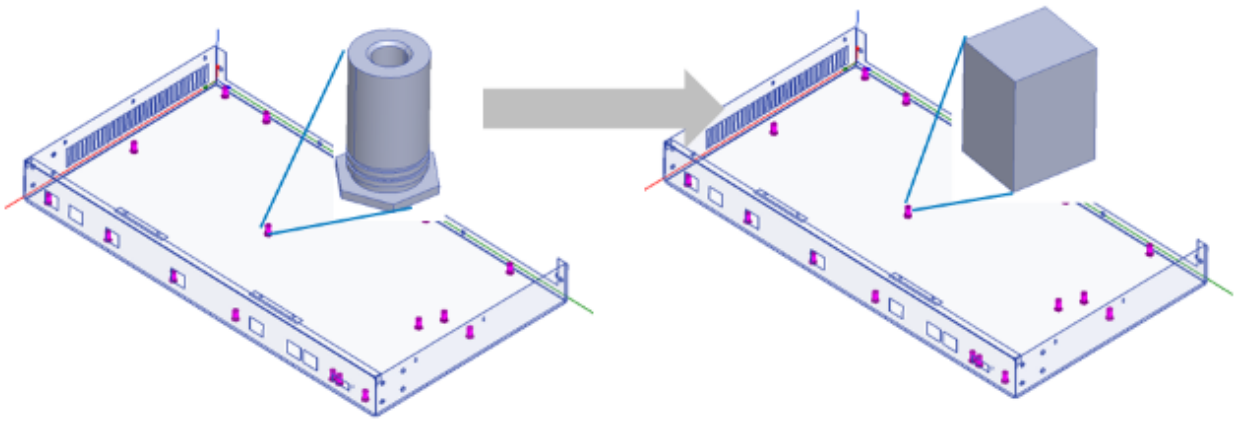
With **Auto Zoom** enabled, the view fits to all the errors, providing an overview of all the mesh errors, as shown in above figure.

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.

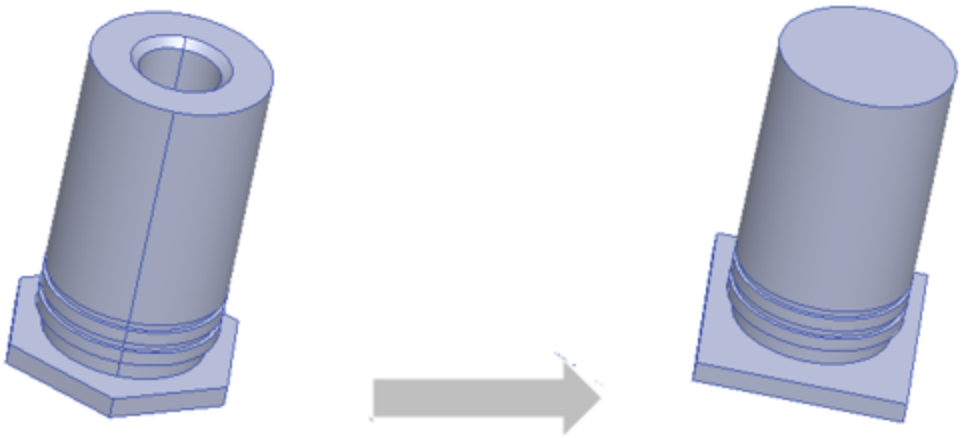
Simplify Command

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Bounding Box Simplification



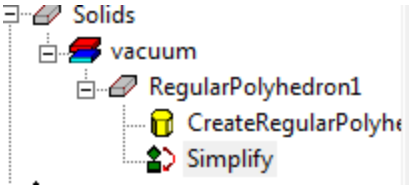
Primitive Fit Simplification

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Polygon Fit Simplification

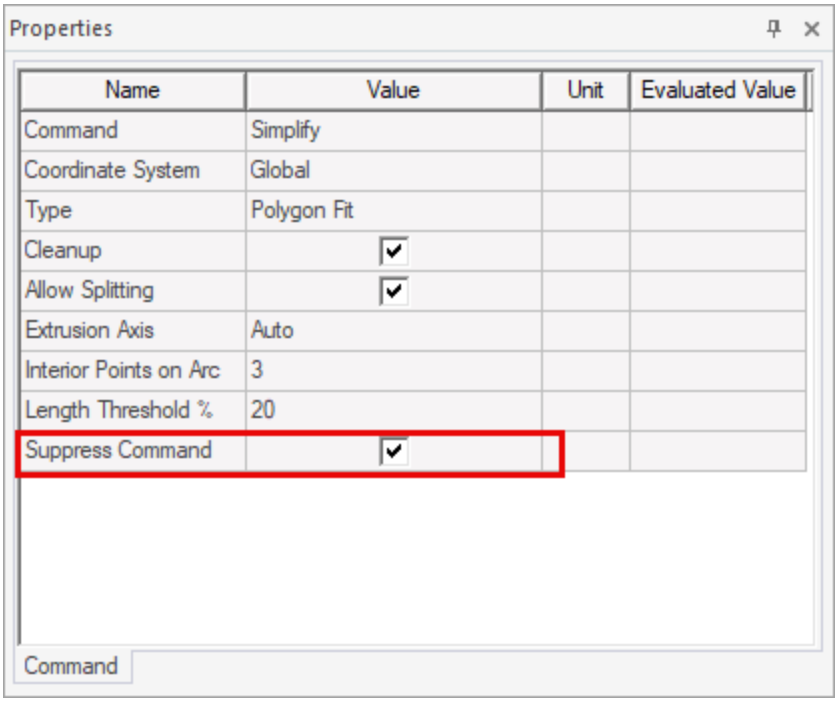
The History tree shows the **Simplify** command as being applied to the object.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

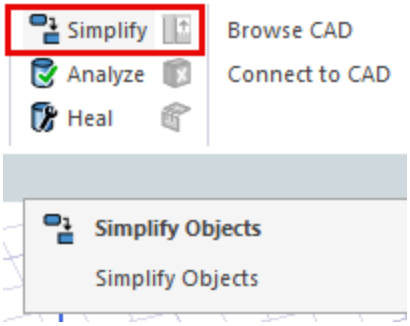
Simplify command type. None of the parameters of simplify operation accept variables.



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**.



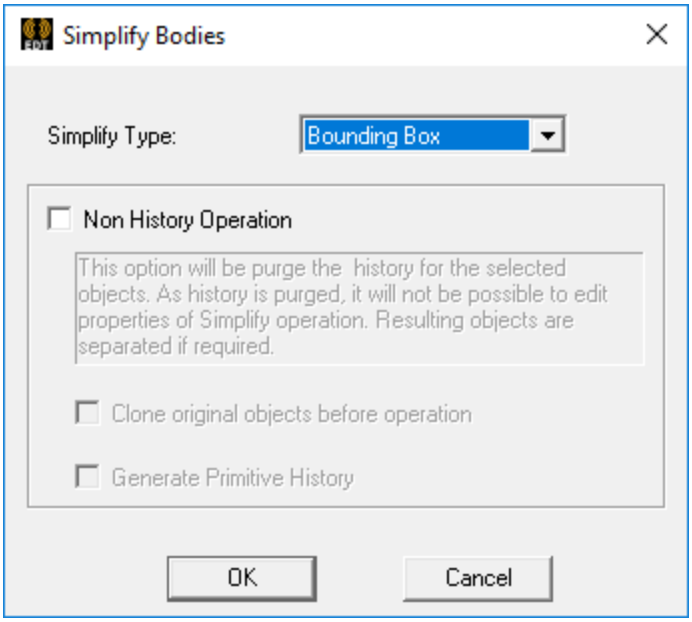
   This opens the **Simplify Bodies** dialog box. A drop-down menu lets you select the degree of simplification, from Bounding Box, Primitive Fit, or Polygon Fit.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

History Operation options, see below.

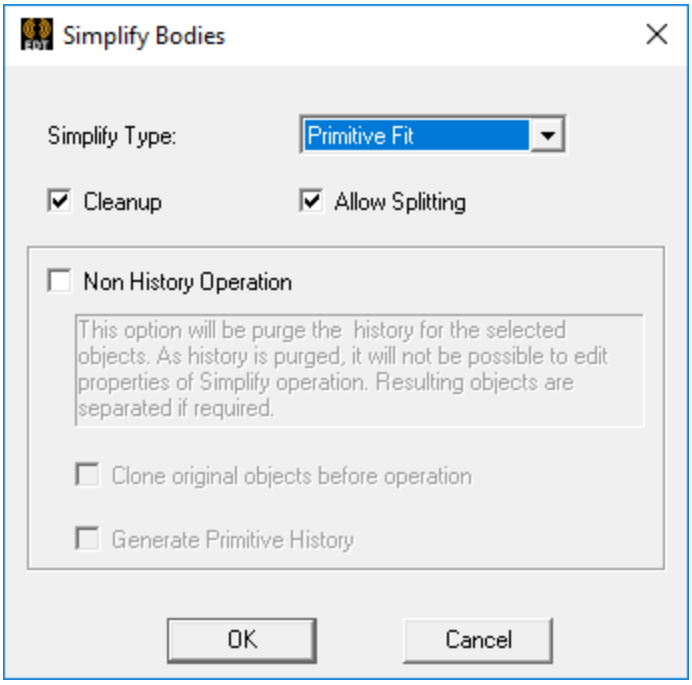


Primitive Fit Simplification

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 2026 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:

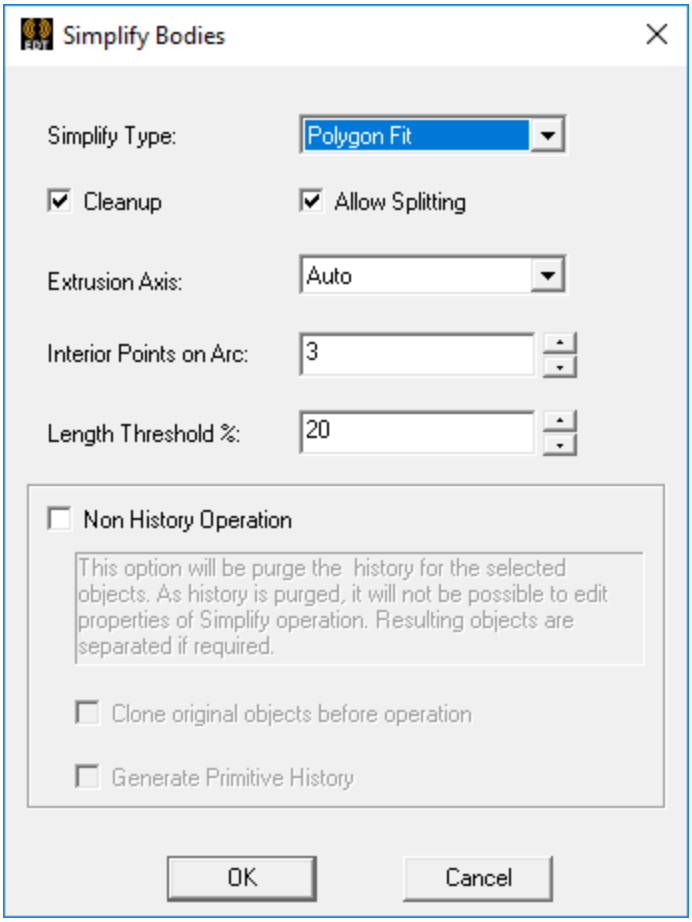
- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object 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 2026 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 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

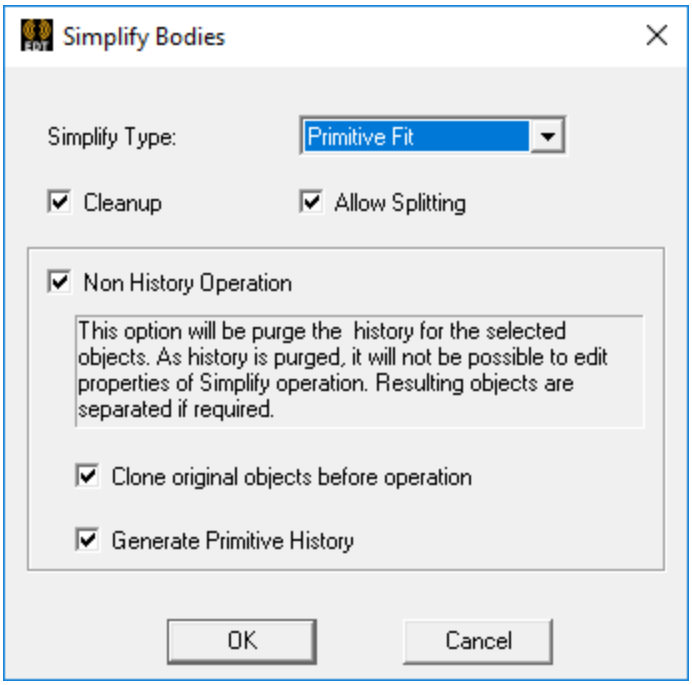
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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

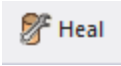
The **Heal** command provides a way to correct geometric violations and to remove specific kinds of small features. When models are imported, two types of errors can occur - geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors 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.

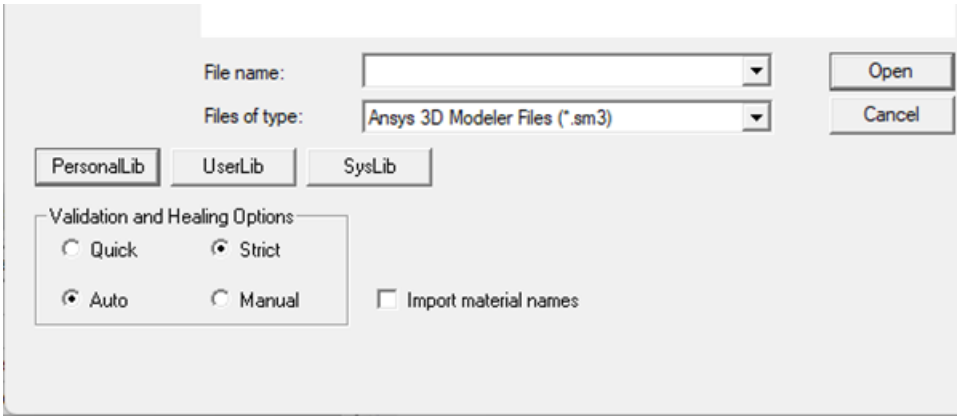
  

- Some formats permit healing during **Modeler** > **Import**. These are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 3D Modeler file (*.sm3),
- SAT file (*.sat),
- 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 Ansys Electronic Desktop 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 box 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.

Ansys Electromagnetics Suite 2026 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

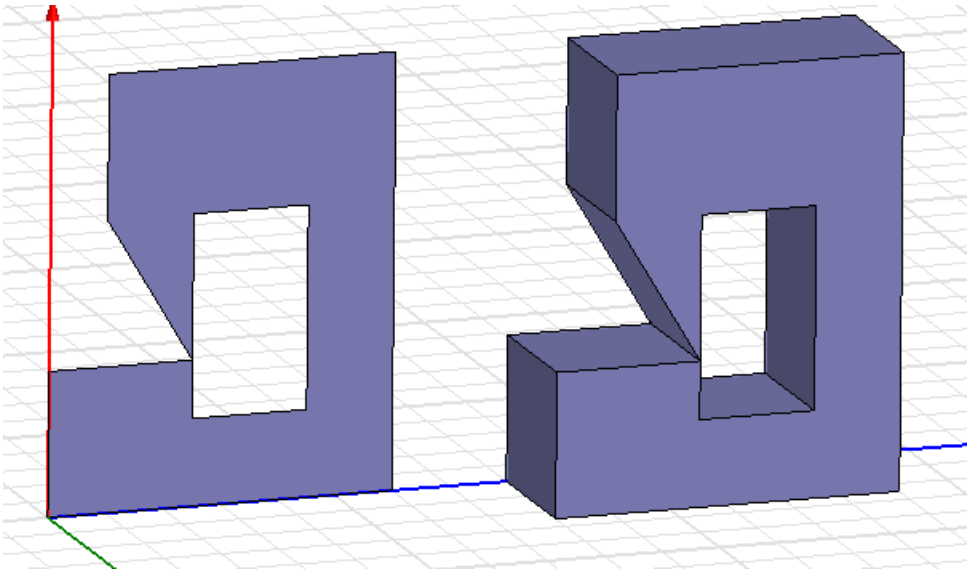
The **Import File** dialog box contains shows different Validation and Healing options for the seven formats listed here. The Quick option allows you to import these formats without invoking the Ansys Electronic Desktop healing.

Format	**Options Available**			
3D Modeler file (*.sm3),	Quick	Strict	Auto or manual	
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	

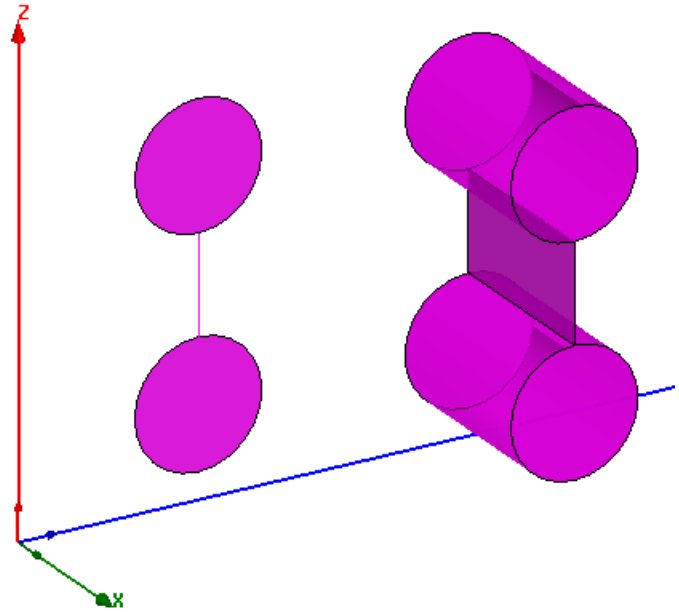
Healing Non-Manifold Objects

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 2026 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.



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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 box 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**.
     - **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remove Small Entity Options:

- **Small Edges, Length Less Than**, less than a specified value.
- **Small Faces Area 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.

   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**.

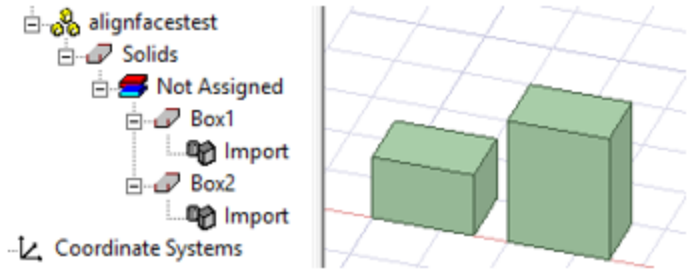
3. Click **OK**.

   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.

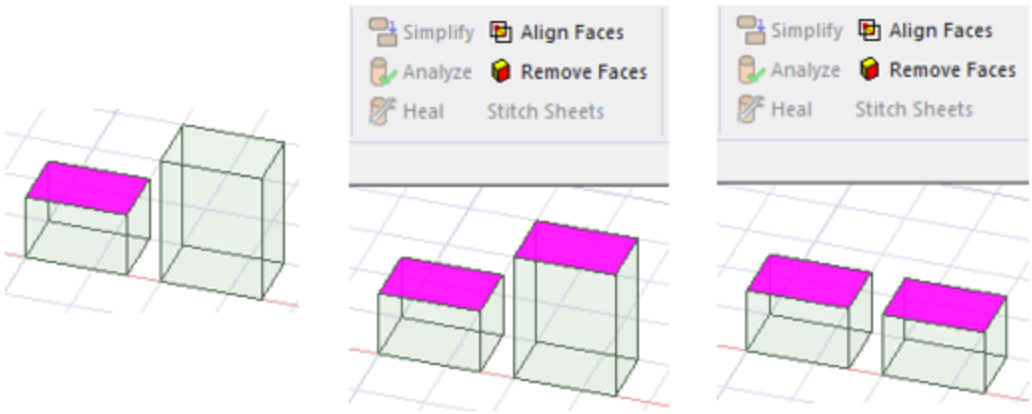
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

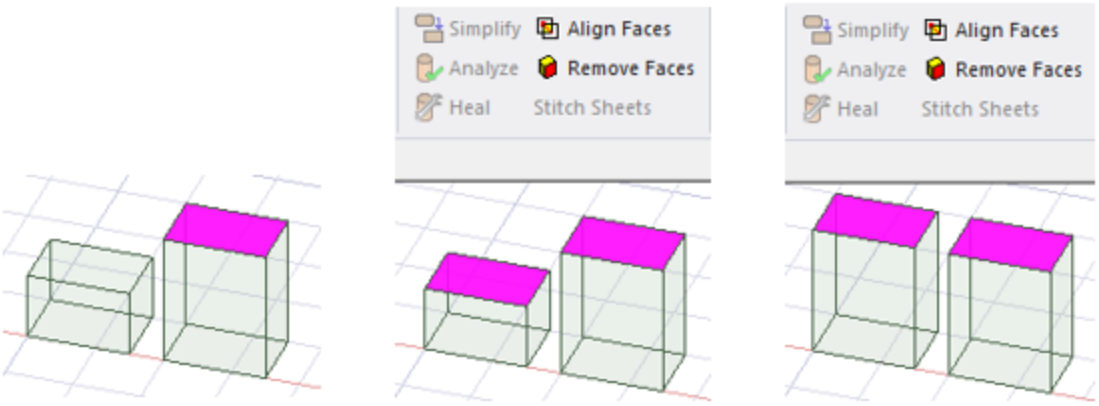


The **Model** ribbon also includes an **Align Faces** icon, which is enabled when you make appropriate face selection. The first selected face defines the plane of alignment for subsequent selection, that is, tool faces to blank faces.

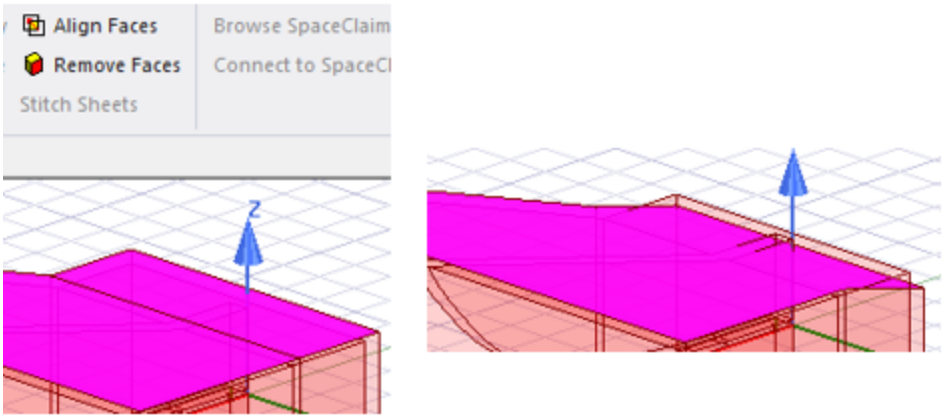


Selecting faces in a different order changes the alignment.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

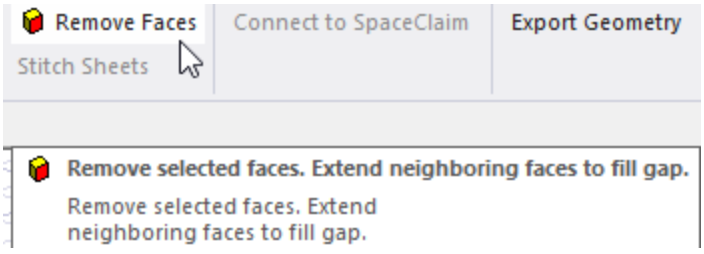


The selected faces can touch.



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 2026 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.

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 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 **Icepak 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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:

```
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 **[*PersonalLibPat*h]personallib/UserDefinedPrimitives** directory.
10. To view your primitives, click **Draw** > **User Defined Primitive** > **Update Menu** and then click **Draw** > **User Defined Primitive** > **[*PrimitiveToDraw*]**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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.

To create a UDP, see Creating 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.

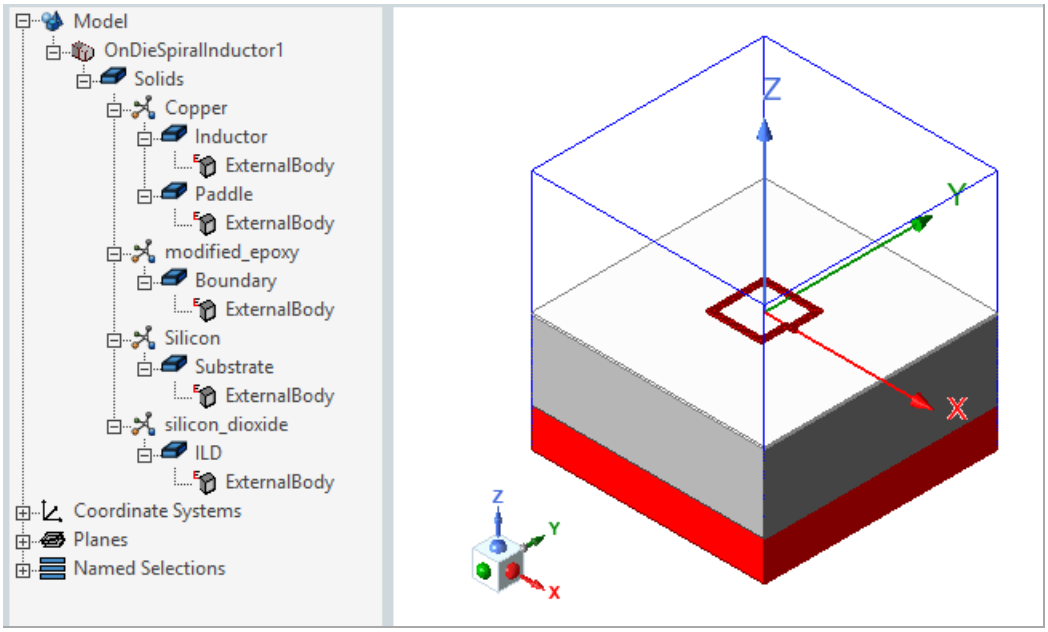
User-Defined Model (UDM)

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.

- A UDM includes part attributes (like name, color etc) and material assignment.
- A UDM can also have external coordinate systems and corresponding planes.
- A UDM parts can be parameterized and manipulated in Ansys EM modeler just like any

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

other part.



A UDM can either represent static geometry models or dynamic links to models of external geometry editors. It is used for supporting CAD integration in WorkBench and the Ansys Electronics Desktop CAD Link functionality.

A UDM uses same plugin technology as a User-Defined Part (UDP).

See the following sections:

UDM compared to User-Defined Primitives

Insert UDM Command on Draw Menu

UDM Properties

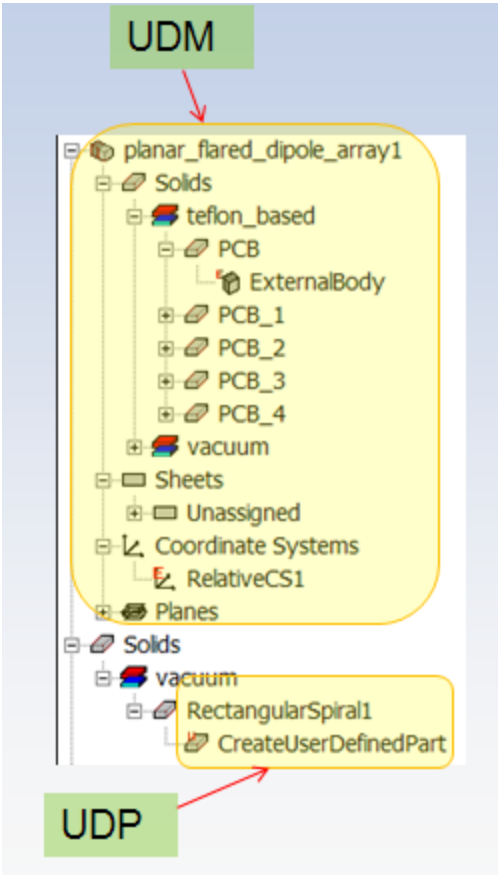
CAD Link

See "Creating User-Defined Primitives and User-Defined Models in Python Scripts" in the **Icepak Scripting Guide** for information.

UDM Compared to User-Defined Primitives

User-Defined Models (UDM) resemble User-Defined Primitives (UDP):

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Ansys EM products can be extended by users through new UDMs.
- UDM plugins are discovered by searching standard directory paths.
- Plugins for s static UDM can build a 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 a UDP:

- A UDM provides multiple Parts/CS/*etc*.

  A UDP provides primitive operation only for a single Part.

- A UDM provides part attributes and material assignment.
- A UDP does not define part attributes or material.

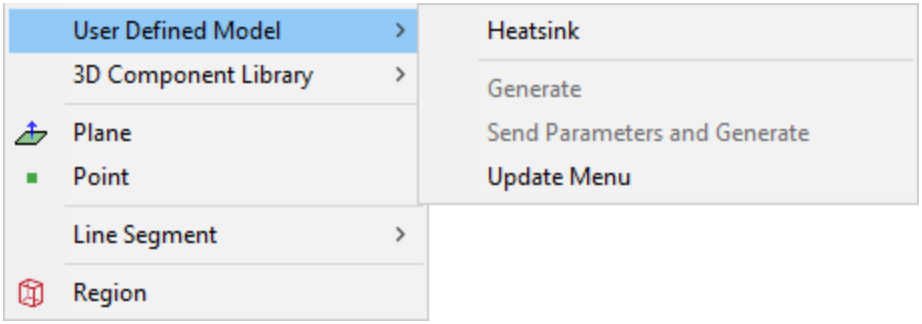
The UDM Properties window have four tabs - Definitions, Parameters, Options and Info.

Ansys Electromagnetics Suite 2026 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

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.

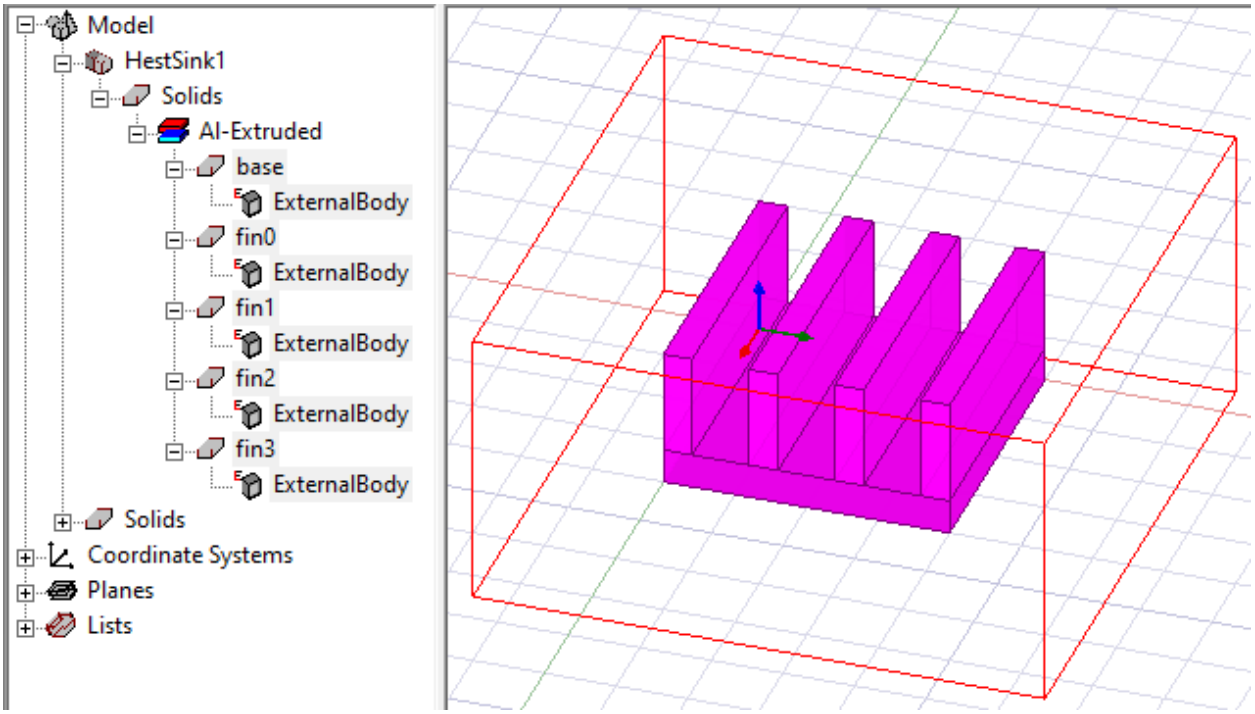


UDM - Icepak Heatsink

From the **Draw** menu, select **User Defined Model** > **Heatsink** to create and insert a heat sink into your Icepak design.

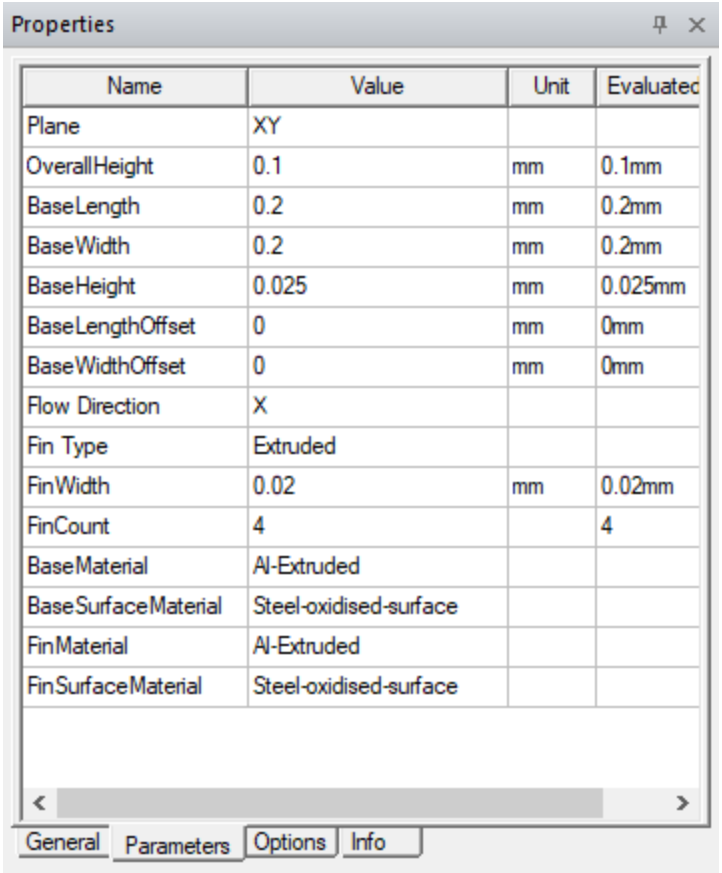
> **Note:** For more information about heatsink properties and how to insert a heatsink 3D native component, see Creating Heatsinks.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



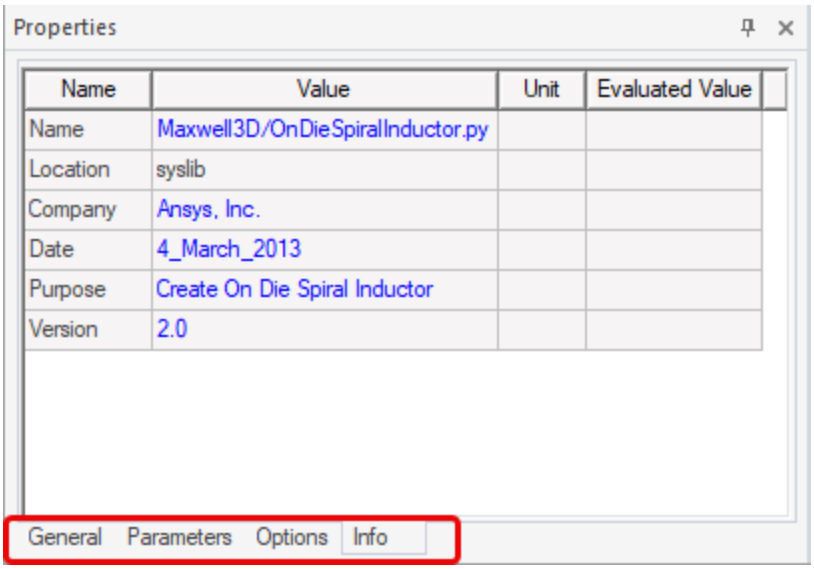
To edit the heatsink's properties, select the heatsink in the History tree and enter new values in the **Value** column of the **Properties** pane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



UDM Properties

UDM Properties have four tabs - General, Parameters, Options and Info.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **General** tab has

- UDM name
- Coordinate system used to position UDM
- May have external reference to file

The **Parameters** tab has the defined parameters and their values.

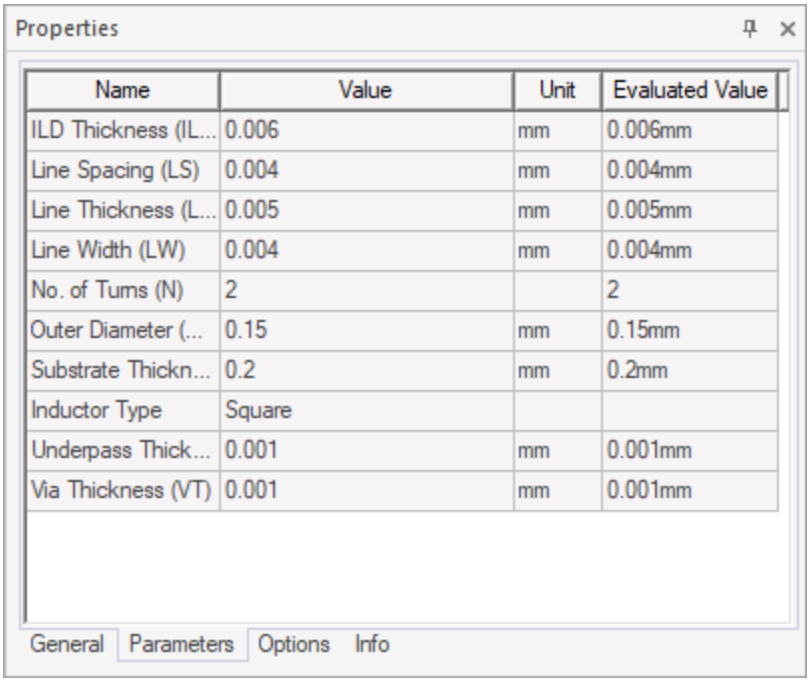
The **Info** tab has

- UDM dll or .py name
- .dll or .py location
- Company name
- Creation date
- Purpose
- Version

The **Options** tab may have options; but may be blank if no options are defined.

UDM Parameters

UDM geometry can be manipulated through its parameters:



These parameters

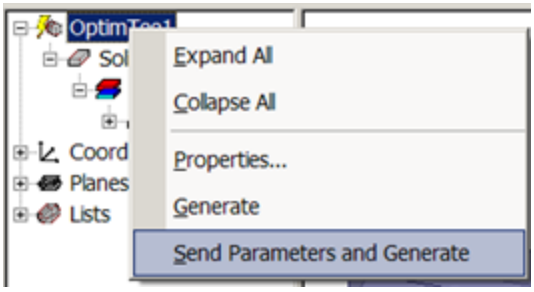
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- can be mapped to design or project variables for animation and parametric analysis;
- IDs are persisted (allowing to retain boundaries) during parameter edits.

The 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:

- 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)

The following part attributes can be modified for UDM parts:

- Model/Non Model flag
- Solve Inside flag
- Part orientation

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** > **CAD Link**) to import geometry into AEDT.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Modeler CAD Link

The 3D Modeler CAD Link allows you to create a User-Defined Model (UDM) from the geometry model in an Ansys Discovery project or in a supported third-party CAD project. In addition to Ansys Discovery, Ansys Electronics Desktop can link to models from NX, Solid Edge, Creo Parametric, Creo Elements/Direct, AutoCAD, and Inventor projects. The CAD link is only available on Windows.

For a list of supported CAD releases, see Geometry Interface Support.

> **Note:**
>
> The CAD link can only be used with Ansys Discovery if it is the same release version as Ansys Electronics Desktop and if both are installed on the same machine.

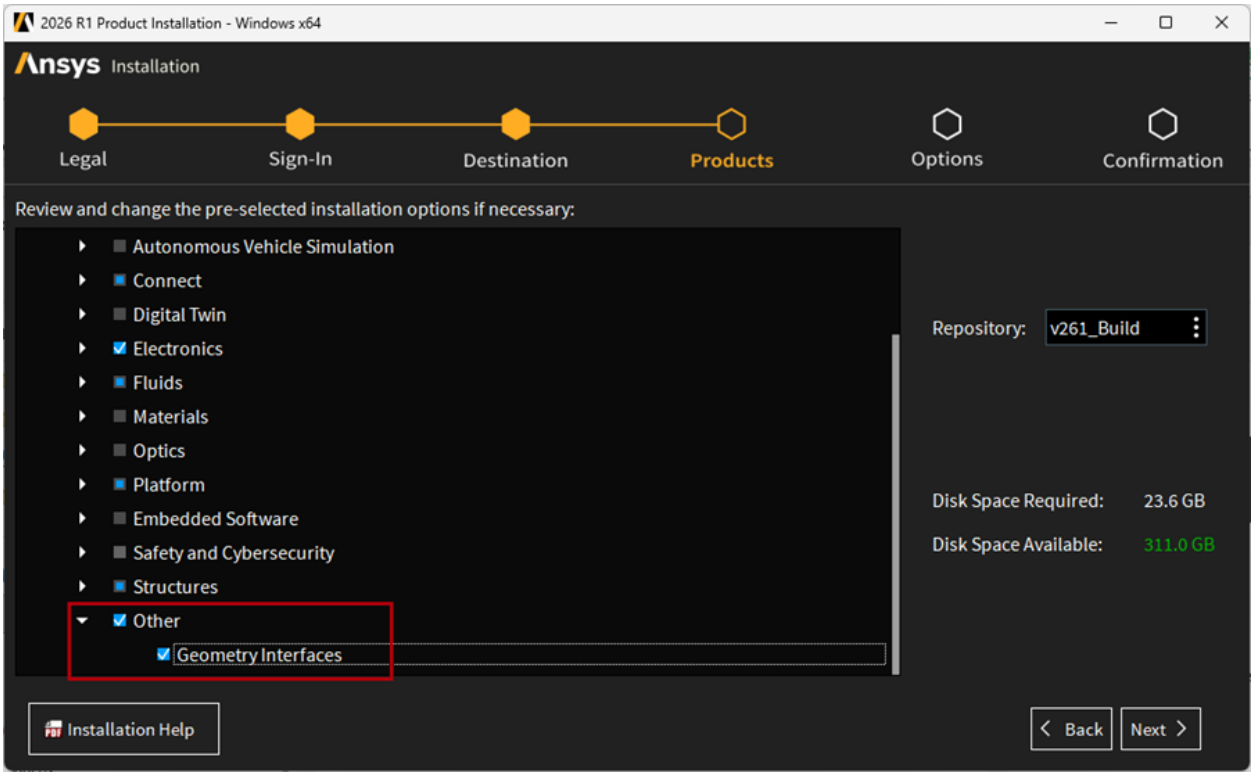
CAD Link provides following functionality:

- Browse and select CAD project files to import geometry directly.
- Import geometry by connecting to an active CAD session.
- The CAD geometry is imported as a UDM along with geometry parameters, rendering attributes and material assignments. For a Discovery model, the geometry parameters are imported with their units. Object, material, and parameter names with non-ASCII characters are not supported.
- If the design supports lightweight bodies, the normal "unfaceted" parts in the CAD model can be imported and faceted in Ansys Electronics Desktop with the specified facet level. For CAD software other than Discovery, the faceted parts cannot be imported. With a Discovery link, the faceted bodies can be imported directly, ignoring the specified facet level.
- CAD assemblies can be preserved if **Create Groups for Sub assembly** is checked in the 3D Modeler Options: CAD Link Options window.
- A CAD document referenced by a CAD link is included in the project archive automatically.
- You can break a CAD link. In this case, all the parts in the UDM are converted to imported objects.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

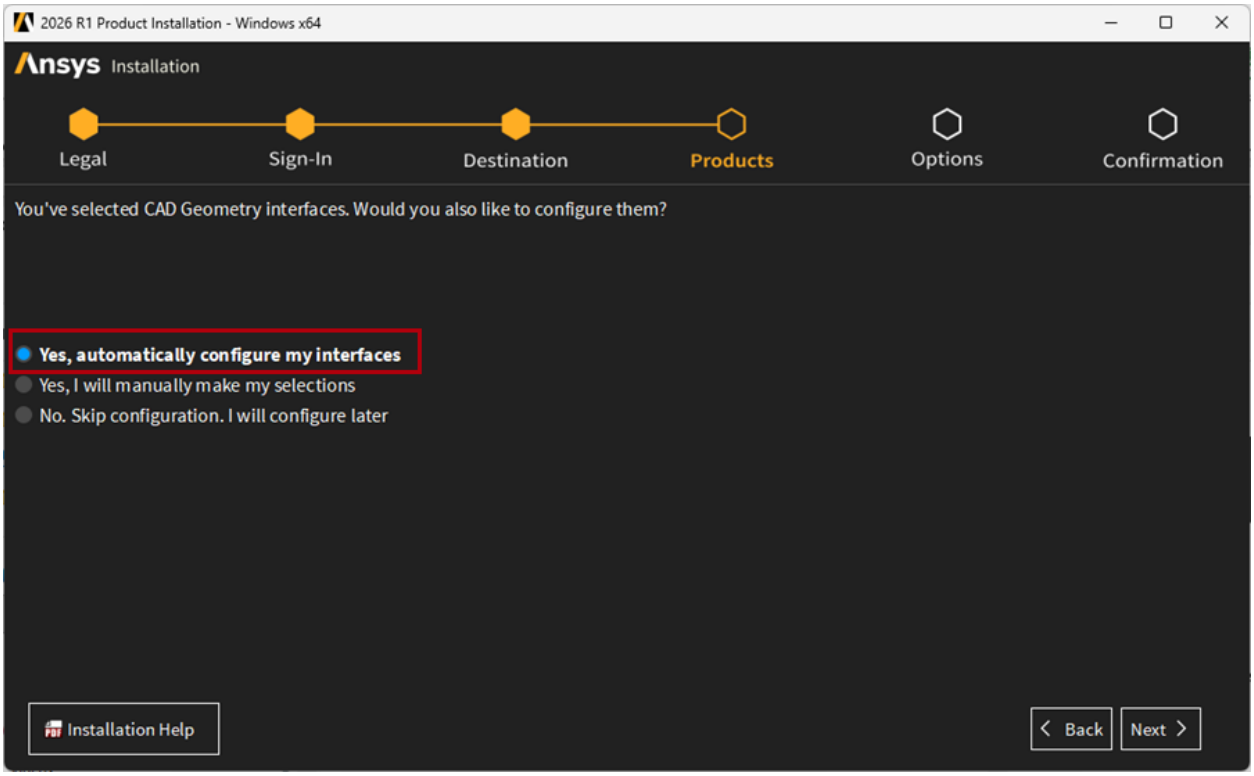
Configuring Desktop for CAD Interfaces

To use the CAD Link functionality, the Geometry Interfaces need to be installed and configured. The Geometry Interfaces can be found in the installer:



During installing, the Geometry Interfaces can be configured automatically:

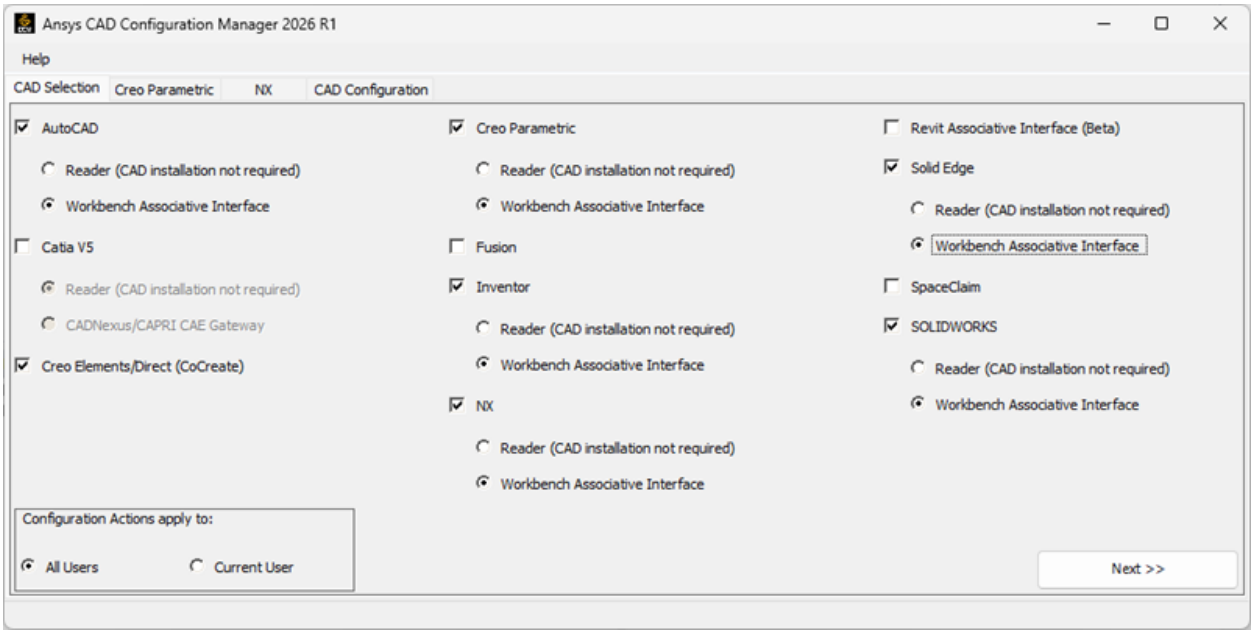
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you did not configure the geometry interfaces during the install, you can use CAD Configuration Manager, which is accessible from the Ansys 2026 R1 Start menu.

Check each CAD program you will use, then select the **Workbench Associative Interface** option. Go to the **CAD Configuration tab**, and click **Configure Selected CAD Interfaces**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



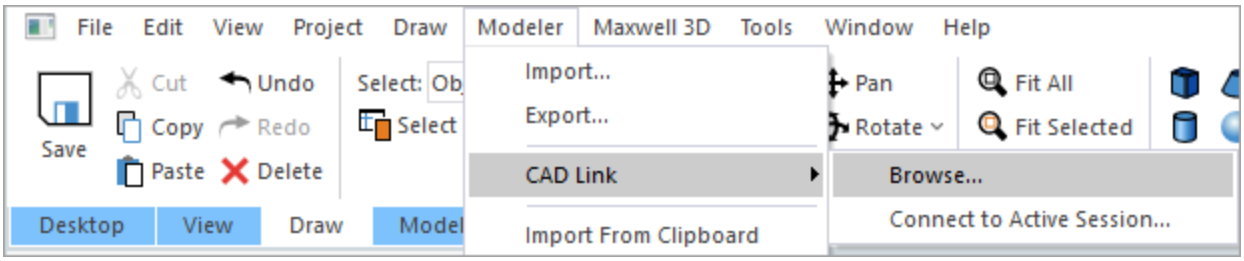
Note:

For more information on this functionality, see Using the CAD Configuration Manager.

Link CAD Geometry to Ansys Electronics Desktop

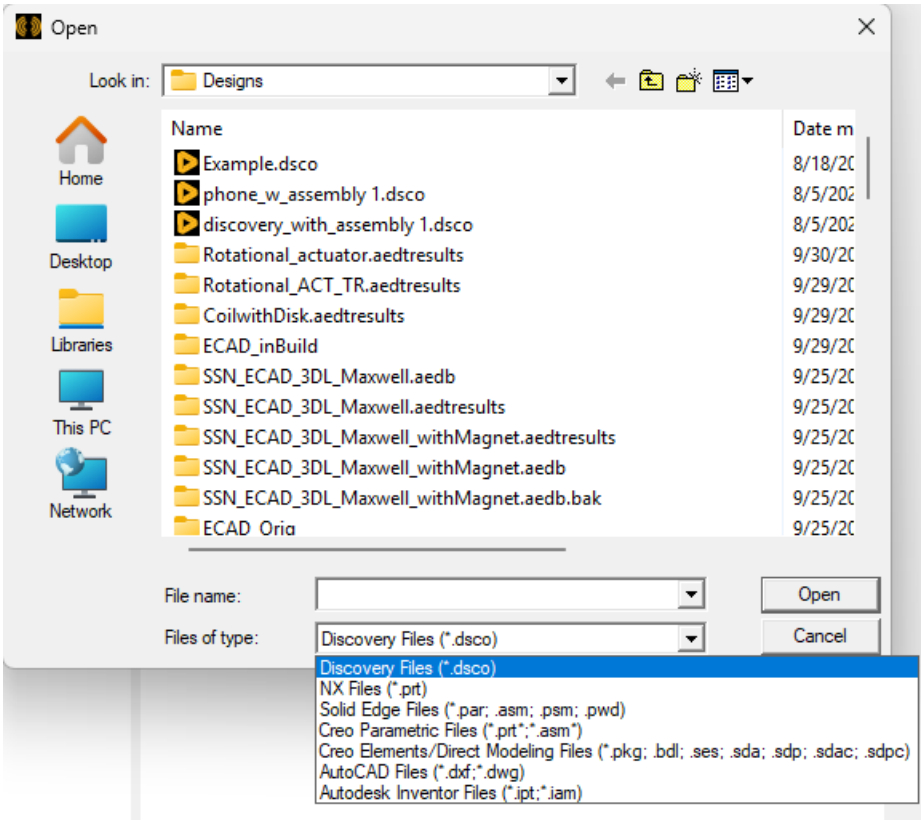
To select a CAD file:

1. Select **Modeler** > **CAD Link** > **Browse** or click the **Browse CAD** icon on the **Model** ribbon.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Navigate to and select a project to load into the Modeler window.



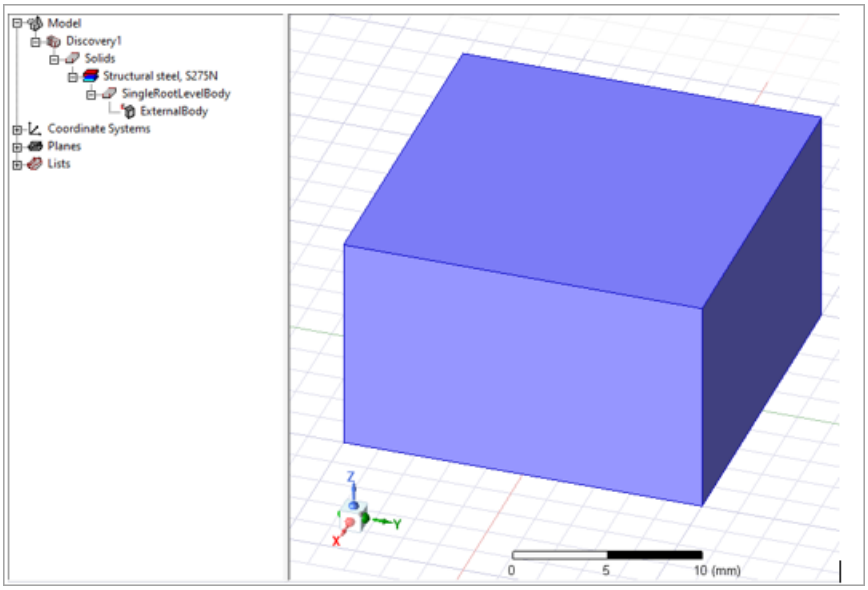
> **Important:**
>
> The **Files of Type** field is not only for filtering the files, but also for specifying which CAD interface will be used. For the file extensions used by multiple CADs, for example,“.prt” is used by NX and Creo Parametric, make sure to select the correct file type according to the CAD interface used to generate that file.

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 the name of the CAD software. In the example below, a Discovery model was linked, so the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

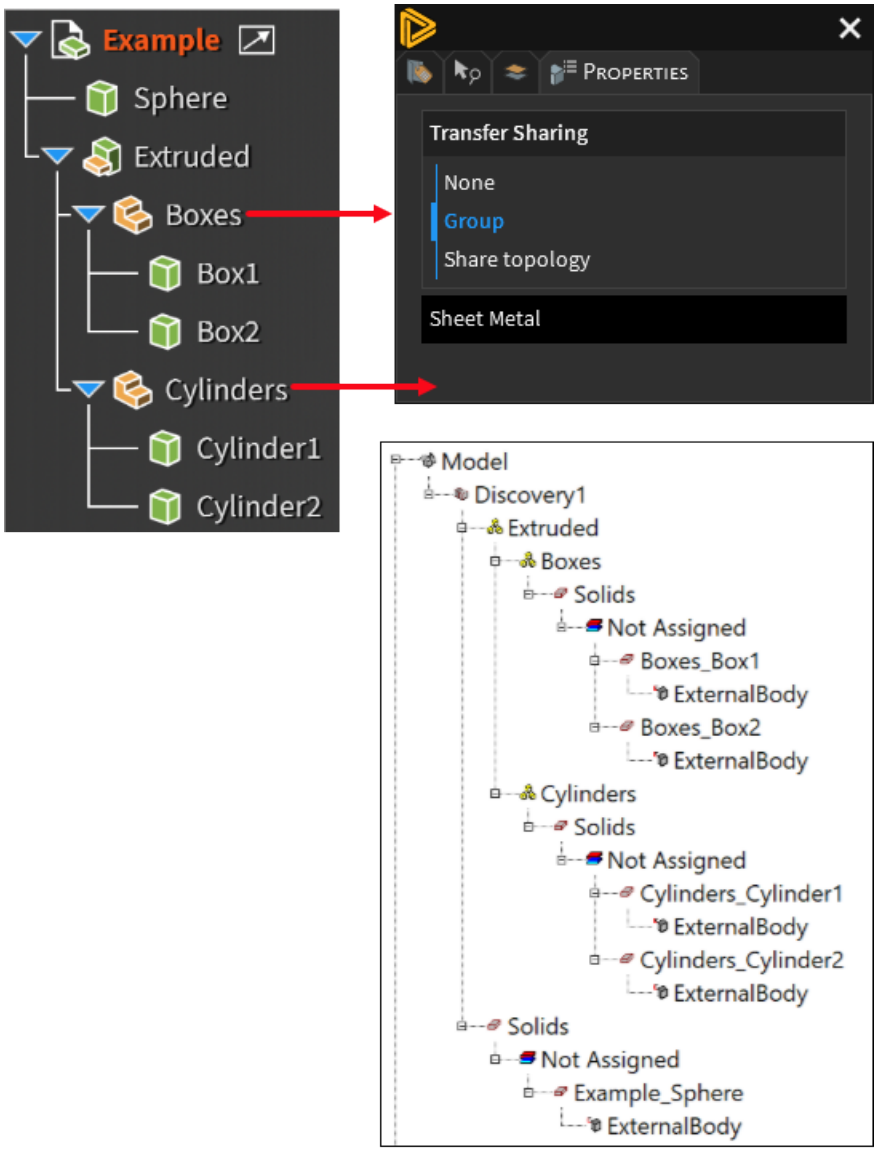
UDM is labeled **Discovery1**.



If there is an active CAD session that is using the selected document, the created UDM will be linked to the active CAD session. If there is not one, a new non-graphical CAD session will be used, which makes the creating and refreshing slow. Afterward, if an active CAD session is launched and opens the document, the UDM will be linked to the active session.

To preserve assemblies from the CAD software as groups in Modeler, the **Create Groups for Sub assembly** must checked in the **3D Modeler** > **CAD Link** options:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

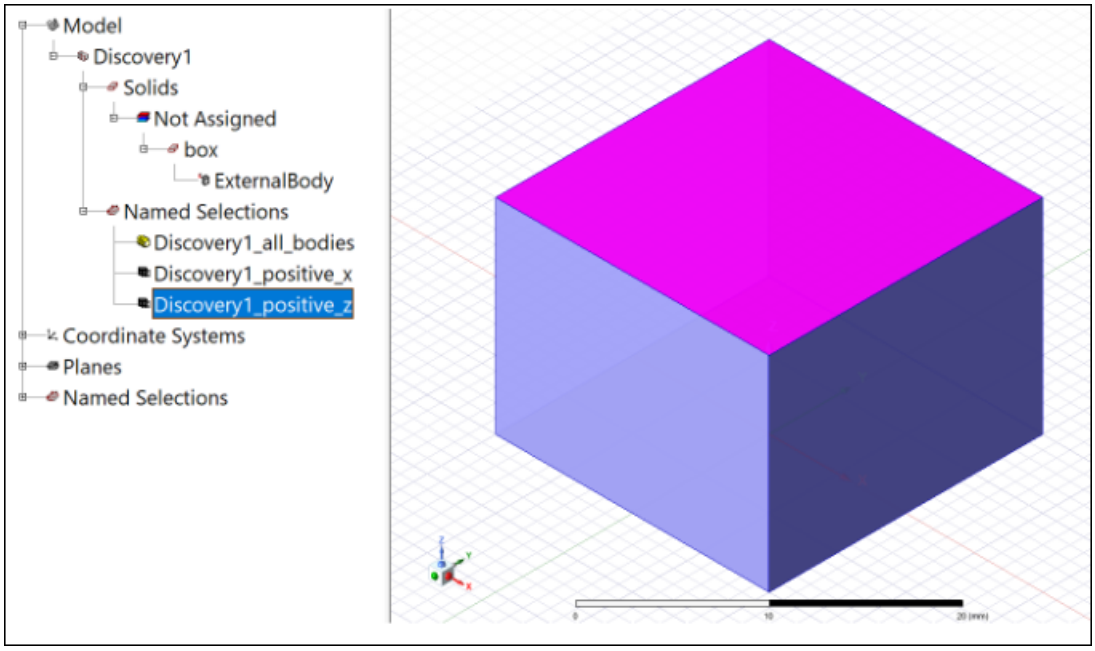


For Discovery, multi-body parts should have the **Transfer Sharing** property set to **Group** from within Discovery.

Note that group hierarchy is read-only, but individual parts of the CAD model can be duplicated or moved by right clicking on the part and selecting **Edit** > **Duplicate** or **Edit** > **Arrange** > **Move**, respectively.

Note that if the **Import Named Selections** option is checked in the **Tools** > **Options** > **General Options** > **3D Modeler** > **CAD Link** window, named selections created in the CAD software will be listed in the **Named Selections** branch of the history tree:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



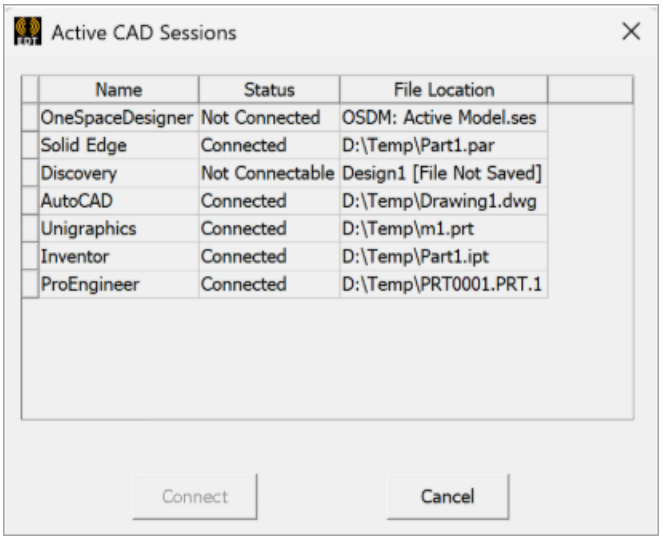
To synchronize imported named selections with the CAD software, right-click on the UDM (*Discovery1* in the preceding example), and select **Generate** or **Send Parameters and Generate**. Within Ansys Electronic Desktop, the imported named selections cannot be directly modified or deleted without deleting the whole UDM.

Connecting to an Active CAD Session

If one or more CAD sessions are running on the machine, CAD Link can connect to them to import geometry.

When you select **Modeler** > **CAD Link** > **Connect to Active Session** or **Connect to CAD** in the Model ribbon, the Active CAD Sessions dialog box appears:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



It shows all the CAD sessions that are currently running and detected by Ansys Electronics Desktop. The CAD software names that appear in this dialog box correspond to the names used in Ansys Workbench:

CAD Software	Name shown in the dialog box
NX	Unigraphics
Solid Edge	Solid Edge
Creo Parametric	ProEngineer
Creo Elements/Direct	OneSpaceDesigner
AutoCAD	AutoCAD
Inventor	Inventor

The **Not Connectable** Status indicates the CAD session does not have an active document or the document has not been saved. The exception is Creo Elements/Direct (OneSpaceDesigner), for which an active session can be connected without a saved document.

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 CAD-linked UDM.

Note the following:

- **NX** does not support opening multiple files with the same name. If the connected NX file has the same name as an opened one in NX session, close the file before refreshing the CAD link.
- **Creo Parametric** does not support opening multiple files with the same name. If the connected Creo file has the same name as an opened one in Creo session, close the file

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

before refreshing the CAD link.

- CAD links to **Creo Elements/Direct** are connected to either an active session or a saved file. The source can be switched by changing the GeometryFilePath property of a created link (see Property Window of CAD-Link UDM). From active session to file, you can just input the file path. From file to active session, input the following: “OSDM: Active Model.ses”.

Preserving IDs During Parametric Variations

When a model is imported, IDs in Ansys Electronic Desktop depend on IDs available from the source CAD software. If parameters in the CAD software are set up such that changing their values causes existing parts to be edited, then IDs in the CAD software and Ansys Electronic Desktop remain constant across parametric variations.

However, if IDs in the source CAD change during a parametric update, then Ansys Electronic Desktop cannot match object/faces across variations. Ansys Electronic Desktop might delete original objects and create new objects, causing deletion of boundary conditions and failures in parametric simulations. Such a situation can commonly occur when an embedded script is used in the CAD software, and the script clears all parts at the beginning of script.

Named selections can be used in such situations to preserve IDs of objects/faces so that parametric simulations can proceed. In such cases, use the following guidelines:

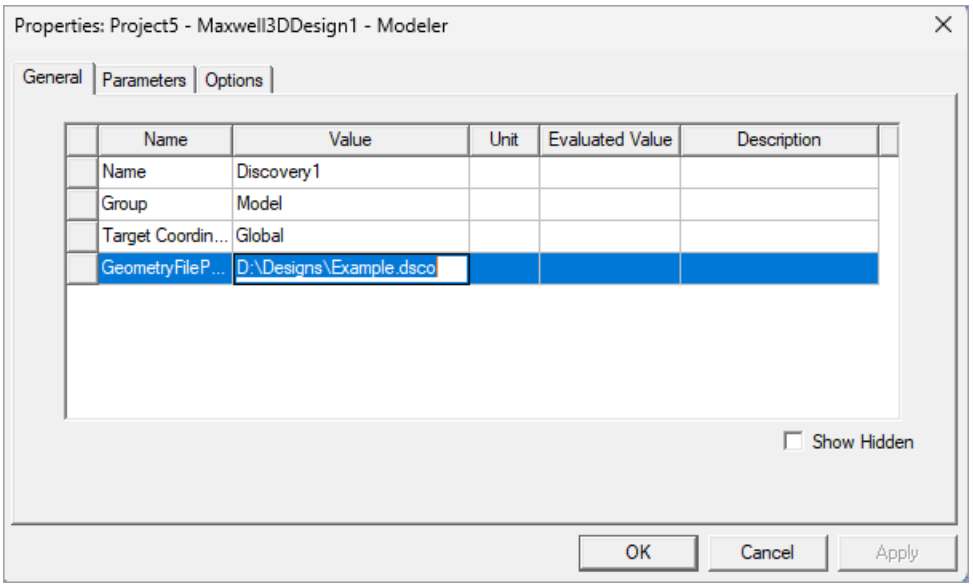
1. The total number of objects must be kept the same across parametric variations.
2. If part IDs are changing in the CAD software, users must include every part in at least one object named selection and keep the name of each part constant across parametric variations.
3. Discovery can have multiple parts with a same name. Ansys recommends assigning a unique name to every part and creating an object named selection that includes all parts in the model. If duplicate names cannot be avoided, then multiple named selections can be created so that parts with the same name are in different named selections. Ansys Electronic Desktop will match objects in a named selection by using their names and preserve IDs across parametric variations. This will preserve boundary conditions or other assignments in Ansys Electronic Desktop.
4. If a named selection has a single face, then the ID of that face in Ansys Electronic Desktop will be kept constant across parametric variations even if it changes on CAD software side. If a named selection has multiple faces and the IDs of these faces change in the CAD software, then Ansys Electronic Desktop cannot guarantee that the ID of each face in that named selection will be preserved consistently. It is possible that IDs may switch among faces in that named selection. Ansys recommends creating a separate named selection for each face on which boundary/excitation is added in Ansys Electronic Desktop.
5. If the number of faces in a named selection changes across parametric variations, then the face IDs may not remain constant.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property Window of CAD-Link UDM

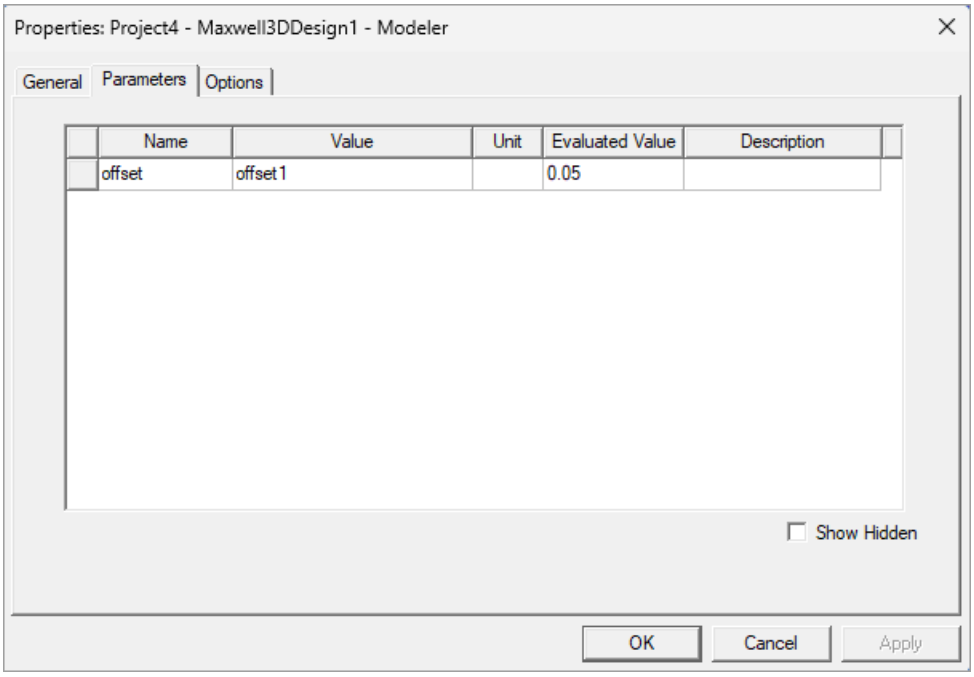
The Property window of a CAD-Link UDM has three tabs:

- The **General** tab shows general information such as CAD document path. Note that the Geometry File Path is editable; users can update the path if the original file is moved.



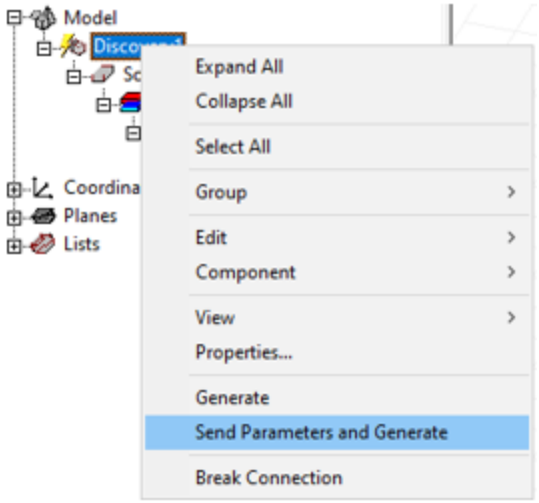
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Parameters** tab shows all parameters.



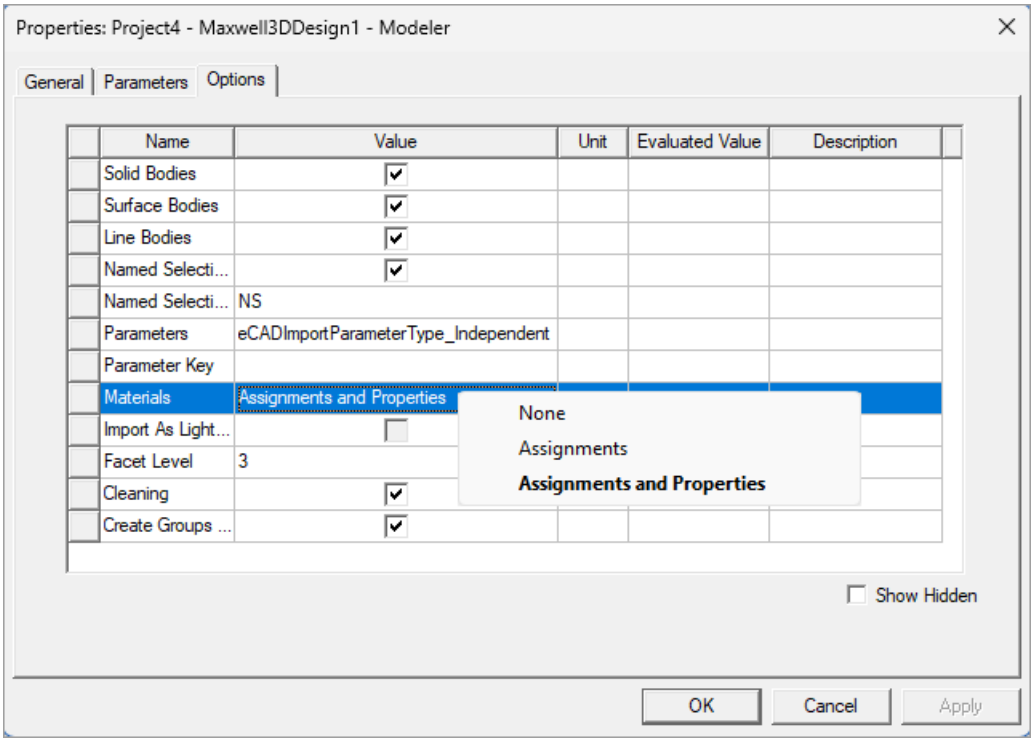
You can edit the parameters and send them back to the CAD software to get the geometry updated accordingly.

After editing the parameter, you will need to use **Send Parameters and Generate** in the context menu of the CAD-linked UDM. If there is an active CAD session linked, its model will be updated, but not saved. If there is not an active CAD session linked, the CAD document will not be updated.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Options** tab shows supported options. See 3D Modeler CAD Link Options for details on the options in this window. The example shown below is from a Discovery model.



  Note that parameter settings for an individual CAD model can be changed from this tab. After changing parameters, refresh the link by right-clicking and selecting **Generate** or **Send Parameters and Generate**.

Archive a Project Containing CAD Link

If you want to share a project with CAD link with other users, you need to archive the project, because the linked CAD file is not included in the project file. When archiving, the CAD documents corresponding to the links will be included in the project archive automatically. Once the archive file is unarchived, referenced CAD 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.

> **Important:**
>
> An archived project only includes the linked CAD file. If the file is an assembly file that has links to other part files, you need to add them manually when archiving.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Break Connection with CAD Link

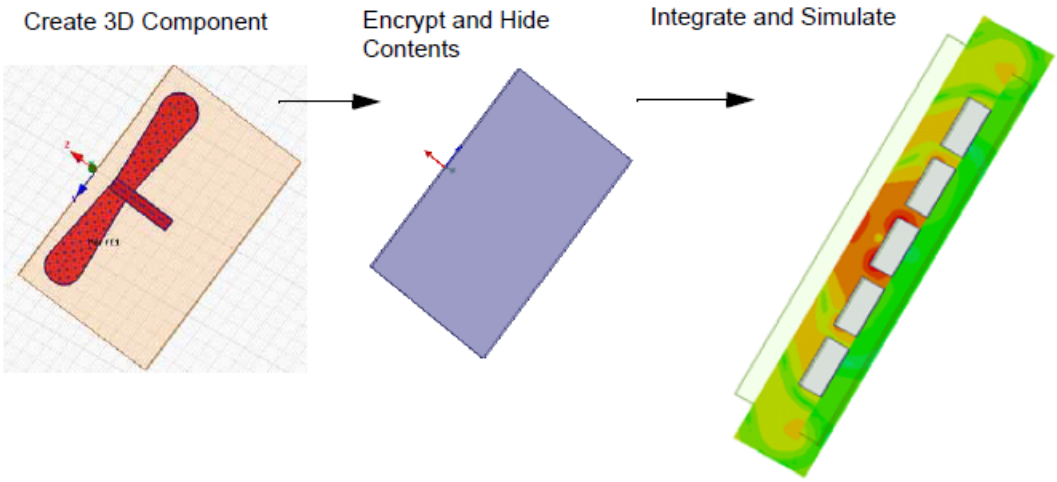
A **Break Connection** command is available in the context menu of the CAD link UDM.

- **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 the CAD model parameters.
- Attributes such as rendering attributes, materials, *etc.*, are preserved.

3D Component Library

The 3D Modeler lets you access predefined 3D component libraries and to create 3D components using:

- Geometry (objects and coordinate systems)
- Component parameter, including variables used by geometries, design data, and material properties
- 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.

The documentation includes the following sections:

Creating a 3D Component from an Existing Model

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

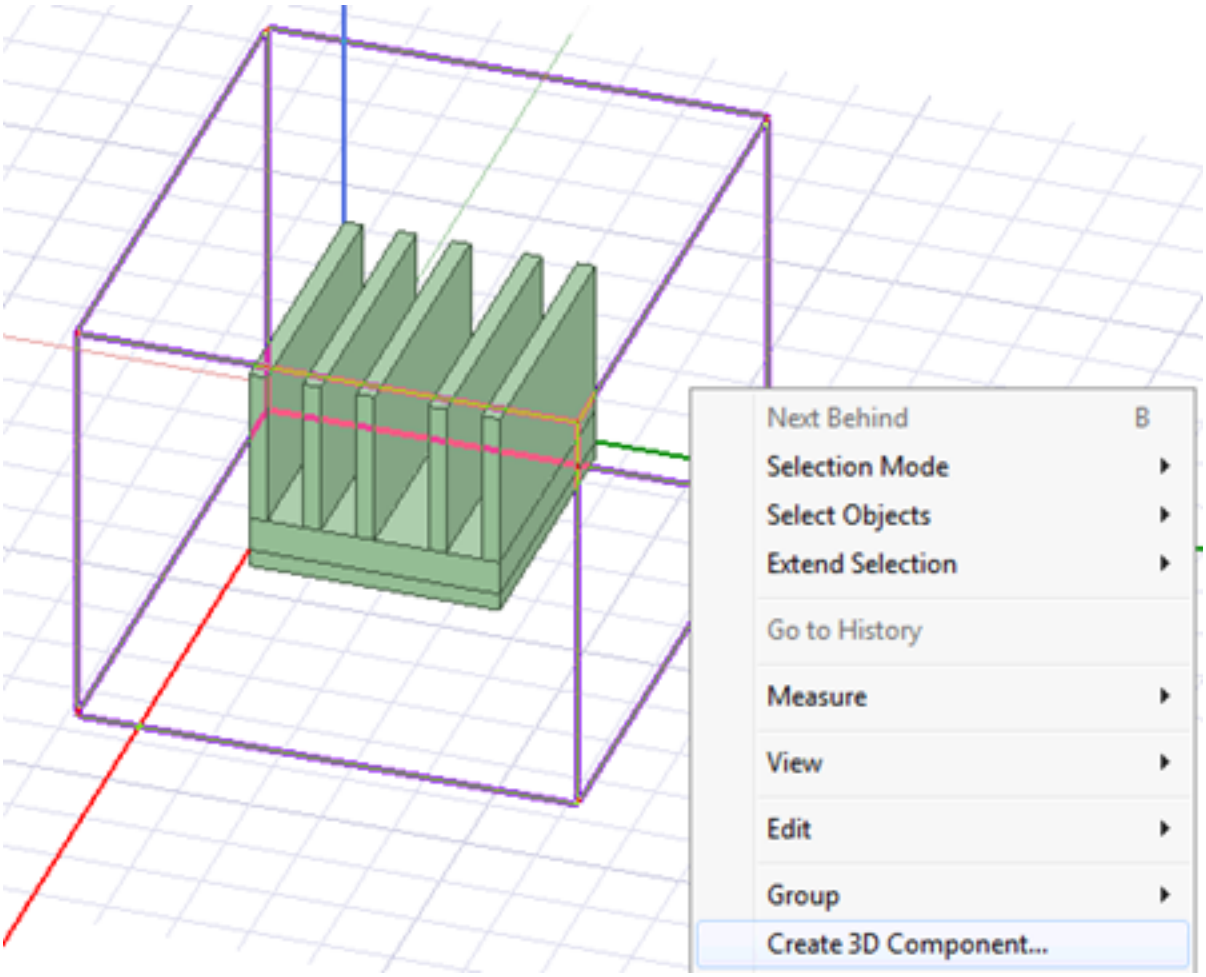
Editing a 3D Component Definition

Update 3D Component Definition

Creating a 3D Component from an Existing Model

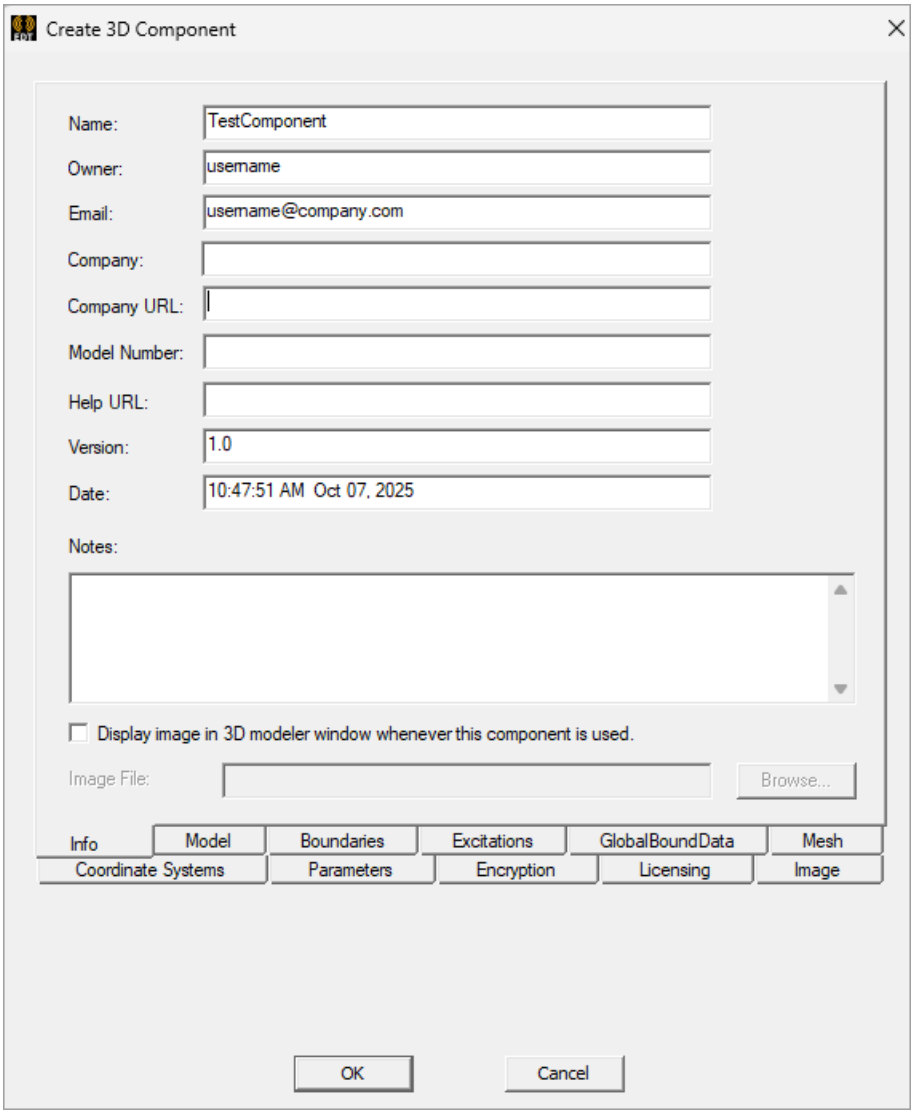
To create a 3D component, select an existing object or model in the Modeler window, right-click, and select **Create 3D Component**. You can also click **Draw** > **3D Component Library** > **Create 3D Component**. You can also go to the **Model** tab of the ribbon and select **Create 3D Component**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens the *Create 3D Component* dialog box.

Ansys Electromagnetics Suite 2026 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:

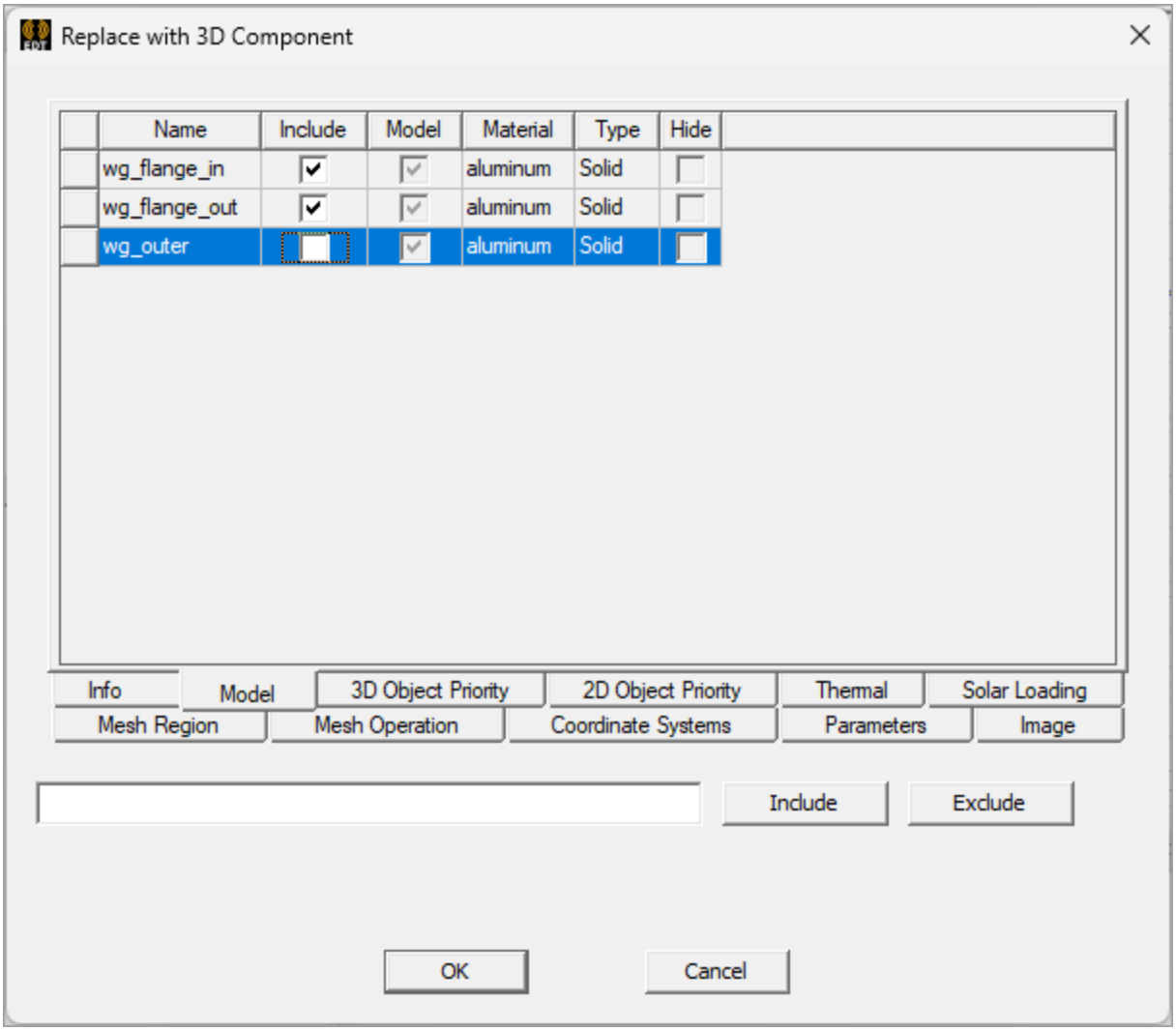
- Info tab
- Model tab
- 3D Object Priority tab
- 2D Object Priority tab
- Thermal tab
- Coordinate Systems tab
- Parameters tab
- Encryption tab

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 that you can then use the command buttons to Include or Exclude.



Object selection for **Include** 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

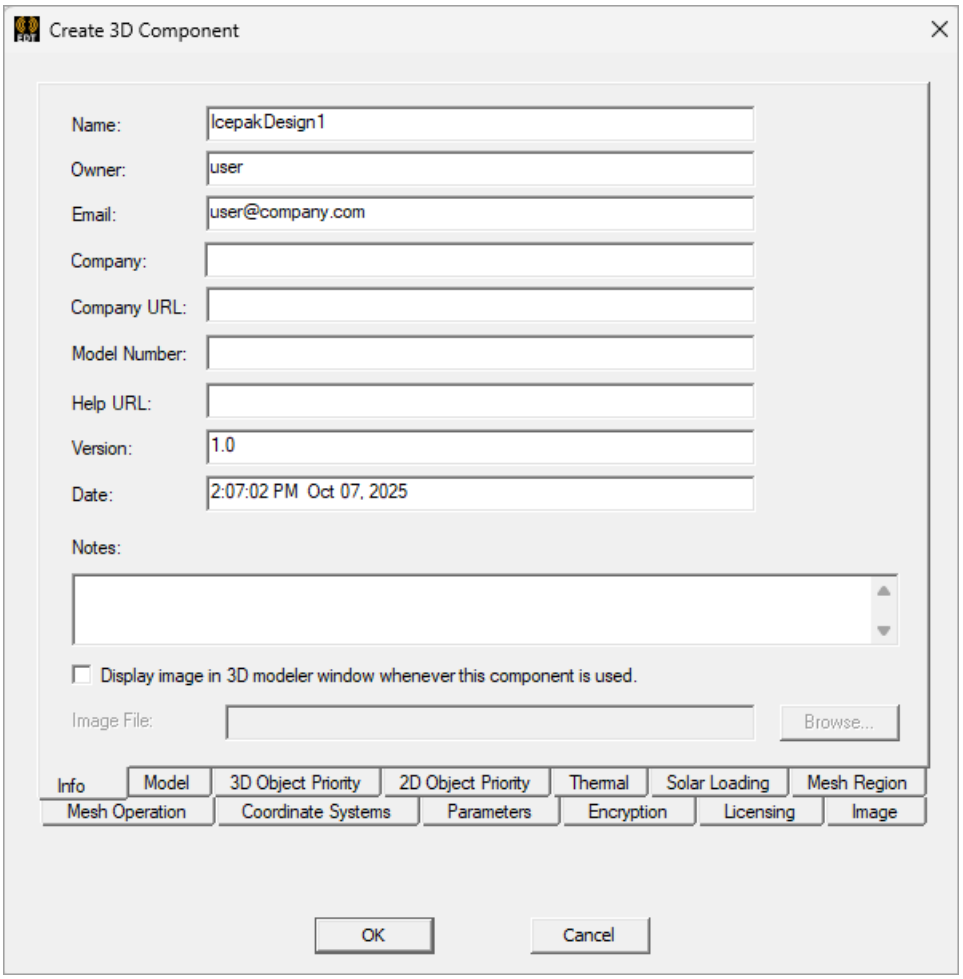
- The **Create 3D Component** dialog does not list design data without an assignment.
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.

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 Info Tab

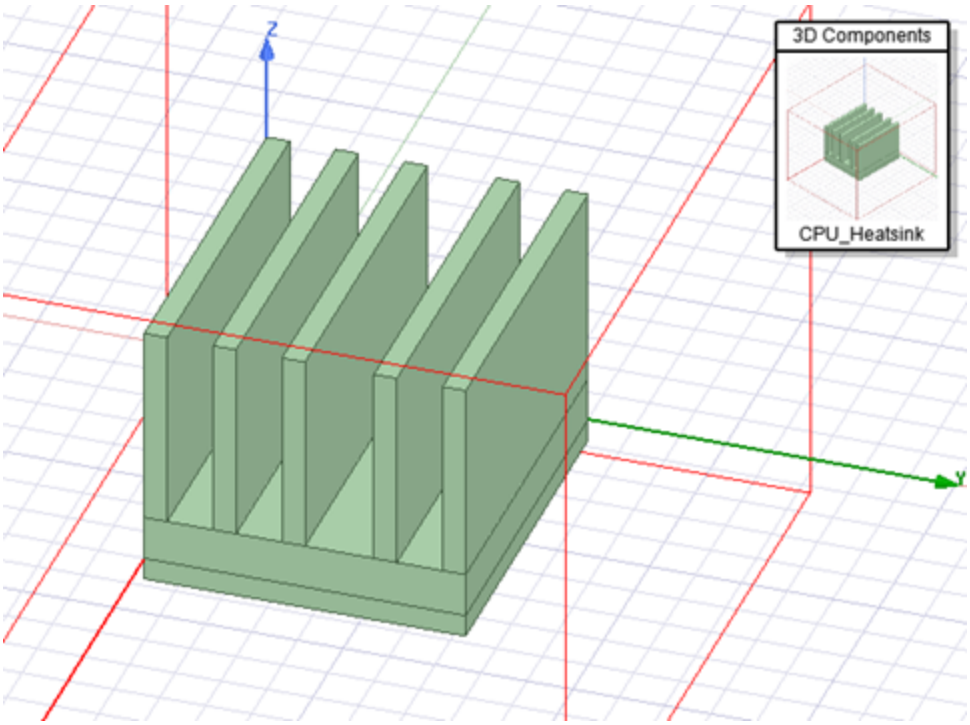
On the **Info** tab you can view and edit component information, as well as add notes. The *Name* in the first field is used when the component is inserted into a design. Other fields include *Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

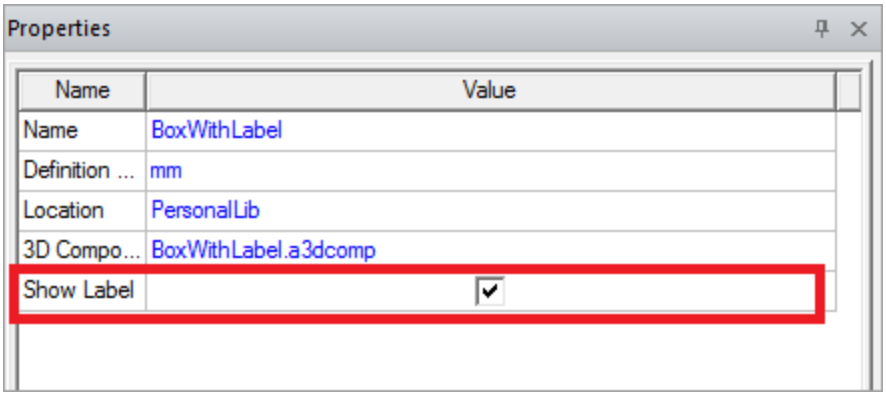


You can also included a .bmp format image file to display in the upper right area of the 3D modeler whenever the component is used. The image file typically contains a company logo. Check the **Display image** box to enable the **Image File** field, and click **Browse** to open a window to navigate your file system select the image.

Ansys Electromagnetics Suite 2026 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:



Create 3D Component: Model Tab

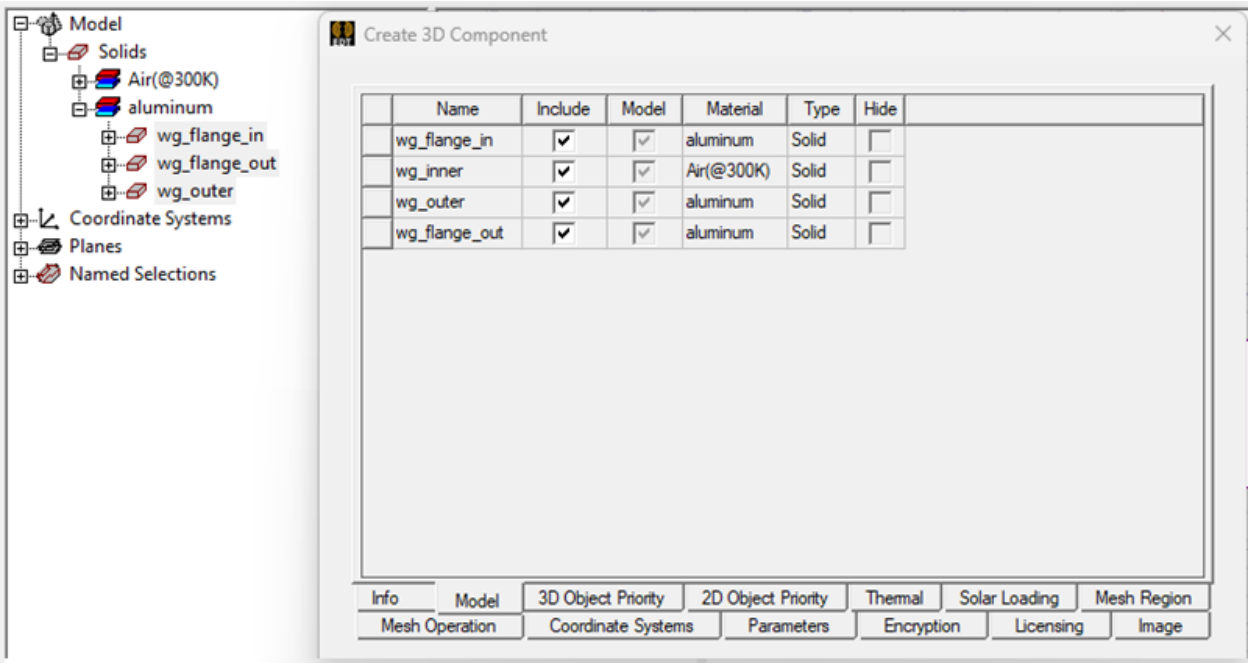
The **Model** tab lists all of objects selected before you invoked **Create 3D Component**. These correspond to the selected objects listed in the history tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

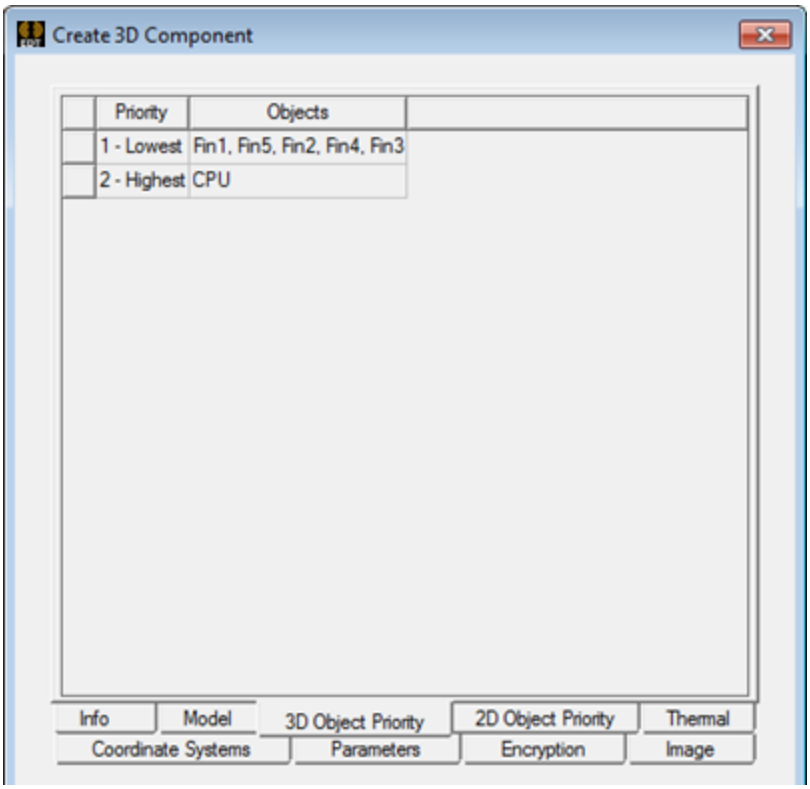
- The **Material** and **Type** columns are read only.
- For Encrypted components, you can also chose which objects to hide.
- The **Hide** column is only enabled if you have the **Hide Selected Contents** option selected on **Encryption** tab.



Create Component 3D Object Priority Tab

On the **3D Object Priority** tab you can view the priority level of each of the three-dimensional geometry selected to create the 3D component.

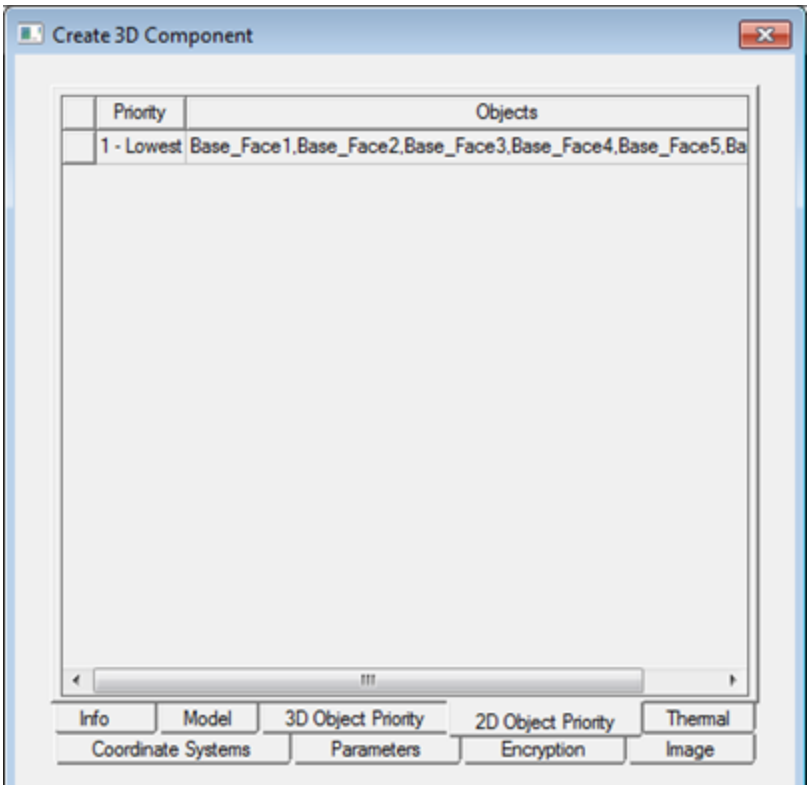
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create Component 2D Object Priority Tab

On the **2D Object Priority** tab you can view the priority level of each of the two-dimensional geometry selected to create the 3D component.

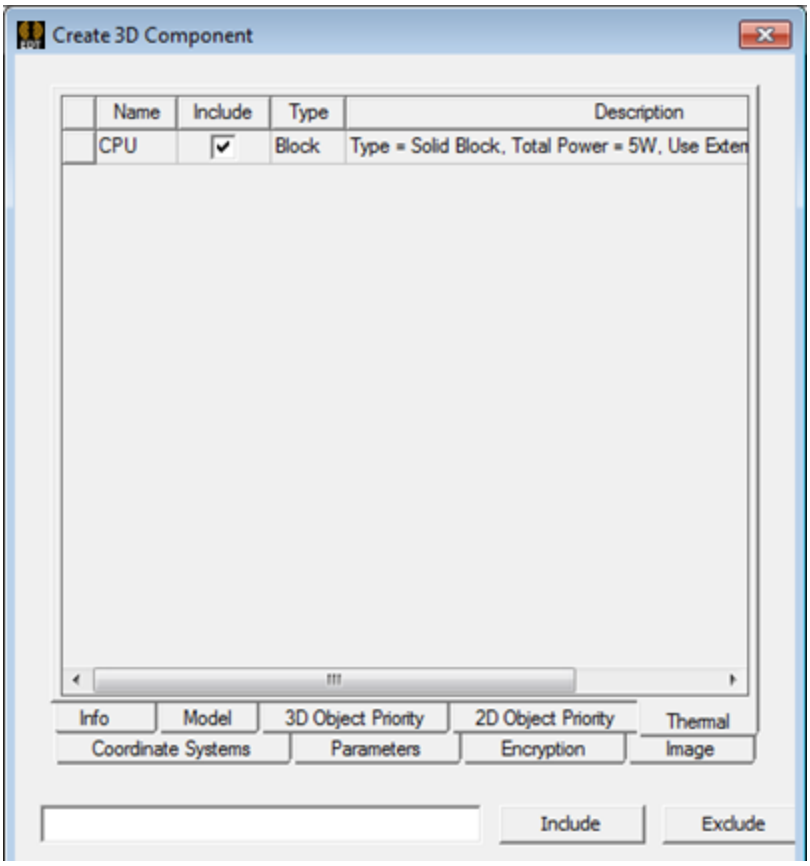
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create Component Thermal Tab

On the **Thermal** tab you can view a list of thermal boundary conditions assigned to the geometry selected to create the 3D component. Use the check box to Include or Exclude variables. You can also use the text field and **Include**/**Exclude** buttons.

Ansys Electromagnetics Suite 2026 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

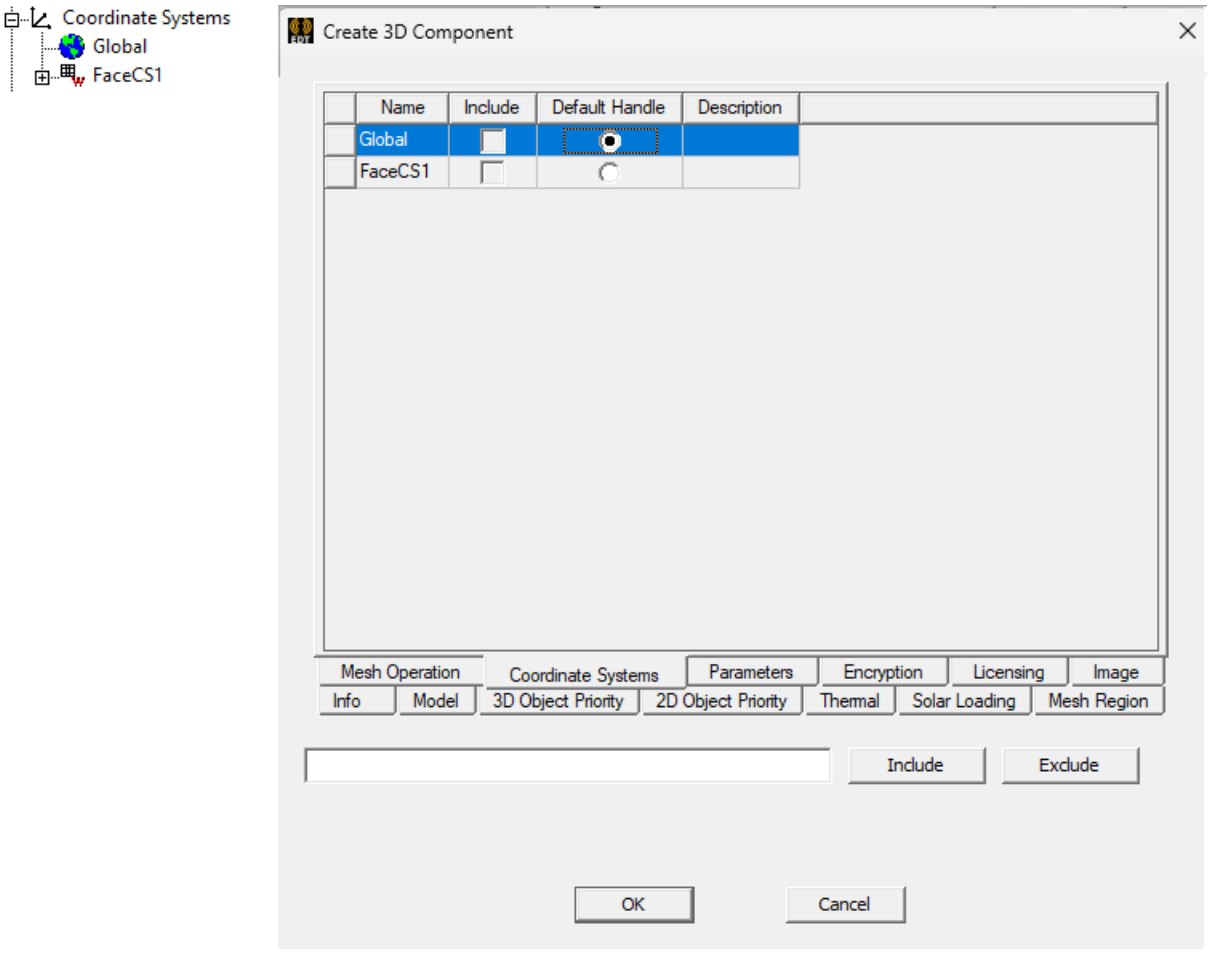
The list is populated with all the Coordinate Systems on selected objects. By default, only the Coordinate Systems used to define the object orientations are included. You can include Coordinate Systems created on specific model parts.

- 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, so you can change the inclusion state of multiple coordinate systems with similar names in a single operation.

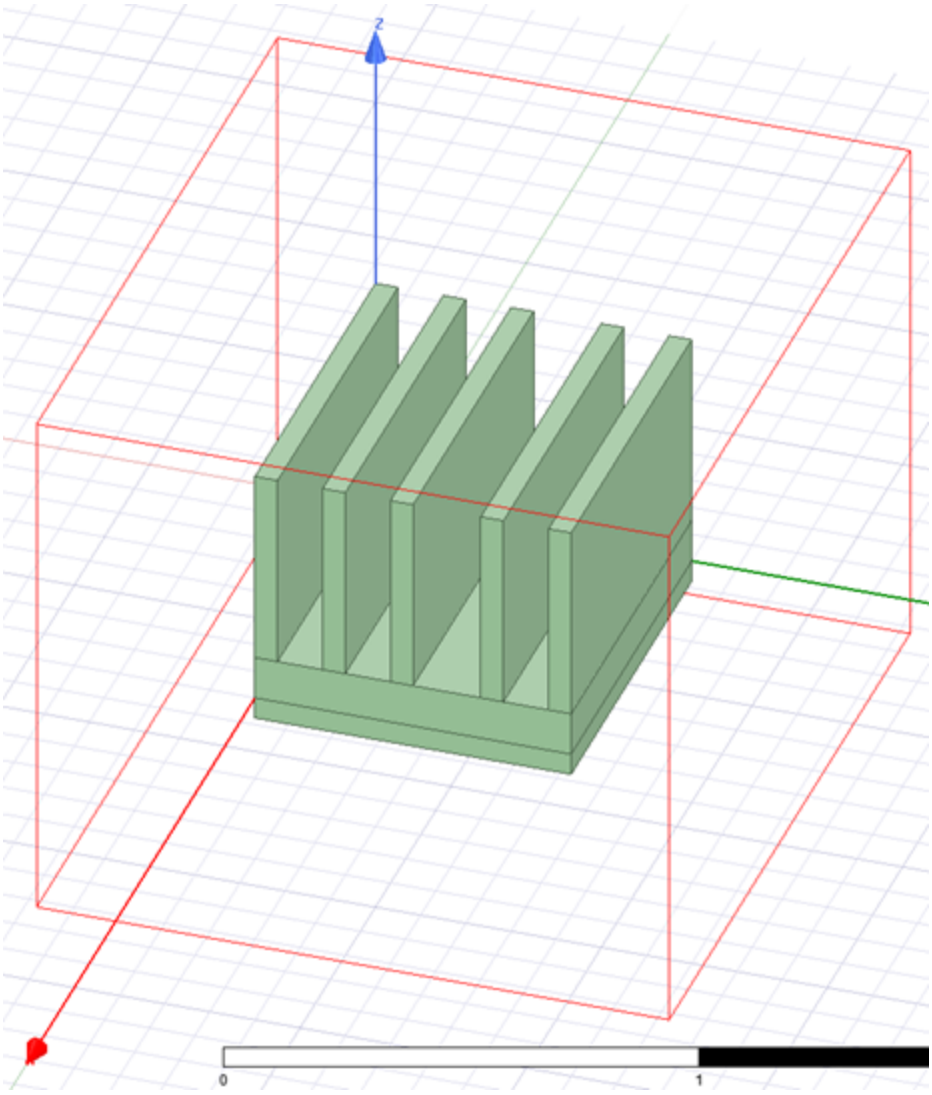
- You can select any of the CS as the default handle. When this component is placed in assembly design, the handle CS is aligned to target CS in assembly design. Note that the handle CS can be changed after import; see Editing 3D Component Properties for more details.
- By default, the current working CS is designated as the default handle CS.
- The default handle CS must be included in the model (except for global).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The specified default handle affects the orientation of the component upon insertion:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Parameters Tab

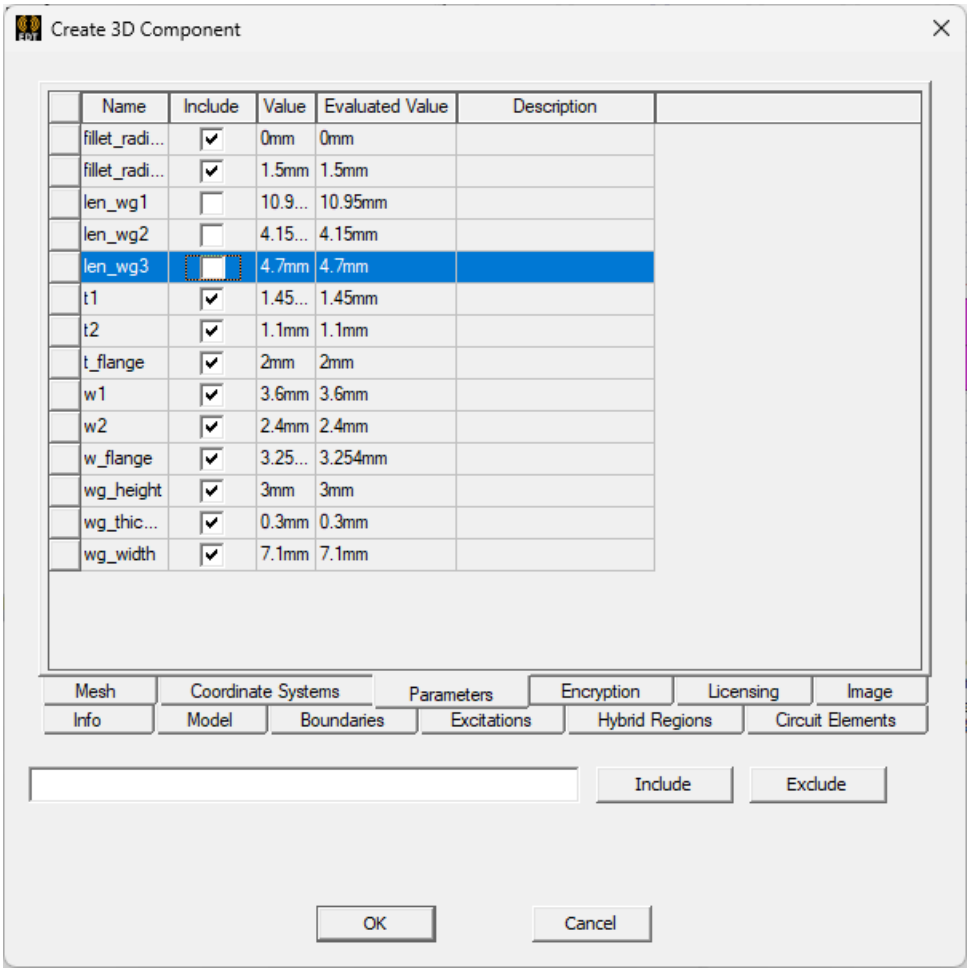
Component properties can be parameterized by assigning variables. The parameters list is populated with all the variables used by objects and coordinate systems, design data, and material properties included in the component.

- 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.

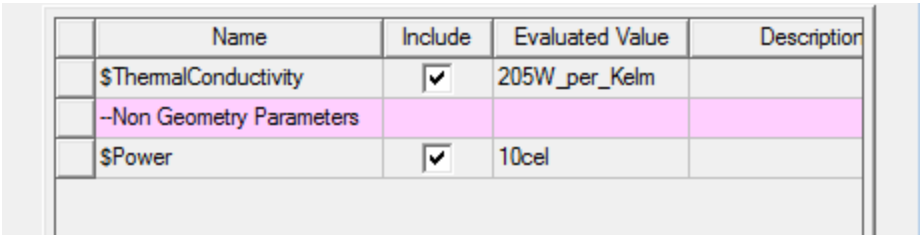
- You can fill in the **Description** field, if desired.

Ansys Electromagnetics Suite 2026 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 (for materials or boundaries), these are listed separately:

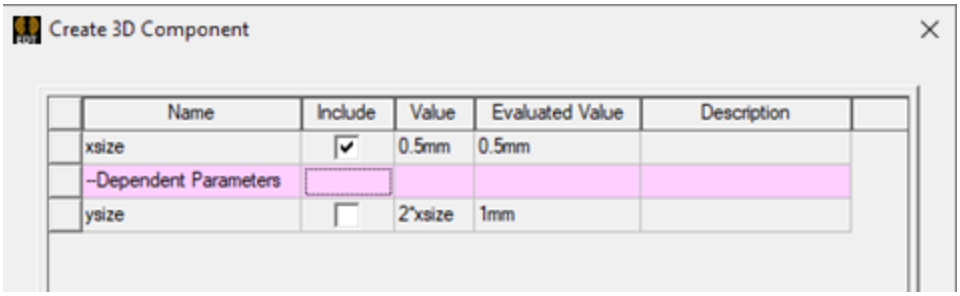


If you exclude any geometry, boundary, or material associated with a variable, the associated variable is automatically excluded on the **Parameters** tab.

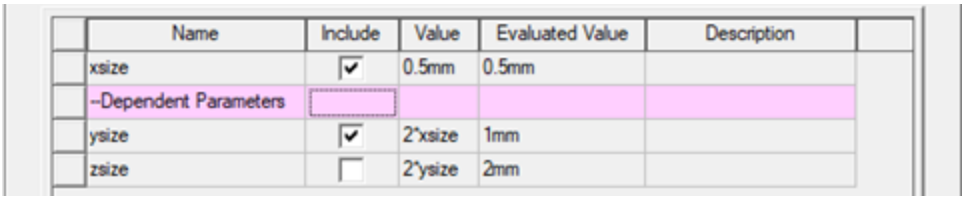
Dependent Variables in 3D Components

If you create dependent variables, you can choose to display the values.

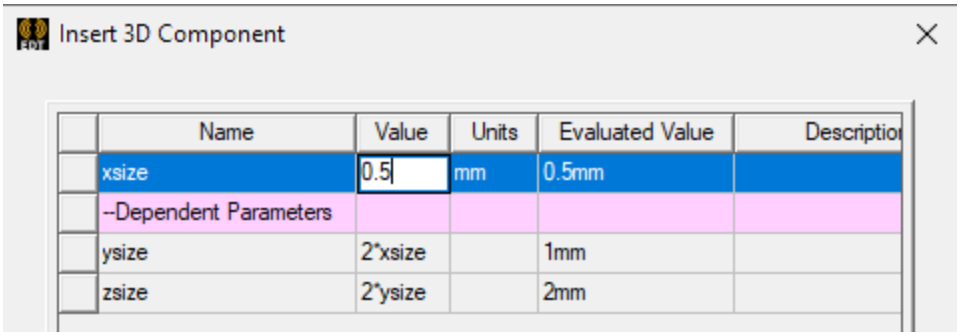
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If more variable dependency occurs for an Included variable, such variables are also displayed with the option to include.



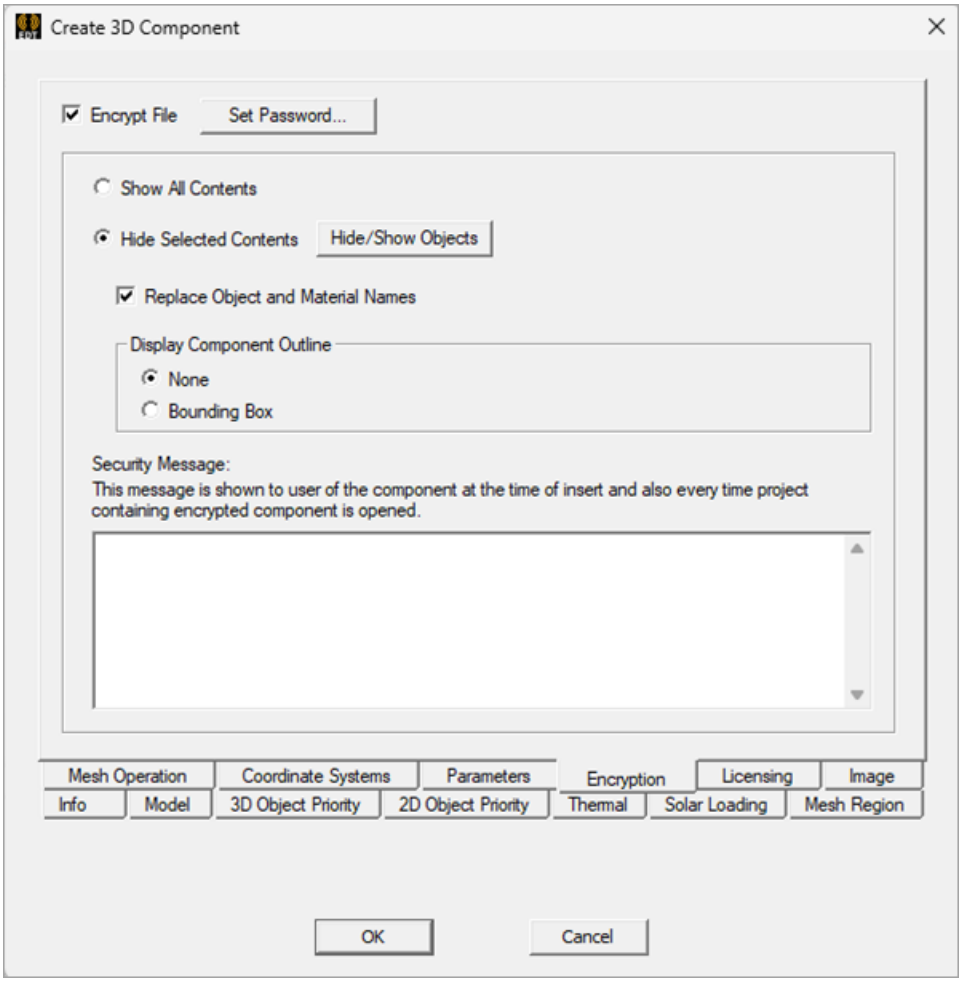
When you Insert a 3D Component with dependent Parameters included, you can edit the Independent variables, but can only view the evaluated values of the Dependent Parameters.



Create 3D Component Encryption Tab

The **Encryption** tab contains options for 3D component encryption.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** Icepak native components (Fan, Heatsink, PCB), EM loss coupling, IcepakFEA-Icepak coupling, and **Start/Continue from a previously solved setup** (transient designs) are not supported for encrypted 3D component.

Selecting the **Encrypt File** check box enables all options and automatically launches the **Set Passwords** window.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

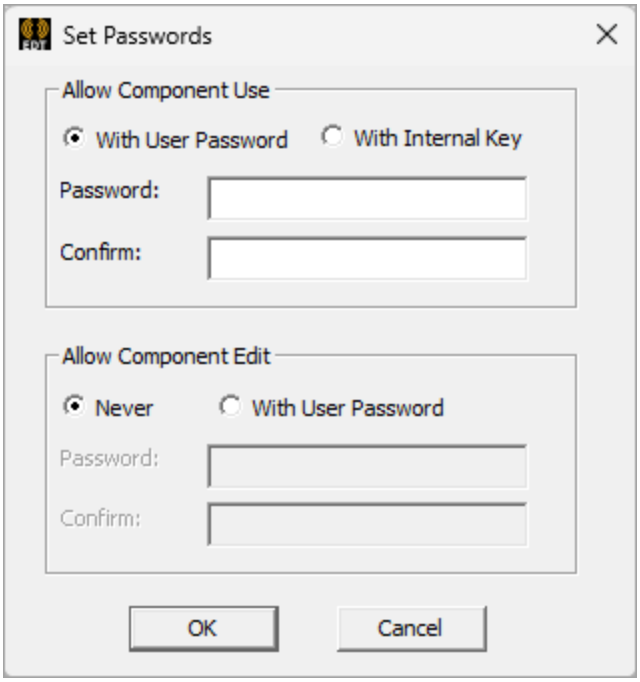
Important:

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 or Maxwell 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 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**.

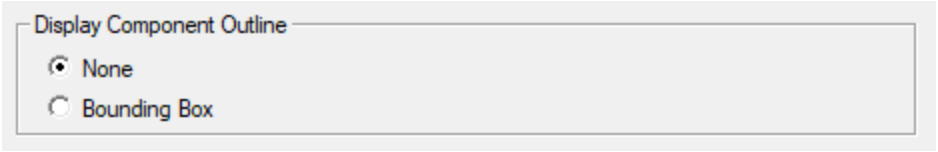


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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

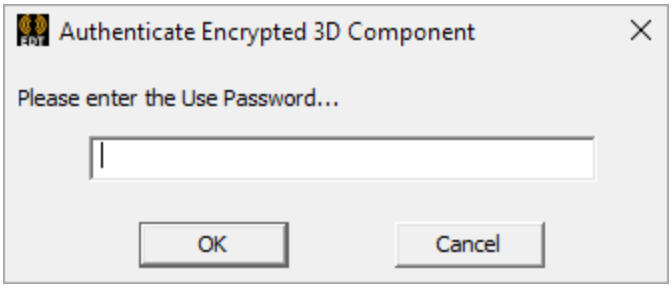
The **Hide Selected Contents** option allows selected parts of the component to be displayed either as a **Bounding Box** or without an outline (**None**).



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.

Inserting Encrypted Components

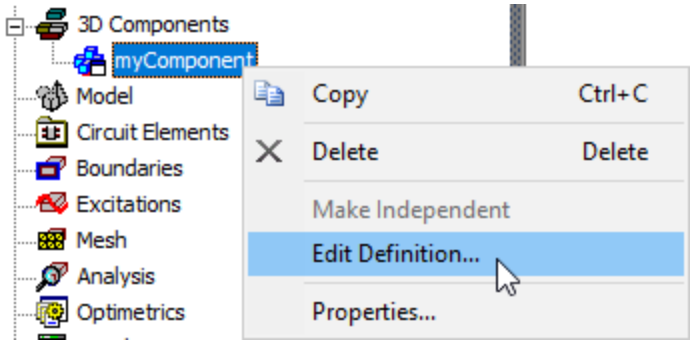
When an encrypted 3D component is added to a design, a prompt will appear asking for the password.



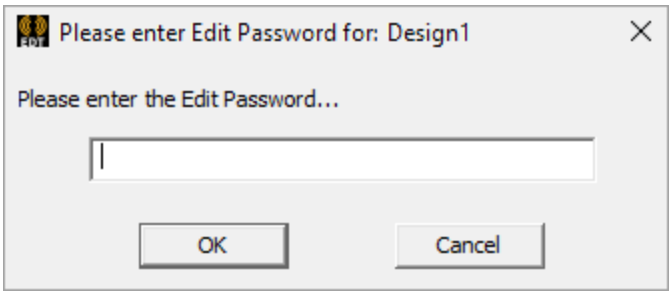
Editing Encrypted Components

3D component definitions can be edited from the **Project Manager**:

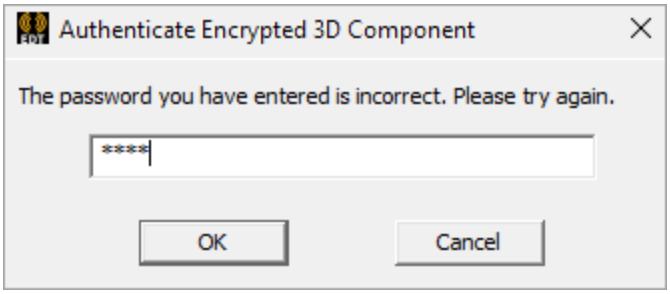
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A prompt appears asking for the password:

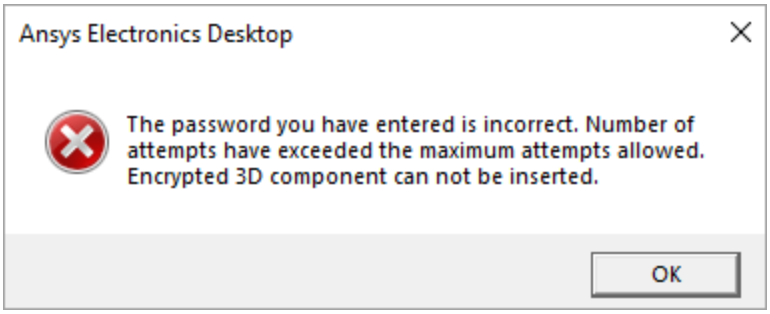


If the password is incorrect, a warning appears:



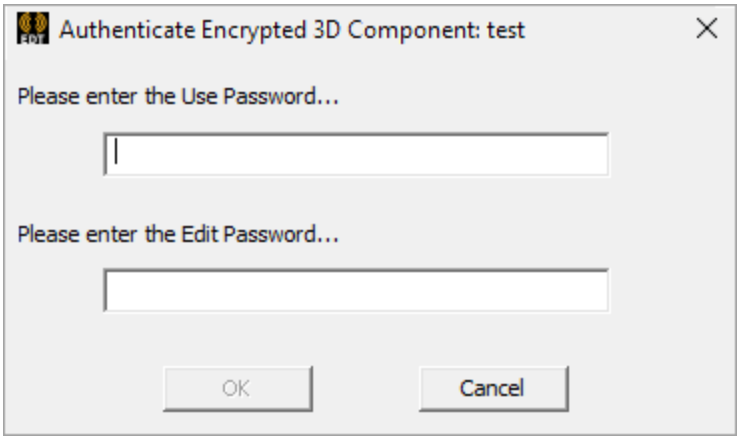
There is a three attempt maximum, after which a warning will appear and no further attempts may be made:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:



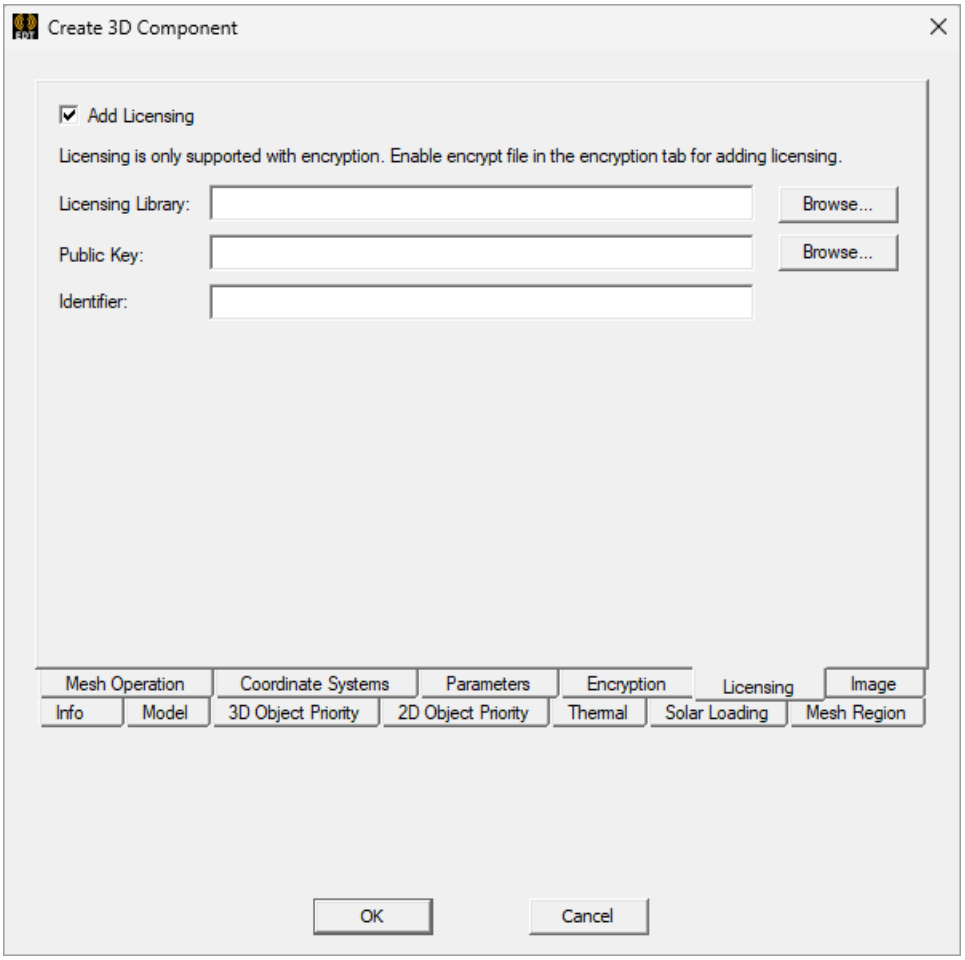
Copying and Pasting Encrypted Components

Encrypted components may be Copied and Pasted. They will be remain encrypted.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

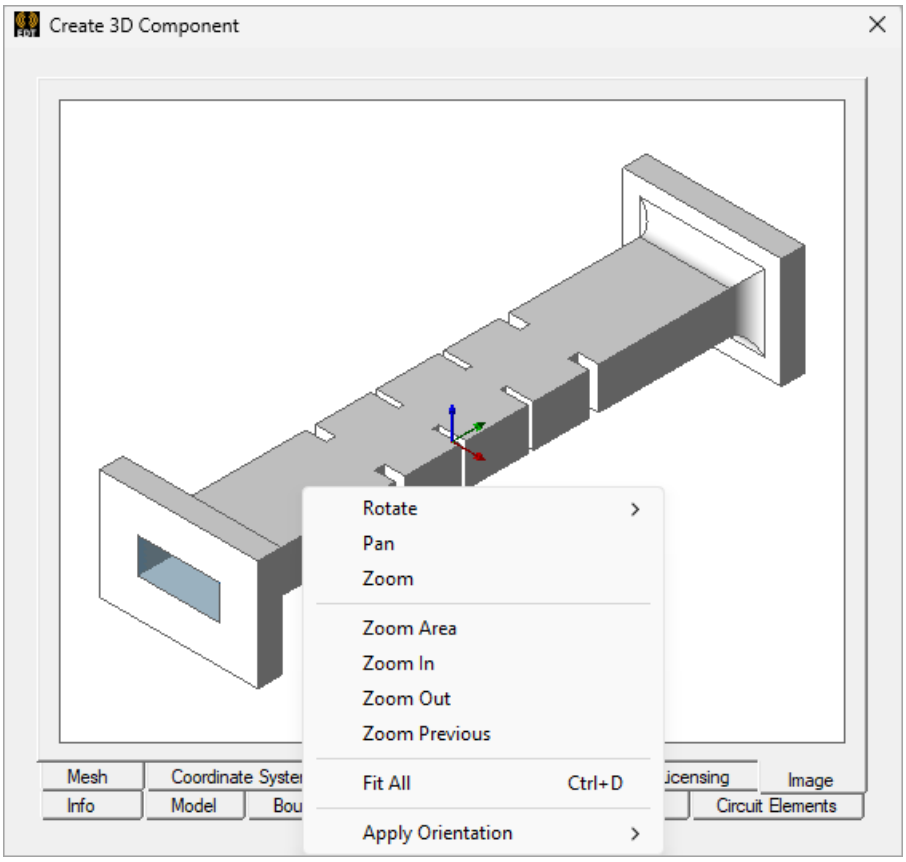


Create 3D Component Image Tab

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.

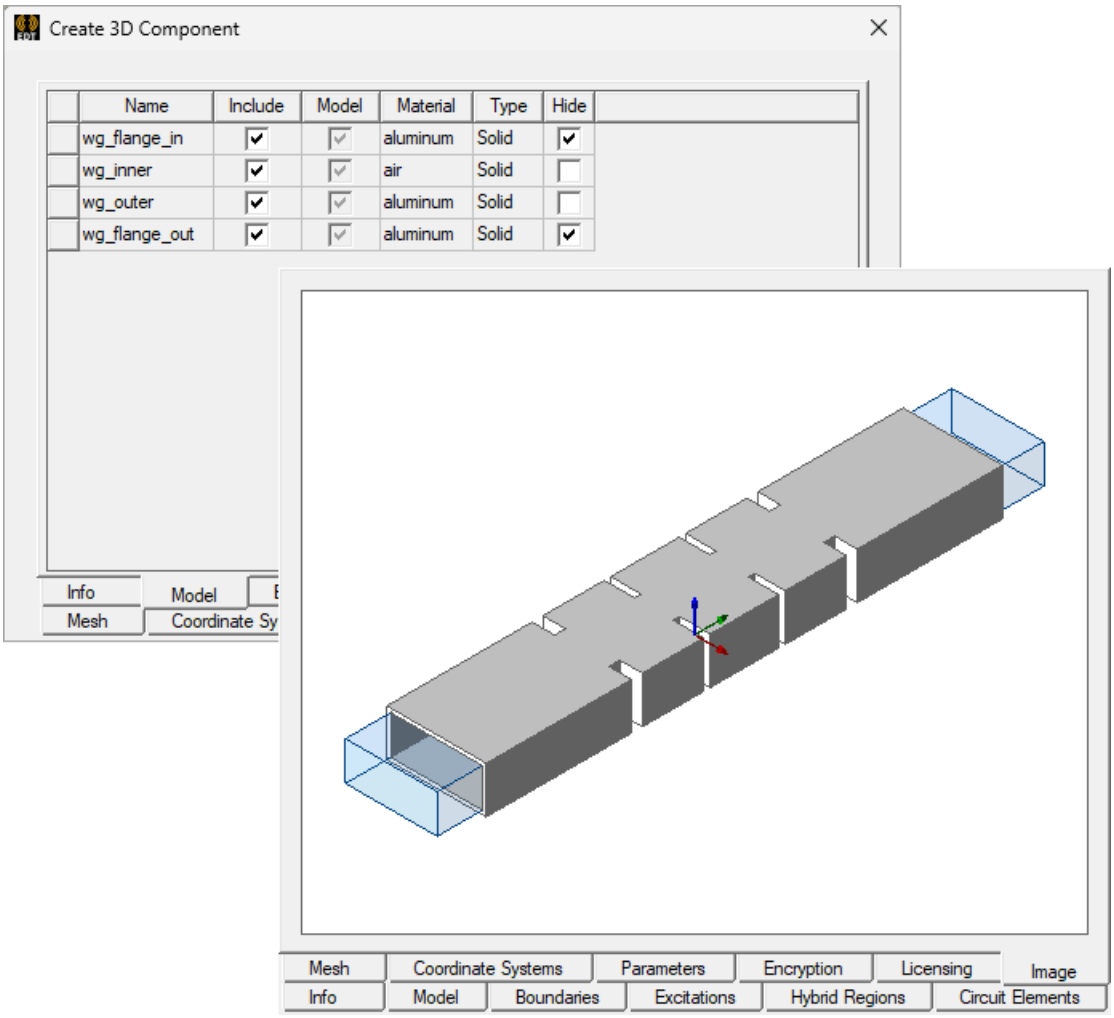
- You can right-click for a shortcut menu to change the view of the preview image.
- Changing the preview orientation does not affect the modeler window view.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



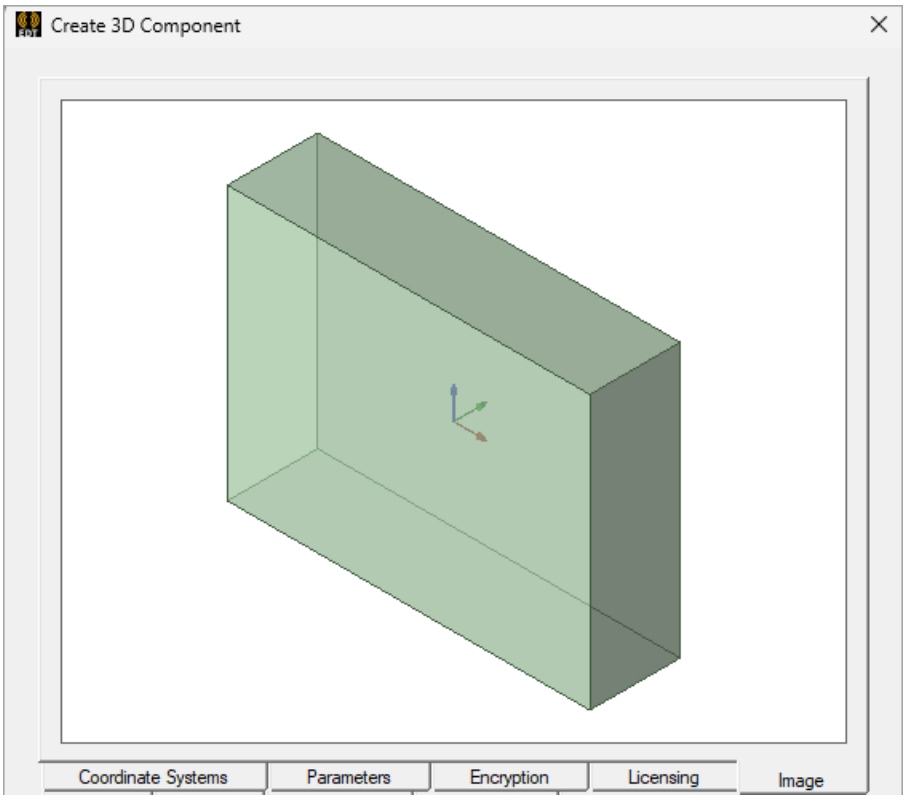
If you use the **Encryption** tab and **Model** tab settings to hide/show the model or specified model objects, the **Image** tab responds. You must use encryption to enable the Hide check boxes on the **Model** tab.

Ansys Electromagnetics Suite 2026 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.

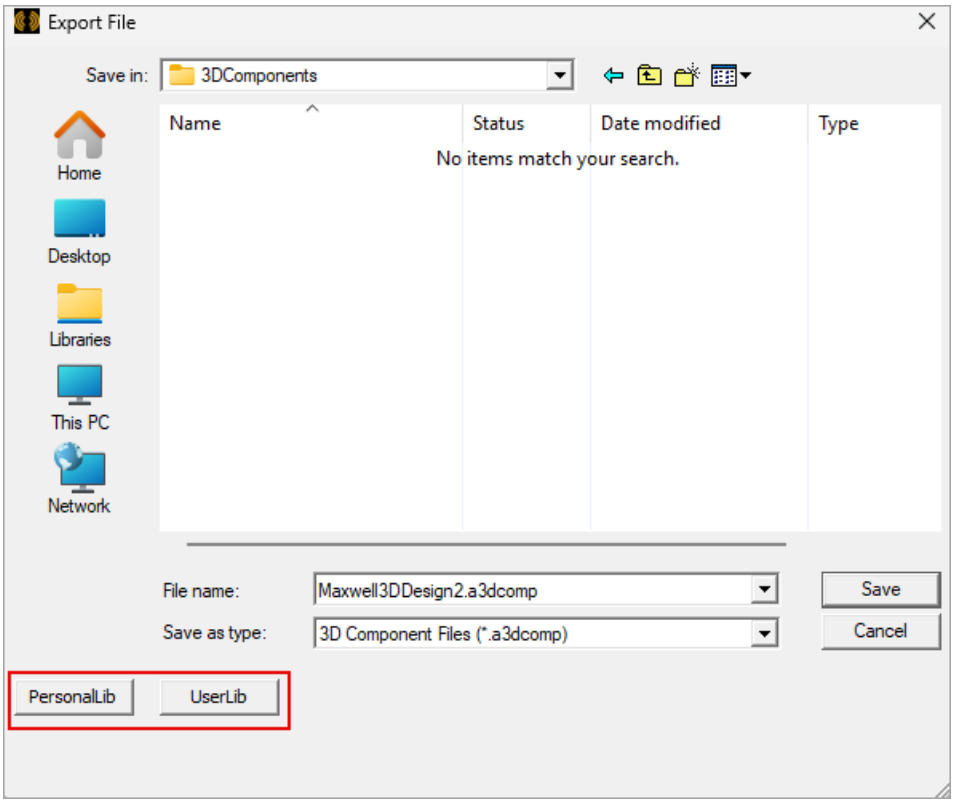
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



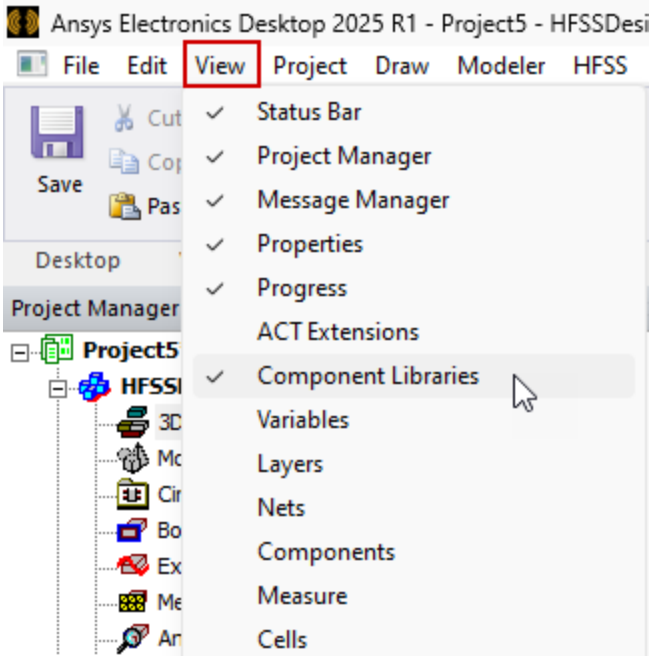
The default **File name** is the component *Name* specified in the *Info* tab.

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 Components** folder in the Project Manager tree or, using the menu bar, click one of the **Draw** > **3D Component Library** commands to browse your folders or libraries.

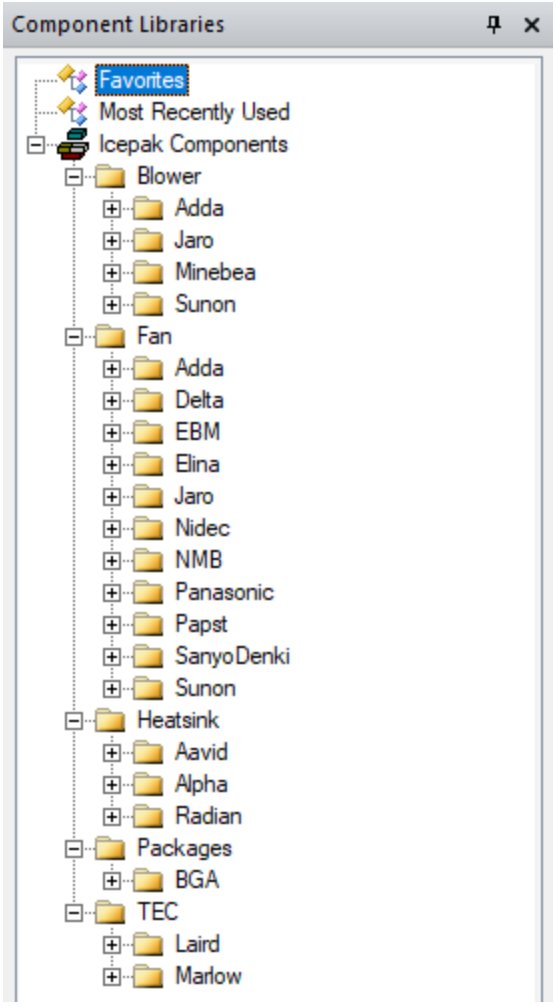
Toggle the visibility of the docked **Component Libraries** window from the menu bar by clicking **View** > **Component Libraries** and navigate the installed libraries from there. Visibility of the **Component Libraries** window is remembered for each design type so that, if context switches between different design types, the window is shown or hidden based on setting for new design type.

Ansys Electromagnetics Suite 2026 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.

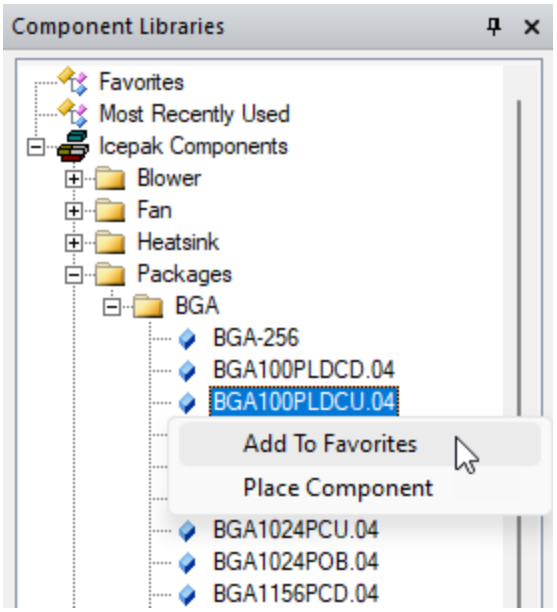
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

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 Icepak should be organized under 3DComponents\Icepak, 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, Icepak will ignore component files placed directly under the HFSS subfolder.

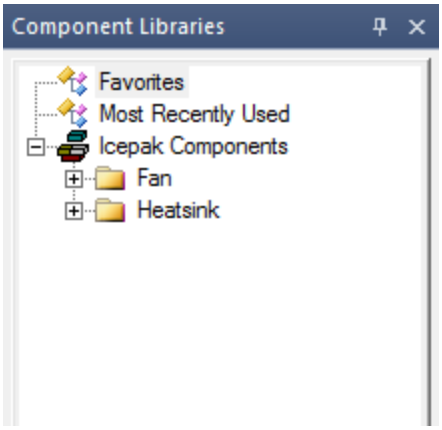
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 do any of the following:

- Right-click the **3D Component** icon in the Project Manager or use the **Draw** > **3D Component Library** commands to browse your folders or libraries
- Use the **View** > **Component Libraries** to display a Component Libraries window to navigate installed libraries.
- 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 to the directory of installed components, as well as any in your Personal Library. The hierarchical tree display includes **Favorites** and **Most Recently Used** branches.



You can select any component from the library, and drag and drop to insert it in the design.

When you drag the component into the Modeler window or use the Browse function to select and place a 3D component, it appears as an outline to make it easier to place. The diamond-shaped marker designates the placement location.

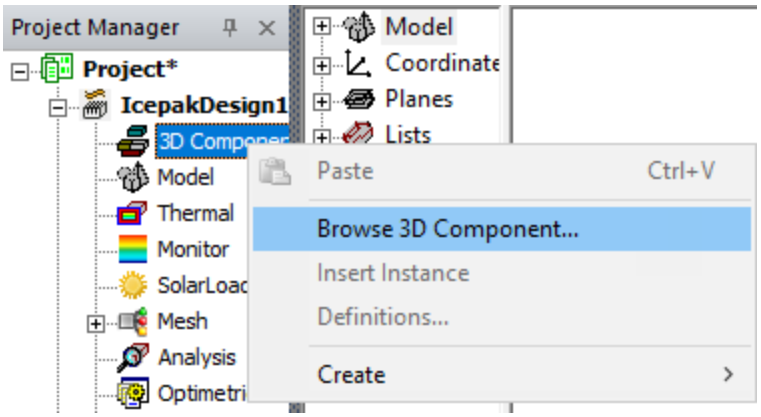
Move the cursor in 3D modeler window to snap to a point where you want to insert the component and then click to place. While the component is being moved, the Measured Data window appears and reports the position in the Reference and Current coordinate systems.

The position of the 3D component will be reported in the Properties window. It can also be edited from this window.

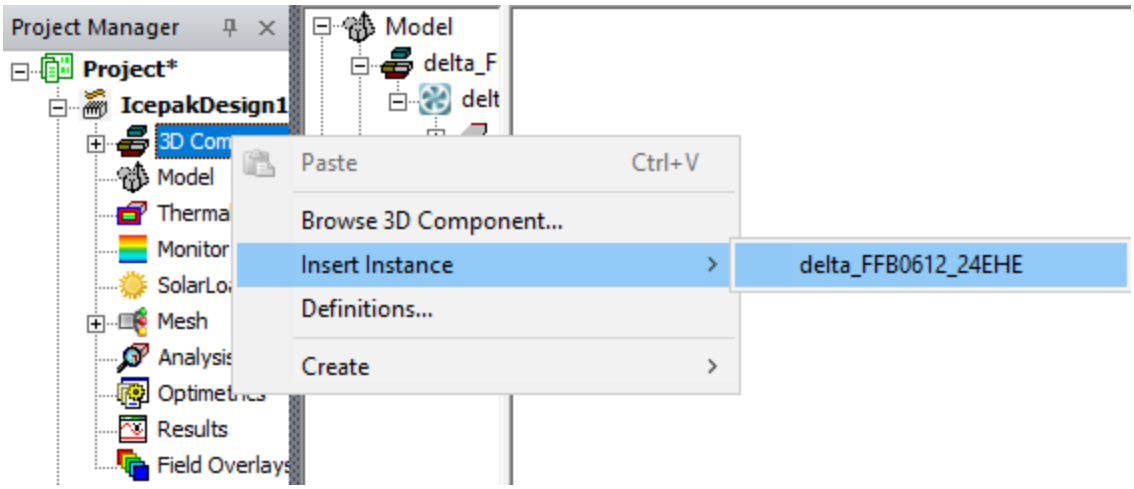
Double-click any model in the component libraries to . You can also view the **Info** tab, and select the target coordinate system.

Selecting **Browse 3D Component** lets you navigate directories via a browser window.

Ansys Electromagnetics Suite 2026 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 easily insert another instance of a 3D Model already in the design.



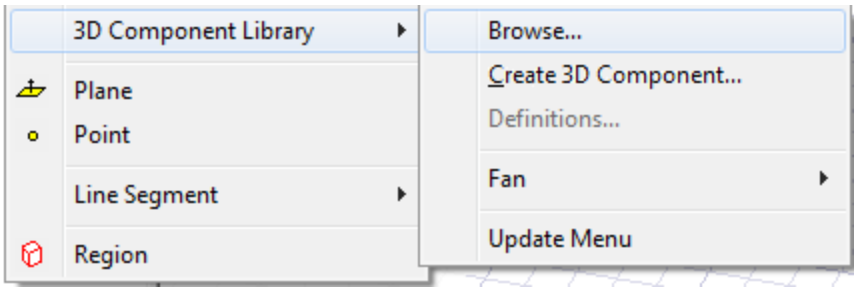
Another option to select a 3D component is to click **Draw** > **3D Component Library** to access the **Browse** and **Insert Instance** menus.

Note:

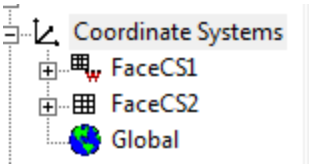
The following vendor components are not supported:

- CAD data-based heatsink components
- Heatsink components created as an assembly

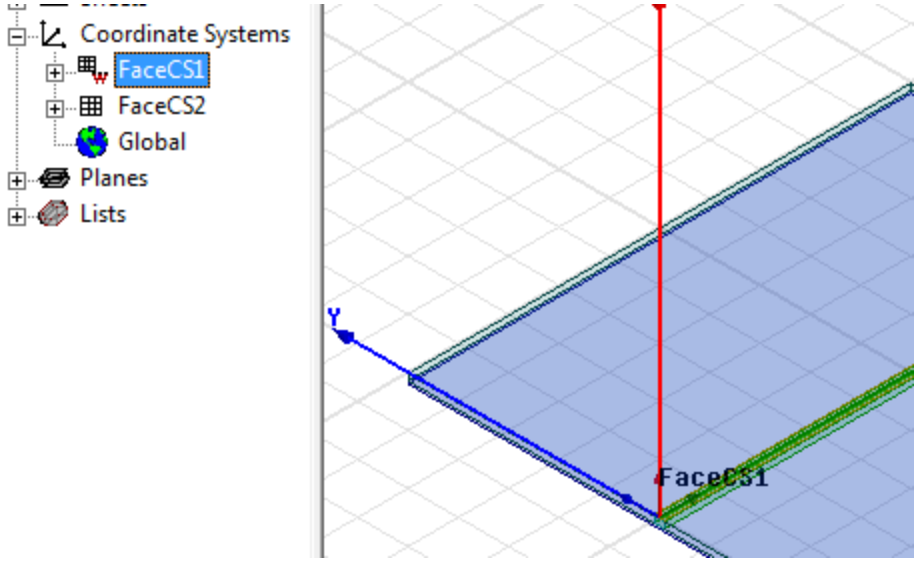
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A target coordinate system provides a location for a component. For example, a design includes two additional coordinate systems:



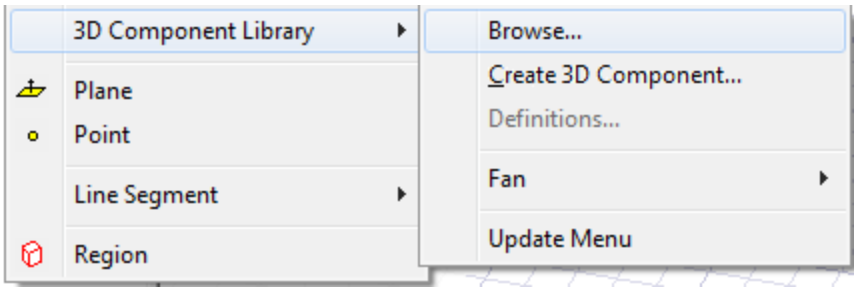
Selection of a coordinate system in the History tree displays a potential location for inserting a component.



To Insert a 3D Component from the Menus

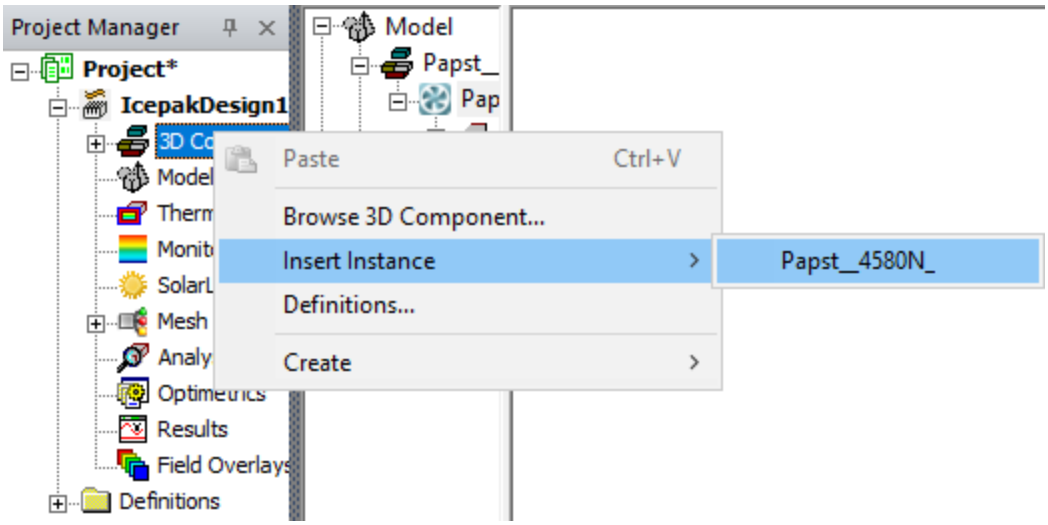
1. Right-click the **3D Components** icon in the **Project tree** to open shortcut menu. You can select **Browse 3D Component** to use a browser window to navigate the file system to the component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



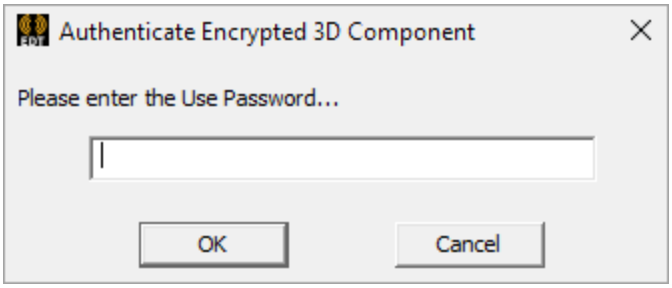
You can edit parameter values, and assign variables or expressions for parameters.

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.



2. 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.



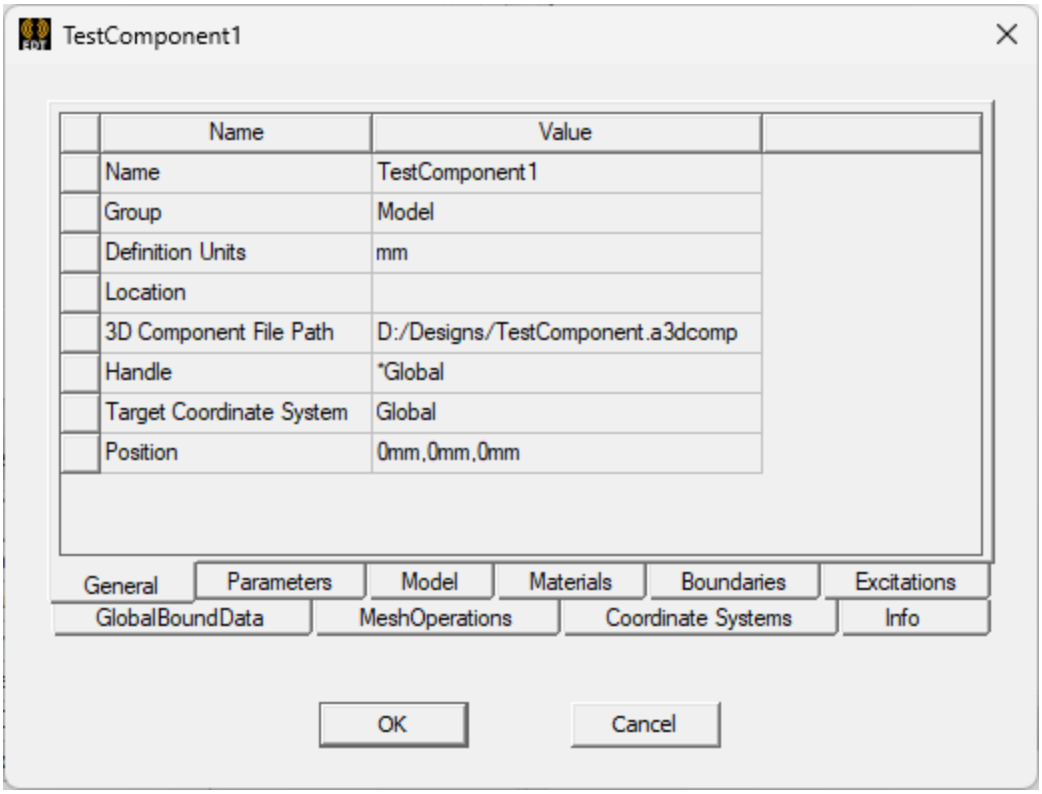
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have three tries to enter the correct password. Upon successfully entering the password, the Insert 3D Component window 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.

Viewing Properties of 3D Components

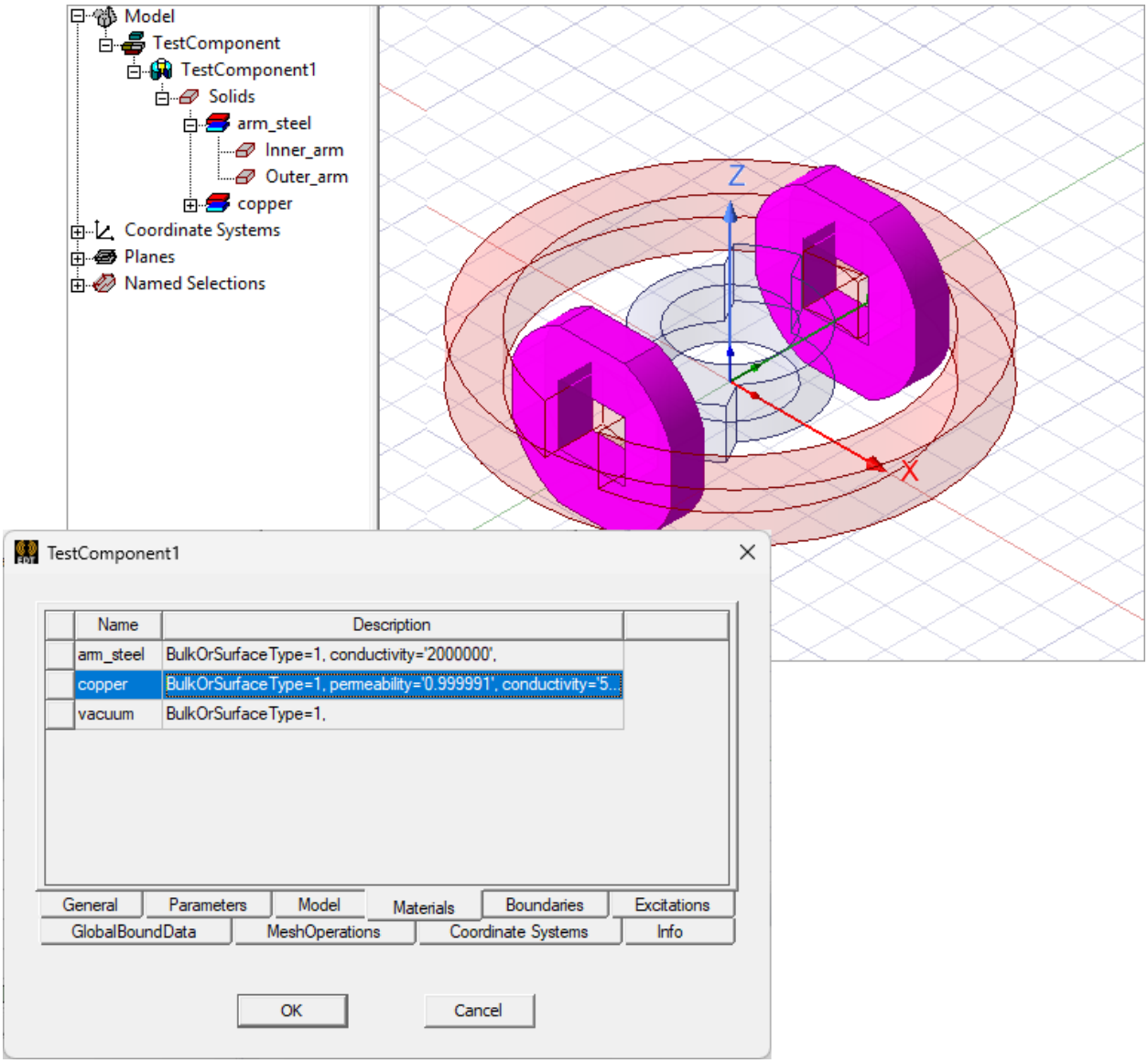
The Properties displayed for a selected 3D component depend on how you select that component. For a complete display of Properties, right-click the selected component (Project Manager, History Tree, or Modeler window) and click **Properties** on the shortcut menu. Alternatively, click **Edit** > **Properties** using the menu bar. This action displays a window with tabs for all component properties.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

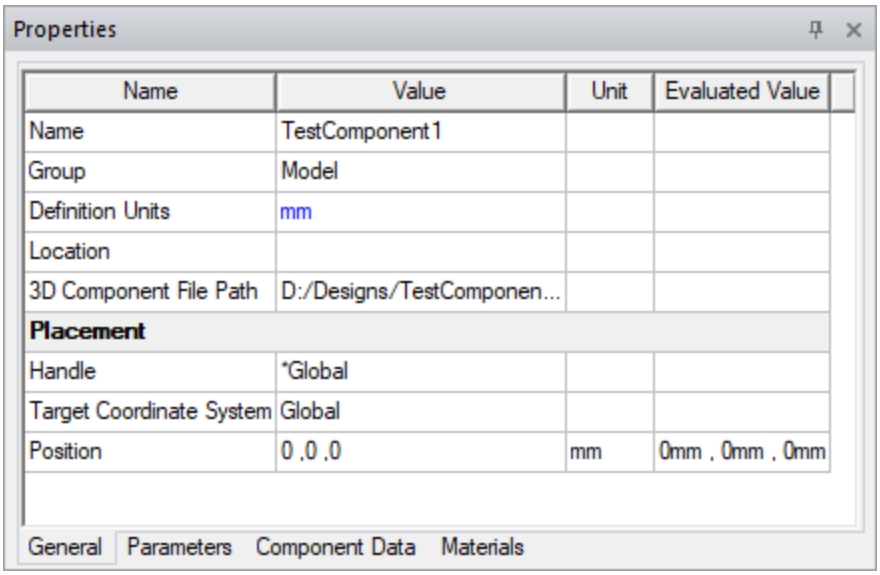
> **Note:** The **3D Component File Path** shown in the Properties window is the original path from where the component file was inserted from; it does not reflect the location of the definition used in the current design, which is saved with the active project.

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.



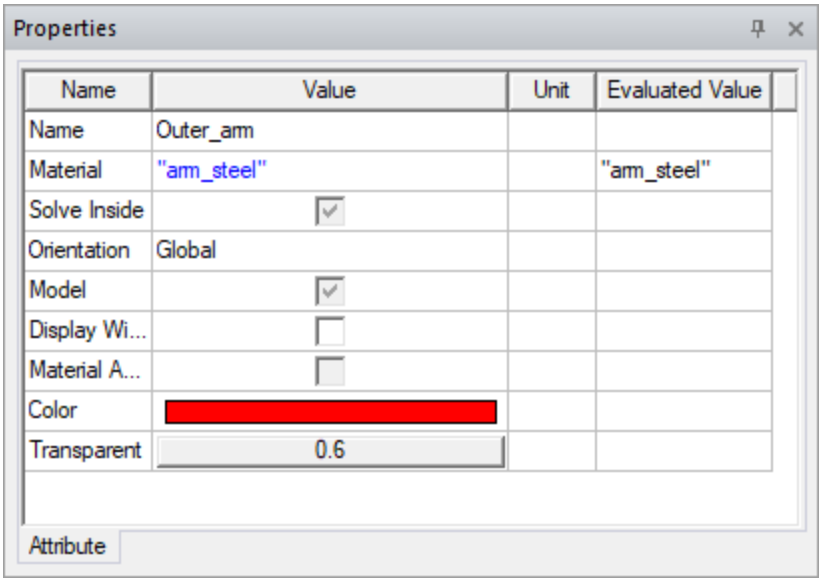
Selecting a 3D component in the Project Manager or the history tree displays tabs for *General, Materials,* and *Component Data* in the docked *Properties* window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



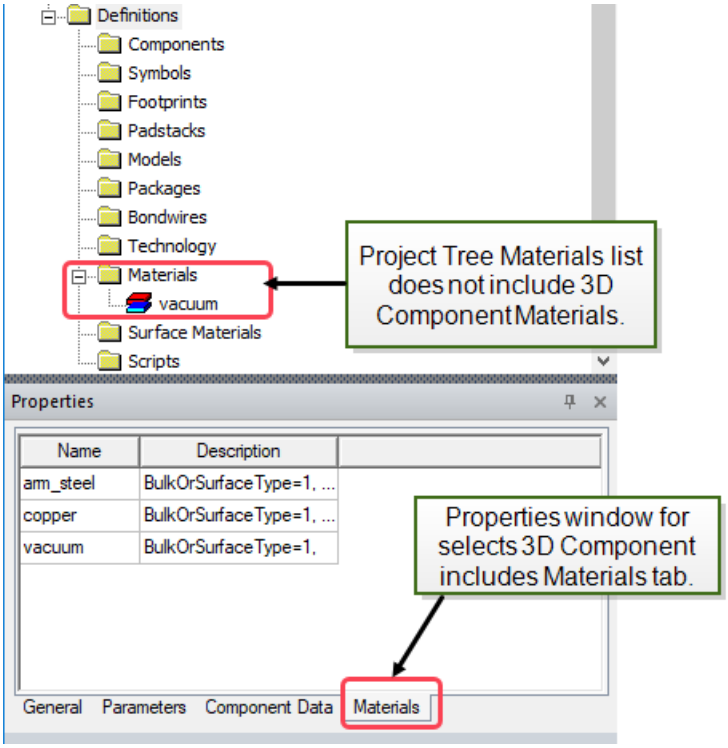
Note that the Handle property lists the handle coordinate system that was selected as reference when the 3D component was created; from this window, it can be changed to any coordinate system in the 3D component.

Selecting a 3D component object in the Modeler window displays the Attributes tab in the docked Properties window:



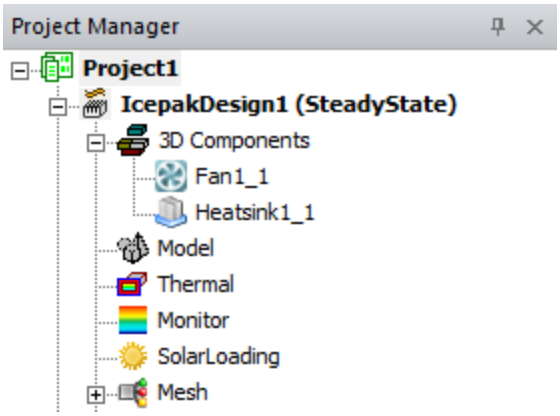
Component materials do not appear in the Project Manager's Definitions list. If you select the icon for an inserted component, the docked *Properties* window contains a *Materials* tab that shows the materials defined for the selected component.

Ansys Electromagnetics Suite 2026 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.

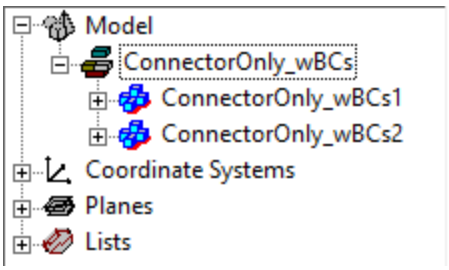


Selecting these inserted components populates the docked **Properties** window, which has tabs that display the instance attributes.

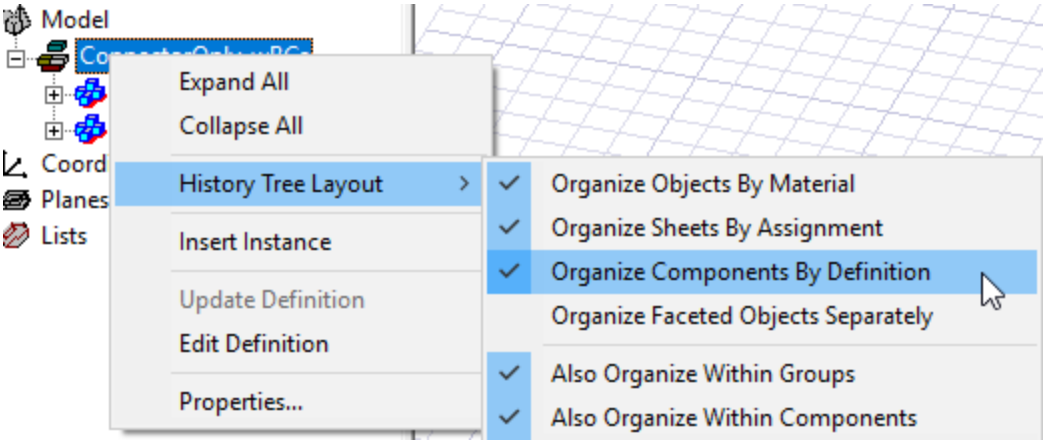
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Components in the History Tree

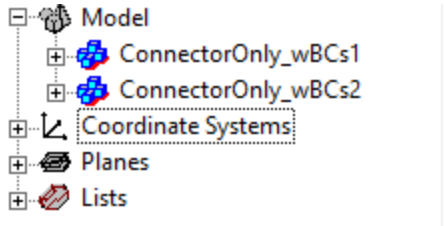
Although Group Commands do not work with 3D components, you can group all instances of a components by the component definition in the History tree.



To disable this grouping by instance, right-click an instance, and unselect **History Tree Layout** > **Organize Components by Definition**.



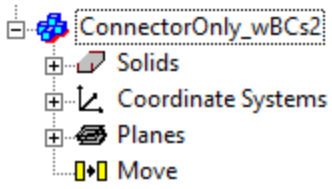
This causes the display to show numbered instances listed without the icon for the primary component.



Component Operations in the History Tree

Ansys Electromagnetics Suite 2026 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.

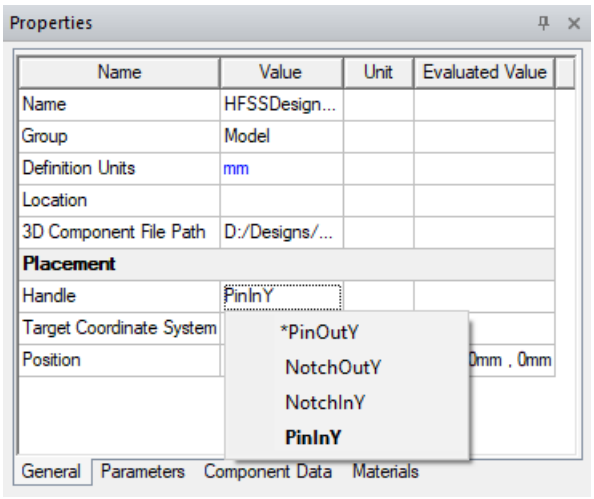


- Component operations acts on parts as well as CS.
- Boundaries, mesh operations are duplicated with duplicate and copy/paste operation.

Editing 3D Component Properties

You can change and manipulate 3D components in several ways:

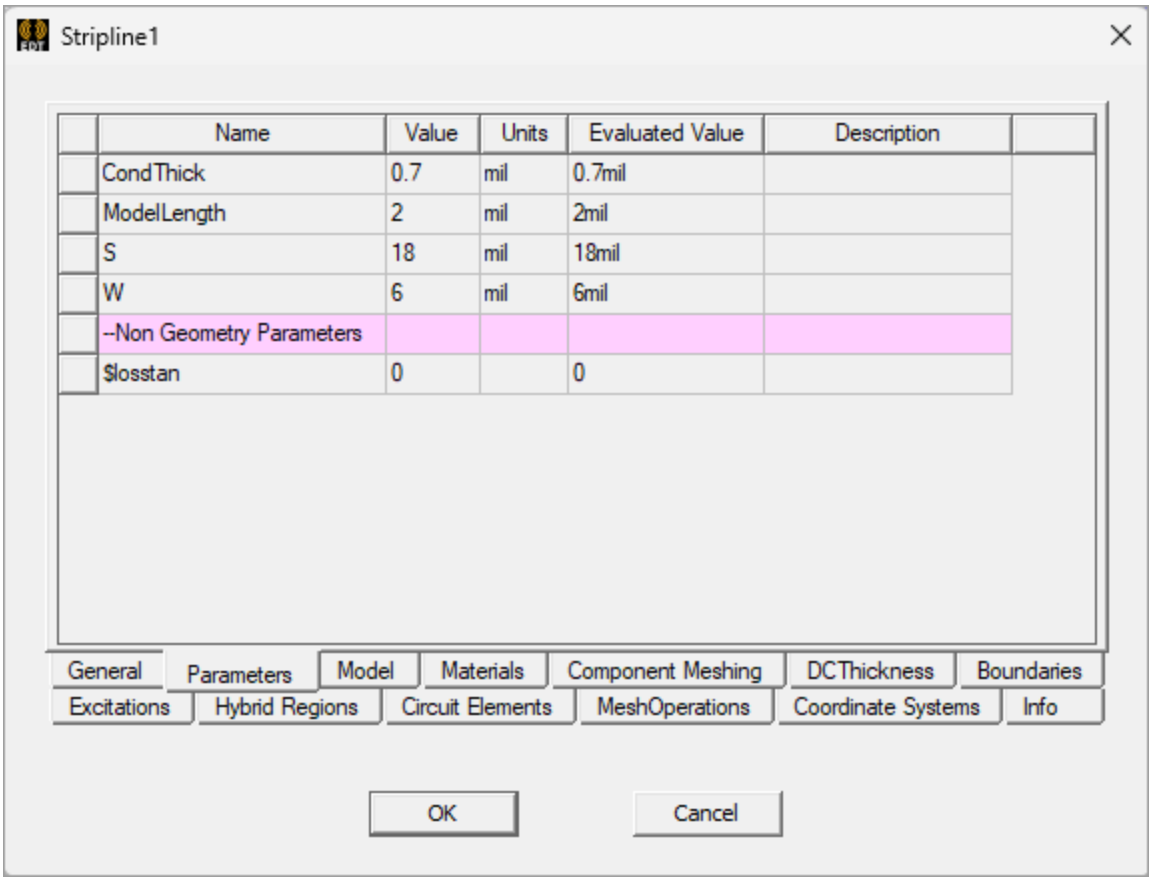
- Edit component handle and target CS in the General tab of component instance properties. All CS that are included in the component are available to be used as the handle. All coordinate systems in the assembly design that are eligible for component placement are available as target CS. Note that the default handle is shown with an asterisk.



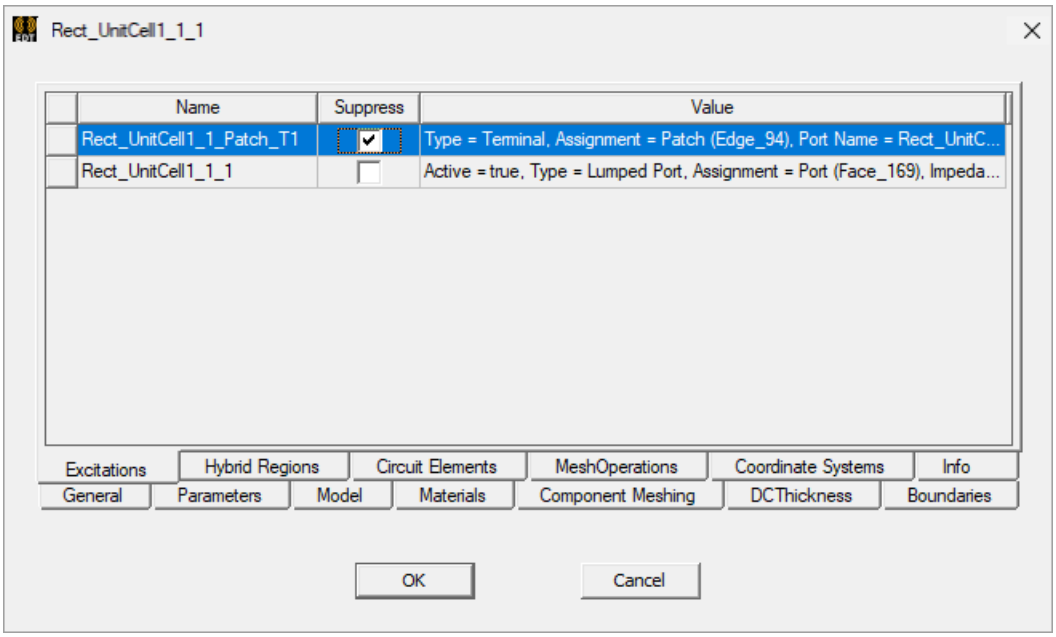
> **Note:** The **3D Component File Path** shown in the Properties window is the original path from where the component file was inserted from; it does not reflect the location of the definition used in the current design, which is saved with the active project.

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



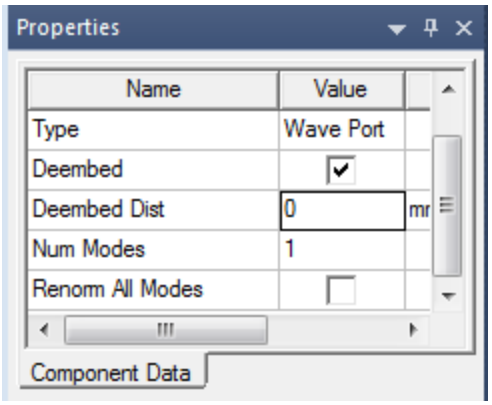
- Edit excitations by suppressing their use by means of the **Excitations** tab in the **Properties** window.



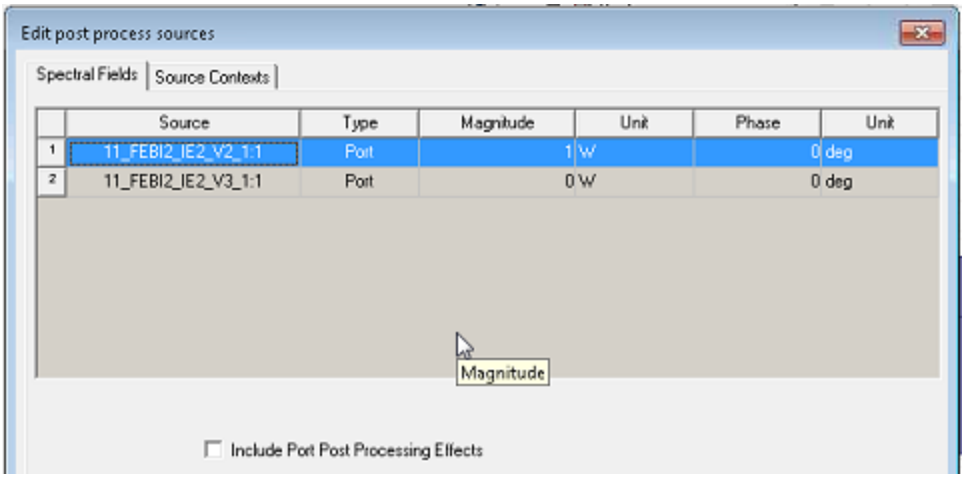
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

- Edit excitation Post Processing such as deembed distance, renormalization.



- Edit Sources for Post Processing



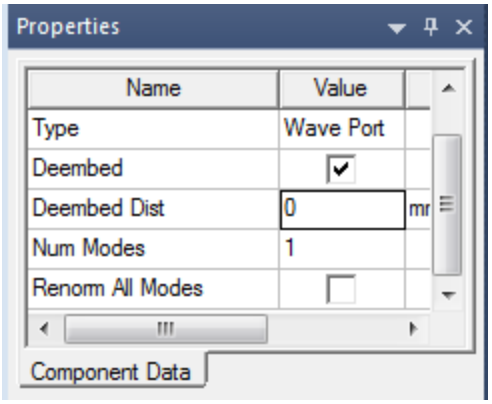
- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of a component are not allowed.

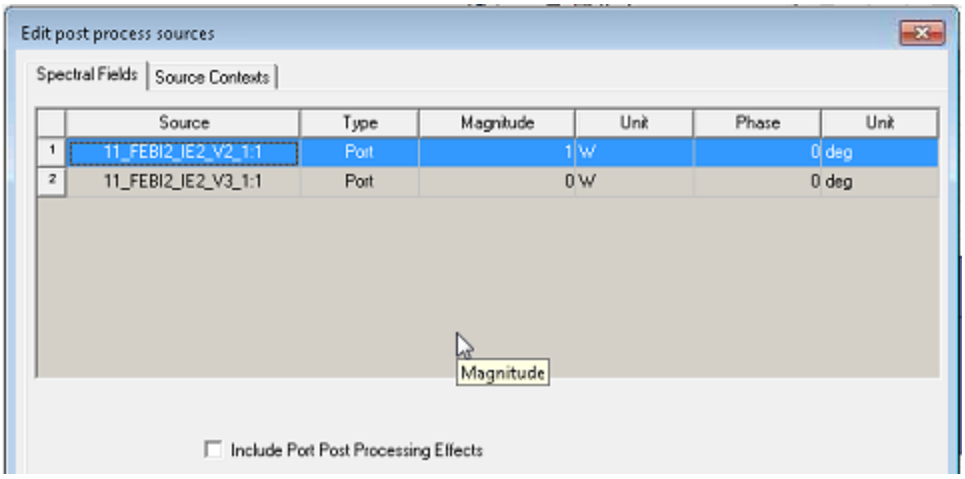
- Copy/Paste component
- Delete component
- Arrange (Move, Rotate, Mirror) component
- Duplicate component (Along Line, Around Axis, or Mirror)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit Attributes for Display Wire frame, Color, and Transparency.
- Use **Edit** > **Select** > **Submodel** to quickly select component
- Edit excitation Post Processing such as deembed 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit Attributes for Display Wire frame, Color, and Transparency
- Use **Edit** > **Select** > **Submodel** to quickly select component

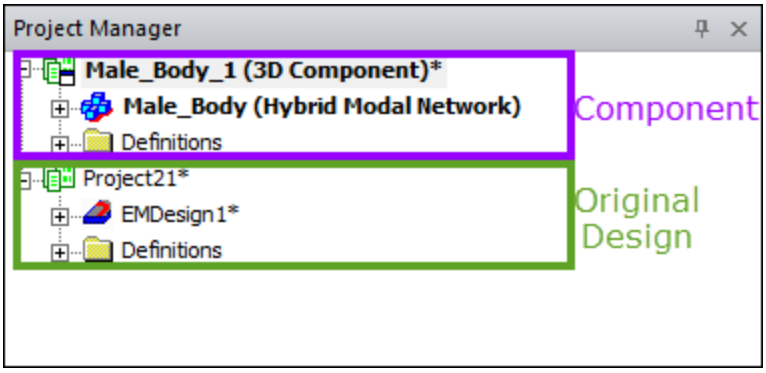
Editing a 3D Component Definition

> **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.

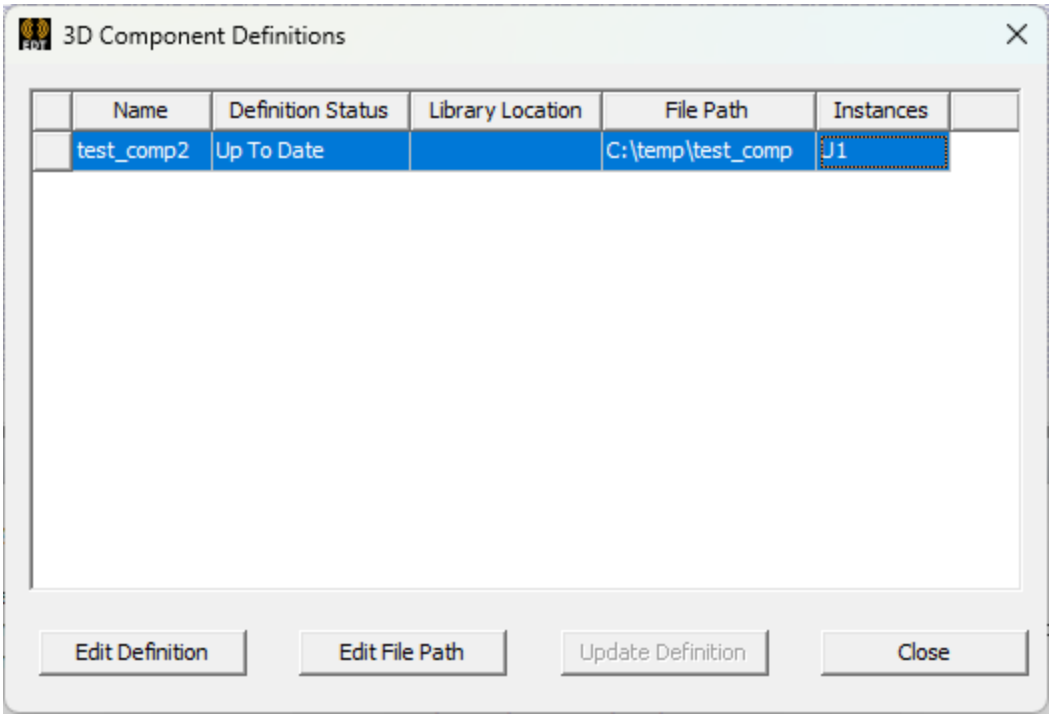
The **Edit Definition** command creates a new project, named for the component, in the **Project Manager** window.



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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

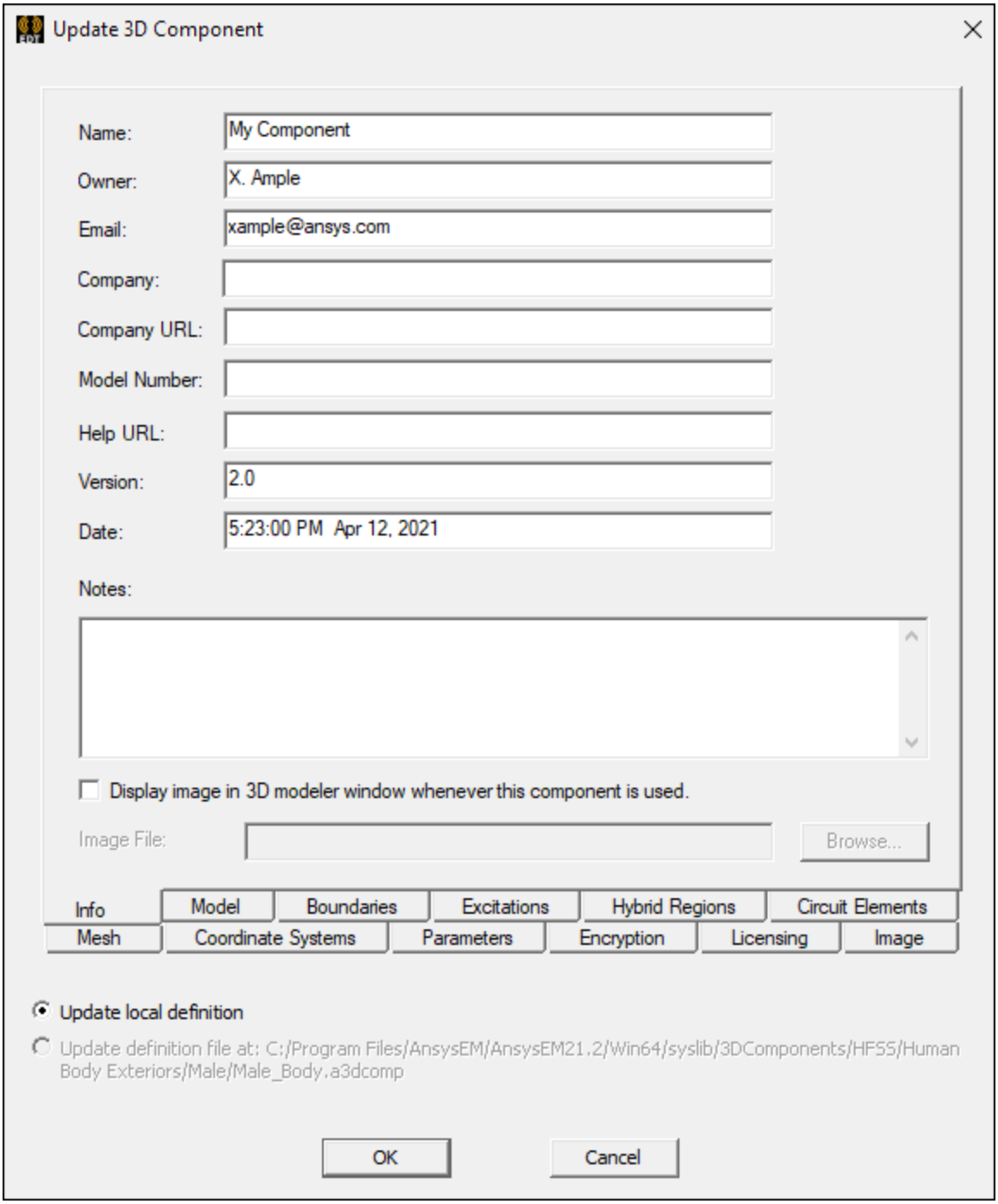
1. Navigate to the **Edit Definition** window by following one of the following paths:
   - From the **History Tree**, right-click the component and select **Edit Definition**.
   - From the **History Tree**, right-click an instance under the component and select **Components** > **Edit Definition**.
   - 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 2026 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 2026 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.

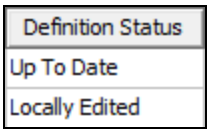
6. Click **OK**.

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.

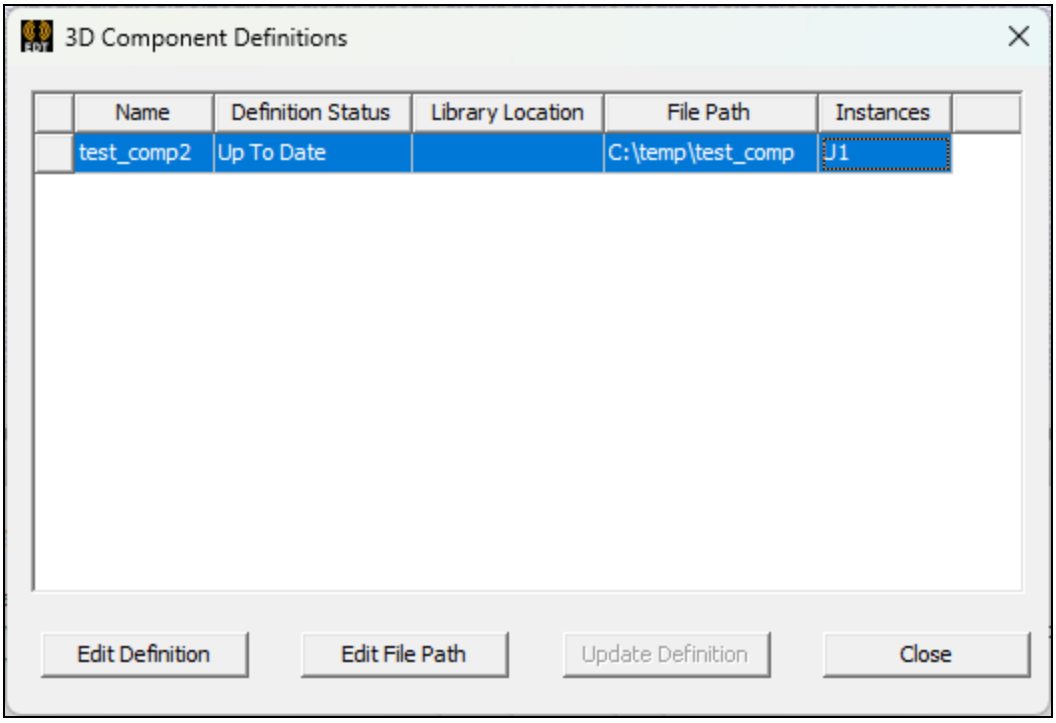
2. Select the desired component and click **Update Definition**.
3. Click **Close** to exit the window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Replacing 3D Components

3D components (and/or layoutcomponents) can be replaced everywhere they have been placed in a design simultaneously from the **3D Component Definitions** window. Complete these steps to make a replacement.

1. Navigate to **Layout** > **3D Component Definitions** to open the **3D Component Definitions** window.
2. Highlight a component and click **Edit File Path** to open an explorer window.



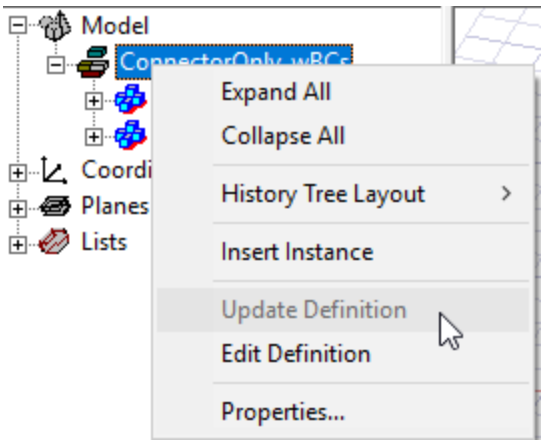
3. Navigate to the new component and either select it and click **Open** or double-click it to immediately replace the selected component. Keep in mind the following:

- All instances of the definition within the active design are replaced.
- All transformations (i.e., modifications) to the instance(s) are retained.
- All ports previously added to the instance(s) are also retained.
- Names are not retained.
- The replacement is irreversible.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

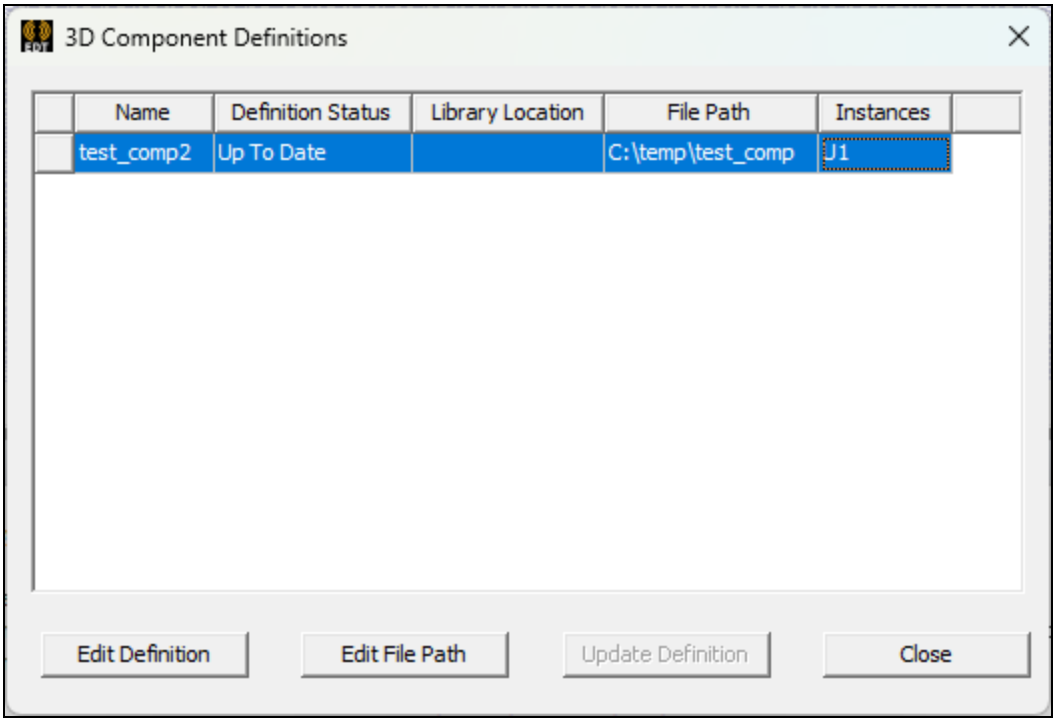
Updating a 3D Component Definition

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 2026 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.

Components 2.0 (Beta)

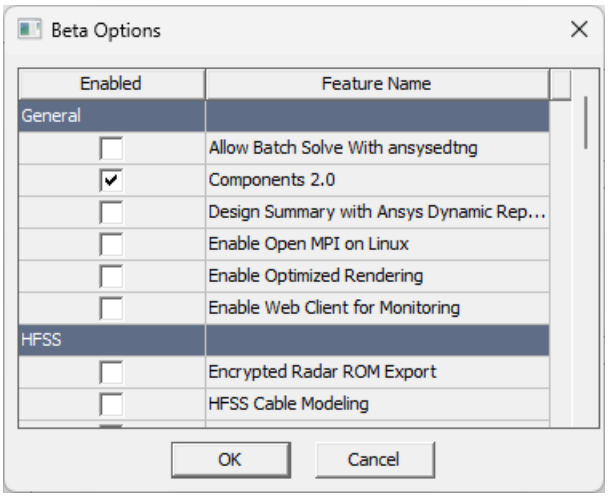
A 2.0 component is a project-level definition that can include one or more 3D designs. It can be thought of as a collection of designs where different solvers can be used to solve for same component geometry. The individual designs in the 2.0 component are referred to as representations. Encryption can be added to the individual representations as they are added to the component, and the component as a whole can be encrypted.

The 2.0 component has its own file extension (.acmp2), and can easily be shared without creating an archive. If you create an archive of a project with a 2.0 component, it will automatically include the .aedb folder so that the component definition is included in the archive.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To access the Component 2.0 functionality, you must check the beta feature in the Beta Options dialog box:

1. Open the **Tools** > **Options** > **General Options** window.
2. Click the **Beta Options** button.
3. From the General options, select **Components 2.0**.



To understand how to use a 2.0 component, see the following topics:

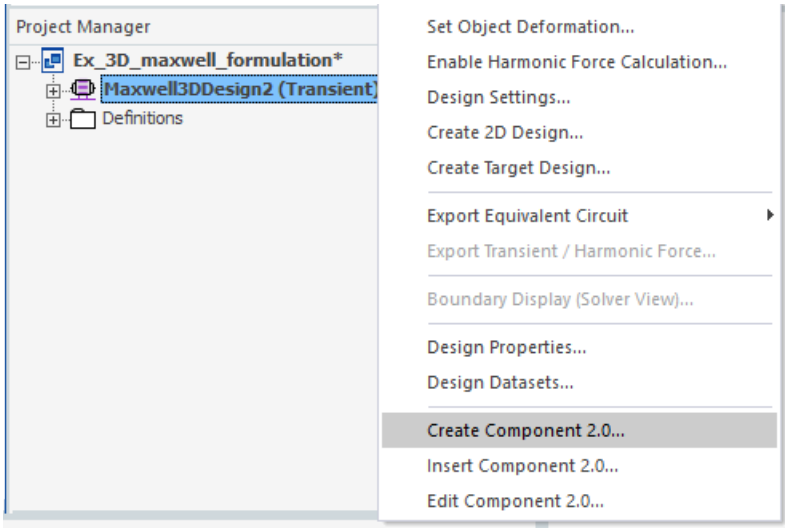
- Creating a 2.0 Component
- Component 2.0 Window
- Editing a 2.0 Component
- Inserting Instance from a 2.0 Component

Creating a 2.0 Component

To create the 2.0 component:

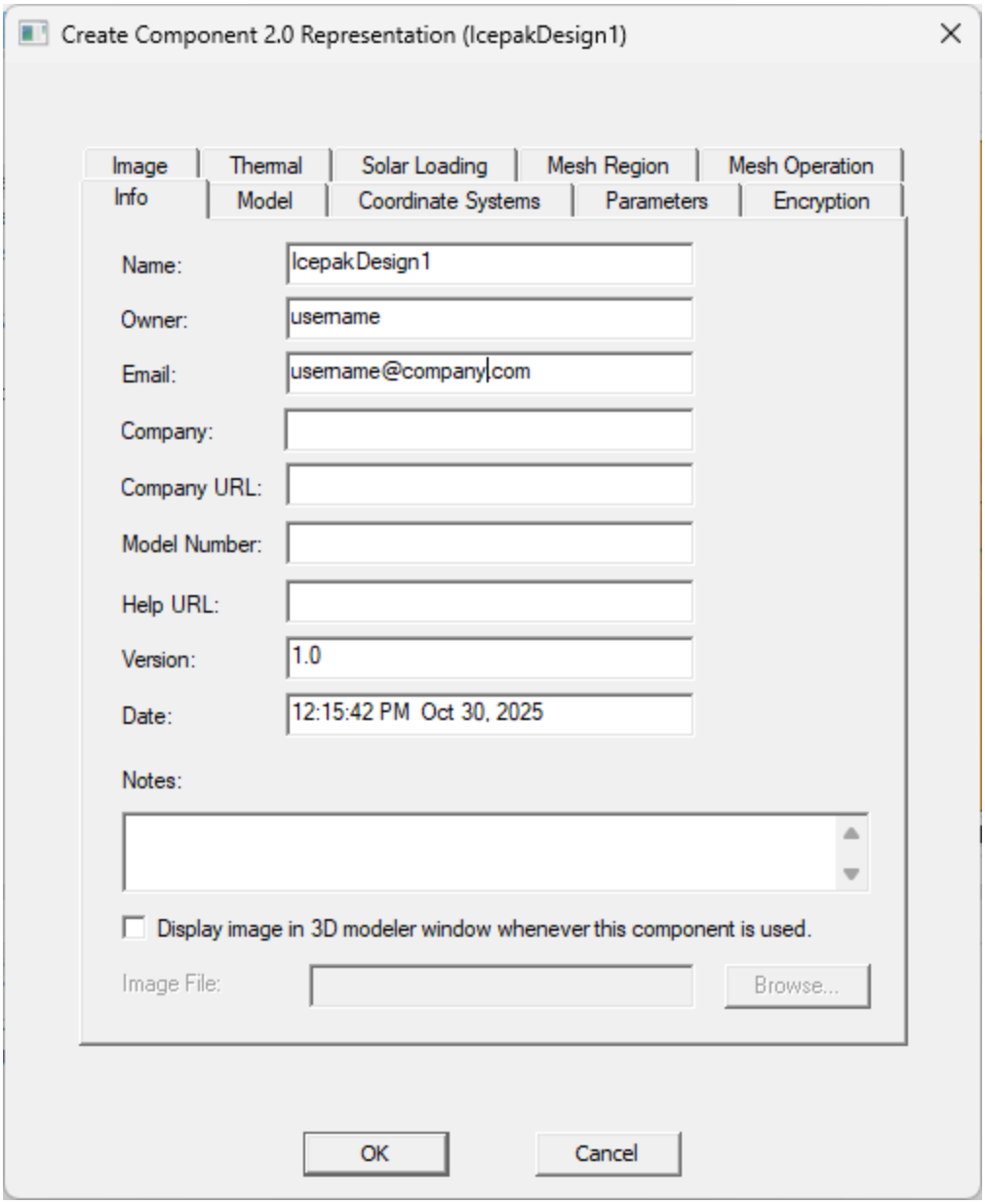
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right click on a design in the Project Manager tree, and select **Create Component 2.0**:



Note that when you first create the 2.0 component, the window that opens is labeled **Create Component 2.0 Representation** because the component does not exist until you add the first design, which is referred to as a representation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. The window opens to the **Info** tab, Use this tab to view and edit component information, as well as to add notes. The *Name* in the first field is used when the representation is inserted into a design. Other fields include *Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.
3. To include an image (typically a company logo) to display in the upper right area of the 3D modeler whenever the representation is used, check the **Display image** box to enable the **Image File** field, and click **Browse** to navigate your file system to the desired image file in .bmp format.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Configure the other tabs to determine what is included in the representation. **Image**, **Model**, **Coordinate Systems**, **Parameters**, and **Encryption** tabs are included for all design types. Other tabs are determined by the design type. These tabs configure the geometry and conditions that will be included in the design representation; see the topics that follow.
5. Click **OK** in the Create 2.0 Component Representation window; the Component 2.0 window will open listing the new representation.
6. Click **OK** in the Component 2.0 window to open a File Save dialog. Assign a file name to the Component 2.0 file. Note that it will have a .acmp2 extension.

Add Representations to a 2.0 Component

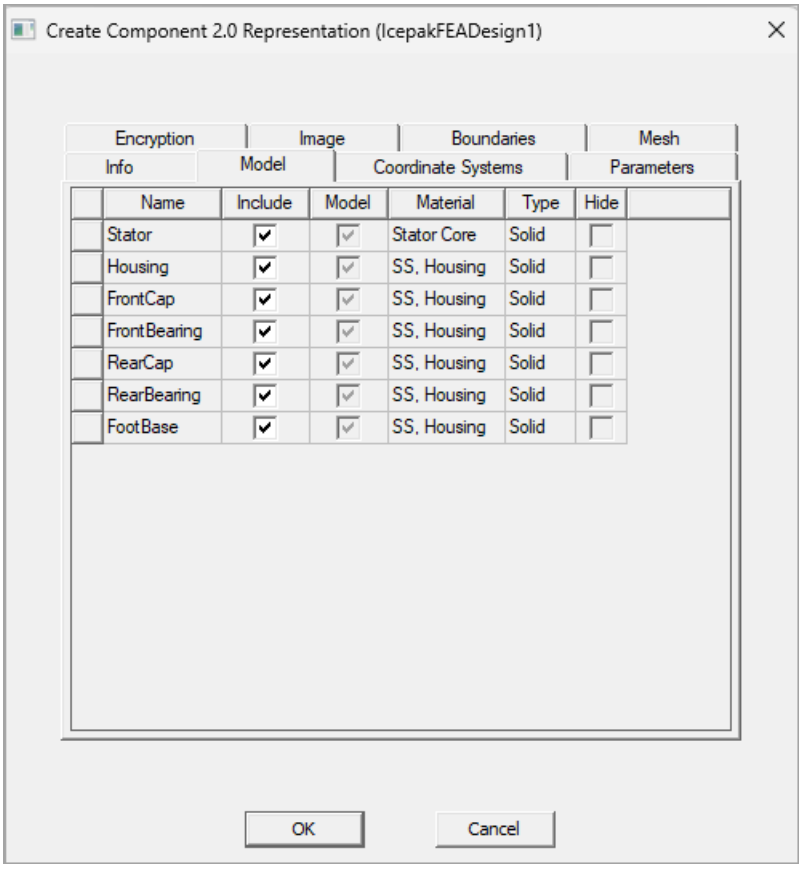
Once you have created a 2.0 component, you can add more designs to it by right clicking on the desired design and selecting **Edit Component 2.0**; see Editing a 2.0 Component for more details.

> **Note:**
>
> Once you have added a design representation to the 2.0 component, it is no longer linked to the original design; it will not update automatically if the original is edited.

Create Component 2.0 Representation Model Tab

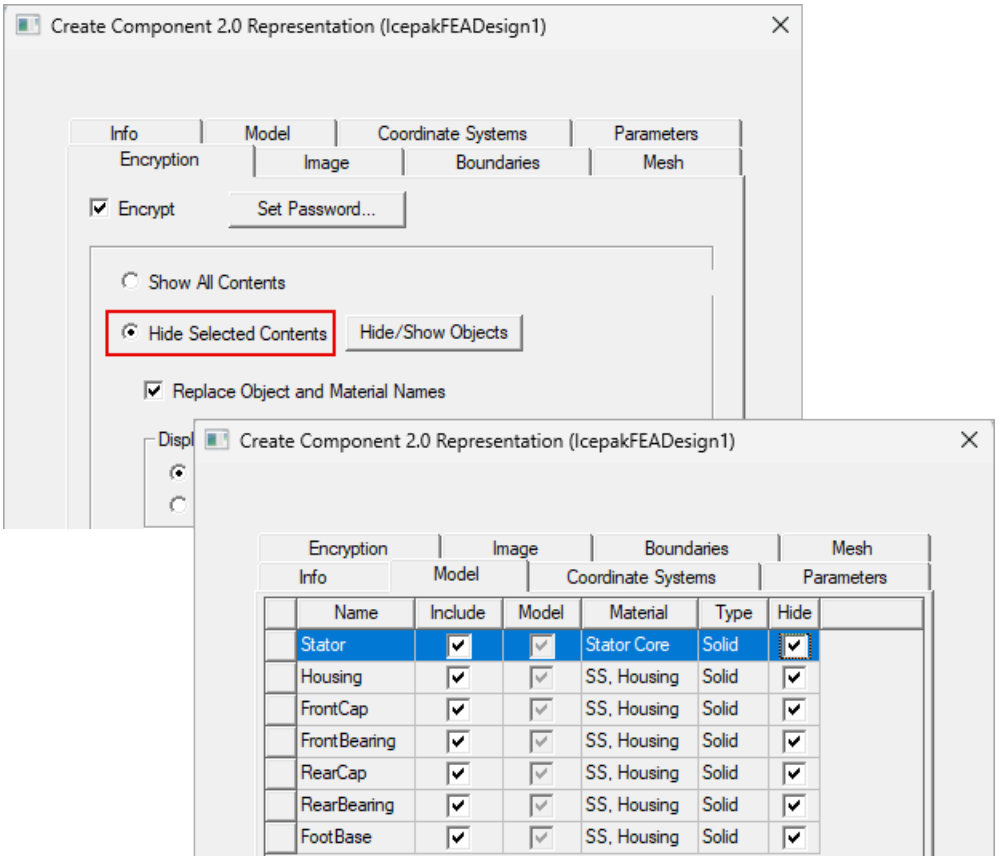
The **Model** tab lists all of objects in the design you selected before you invoked **Create Component 2.0**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Use the **Include** check box to include objects in the component (selected), or to Exclude them from the component (cleared).
- The **Material** and **Type** columns are read only.
- For Encrypted components, you can also choose 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

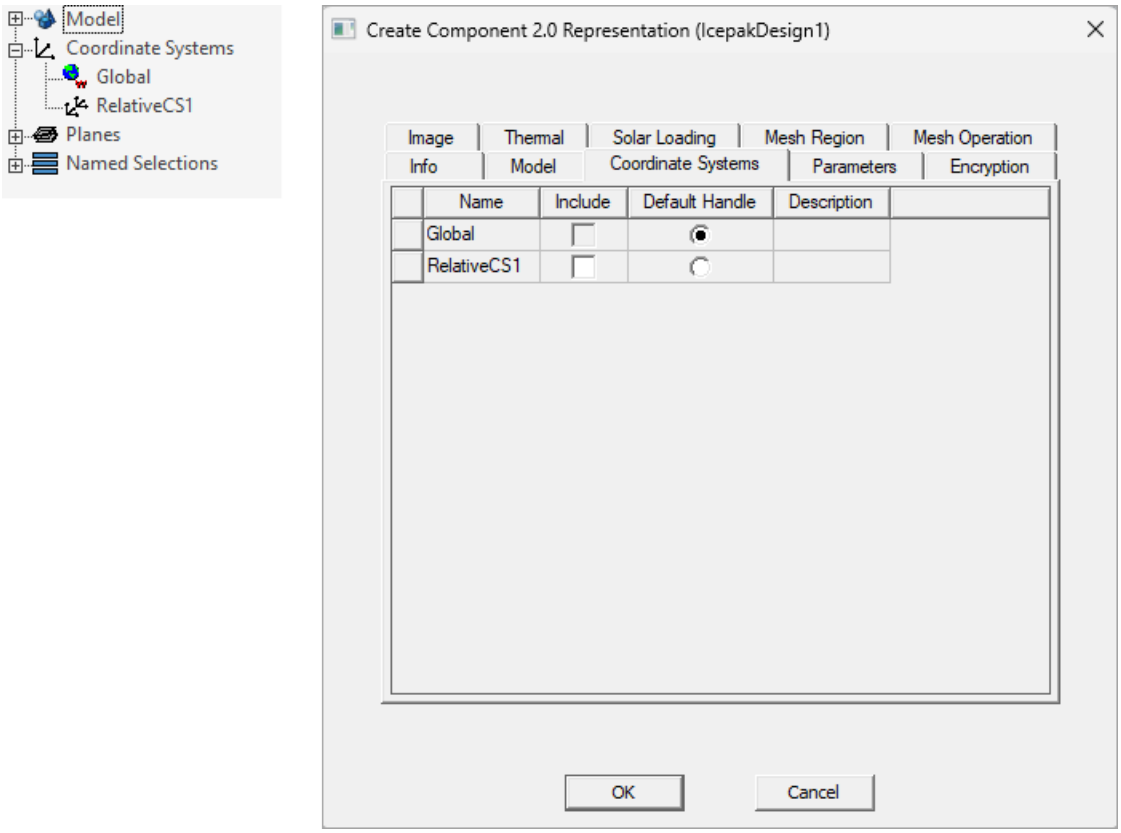


Create Component 2.0 Representation Coordinate Systems Tab

The list is populated with all the Coordinate Systems in the selected design. By default, only the Coordinate Systems used to define the object orientations are included.

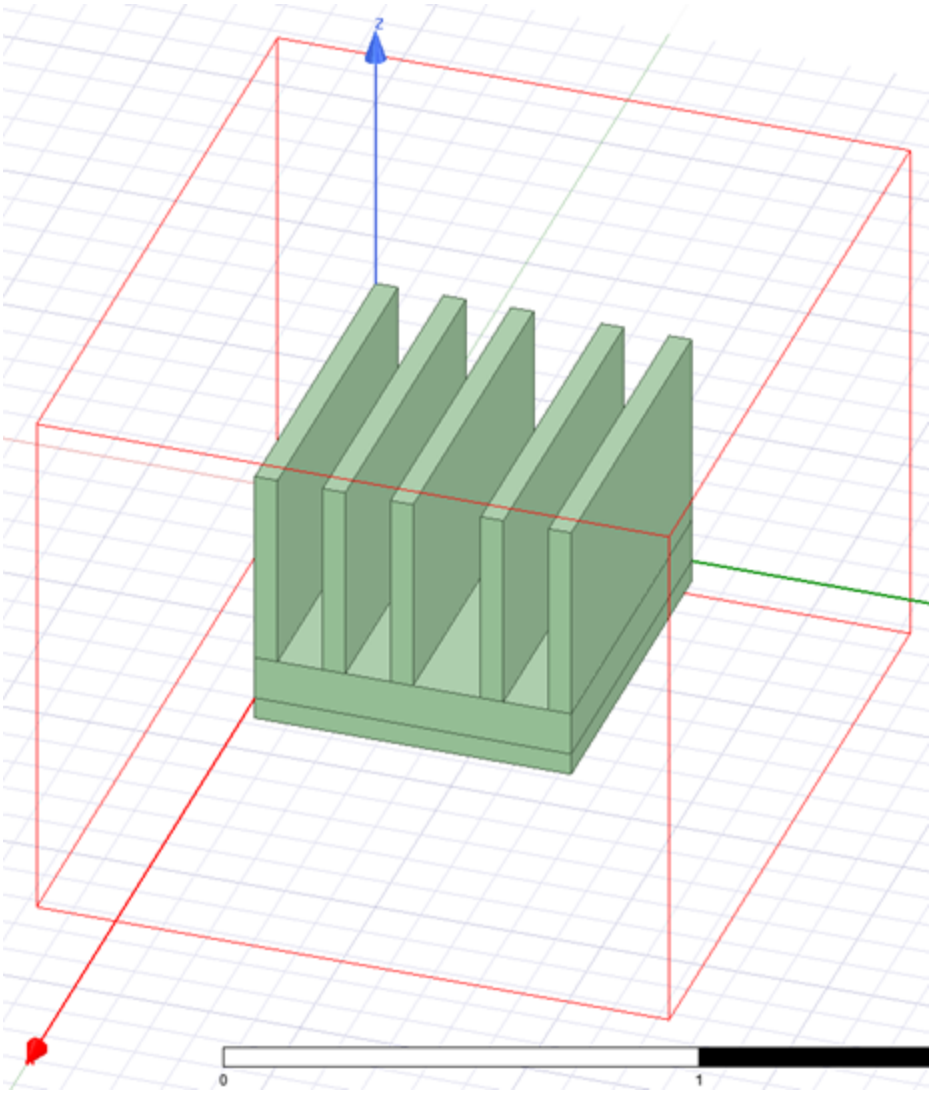
- Use the **Include** check box to include coordinate systems in the 2.0 component representation(selected), or to exclude them from the component (cleared).
- You can select any of the CS as the default handle. When this representation is placed in a design, the handle CS is aligned to target CS in the design.
- 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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The specified default handle affects the orientation of the representation upon insertion:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

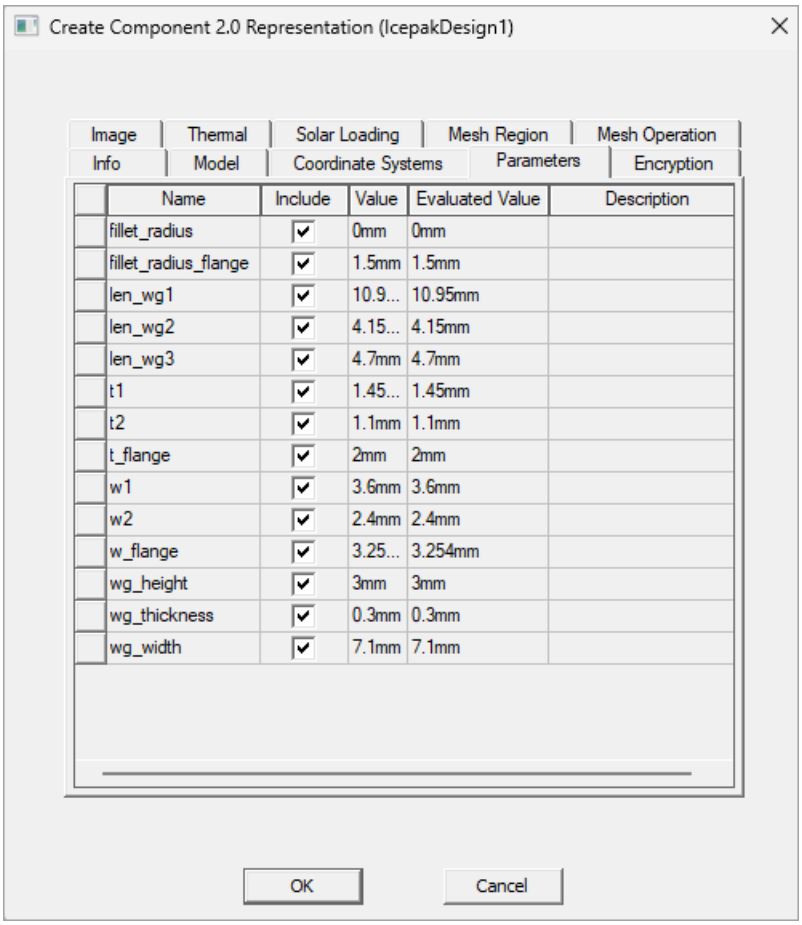


Create Component 2.0 Representation Parameters Tab

Representation 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 representation.

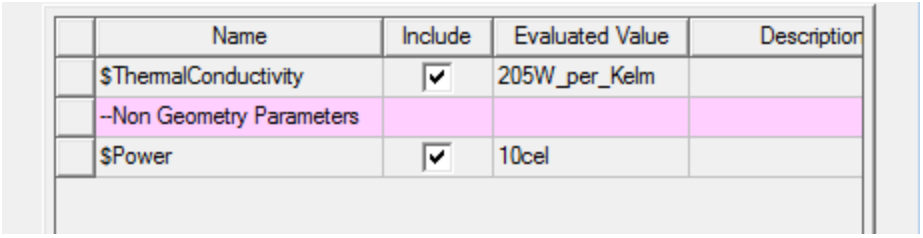
- Use the **Include** check box to include parameters in the representation (selected), or to exclude them from the representation(cleared).
- You can fill in the **Description** field, if desired.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Non-Geometry Variables in 2.0 Components

If you create non-geometry variables (for materials or boundaries), these are listed separately:

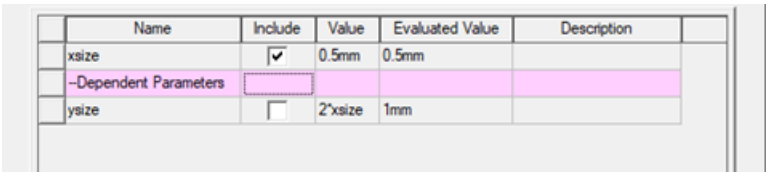


If you exclude any geometry, boundary, or material associated with a variable, the associated variable is automatically excluded on the **Parameters** tab.

Dependent Variables in 2.0 Components

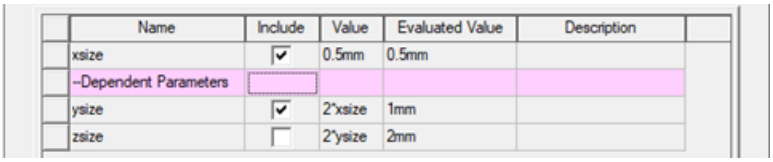
If you create dependent variables, you can choose to display the values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



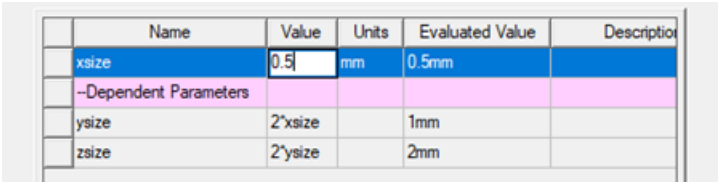
Name	Include	Value	Evaluated Value	Description
xsize	☑	0.5mm	0.5mm	
--Dependent Parameters				
ysize	☐	2*xsize	1mm	

If more variable dependency occurs for an included variable, such variables are also displayed with the option to include.



Name	Include	Value	Evaluated Value	Description
xsize	☑	0.5mm	0.5mm	
--Dependent Parameters				
ysize	☑	2*xsize	1mm	
zsize	☐	2*ysize	2mm	

When you insert a representation from a 2.0 component that includes dependent parameters, you can edit the independent variables, but can only view the evaluated values of the dependent parameters.



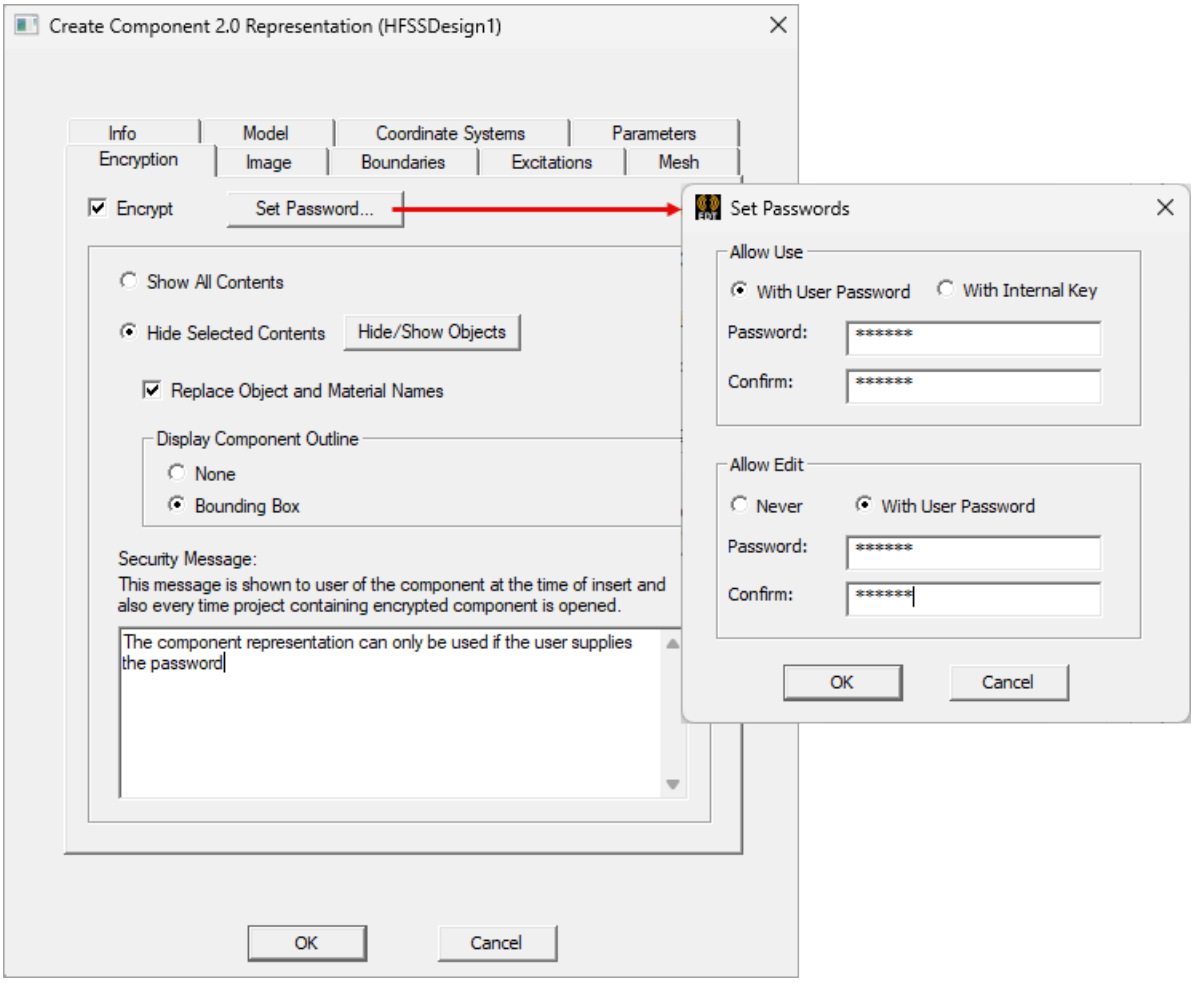
Name	Value	Units	Evaluated Value	Description
xsize	0.5	mm	0.5mm	
--Dependent Parameters				
ysize	2*xsize		1mm	
zsize	2*ysize		2mm	

Create Component 2.0 Representation Encryption Tab

The **Encryption** tab available in the Create Component Representation 2.0 window configures the encryption for the design representation being added to the component.

> **Note:** For details on the security for the 2.0 component, see Adding Security to a 2.0 Component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

There are some additional considerations with respect to using encrypted representations:

- The representation itself and its associated files will always be stored in encrypted form.
- When an HFSS or Maxwell design references an encrypted representation with hidden contents, it will also protect solver inputs and other project files containing data that could be used to extract representation information. However, note that solver inputs associated with SBR+ regions are not protected, even when encrypted representations 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.
- Q3D does not include extended protection of solver inputs and other project files.
- Icepak native designs (Fan, Heatsink, PCB), EM loss coupling, IcepakFEA-Icepak coupling, and **Start/Continue from a previously solved setup** (transient designs) are not supported for an encrypted representation.

Selecting the **Encrypt** check box enables all options and automatically launches the **Set Passwords** window.

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 2.0 component representations, the contents of the component file are encrypted to protect them from external viewing or editing.

- 2.0 components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.
- If the encrypted representation is created with an **Internal Key**, it can be used without additional authorization.
- If the encrypted representation is created with a **User Password**, that password must be entered when adding the representation to a design.
- It is not possible to edit the definition of an encrypted representation within Electronics Desktop unless **Allow Edit\ With User Password** was selected when defining the representation.
- If the **Show All Contents** option is selected, the geometry and other contents will be shown when the representation 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 representation ’s details. When this option is selected, hidden contents will not appear in the user interface.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All geometry will be hidden, except for objects that the user specifies as visible during representation 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.
- For representation 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 representation, but not the containing volume.

  > **Important:**
  >
  > Such plots may indirectly reveal representation 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 representation and have material of the surrounding material environment (e.g., air, vacuum).

  See Hiding Representation Content.

- 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 when the representation is in use.
- The **Security Message** field specifies text that is displayed at representation insertion and when the project containing the representation is opened.

Set Passwords

Clicking **Set Password** launches a window containing fields for setting the representation passwords. This is the same window that automatically launches upon enabling **Encrypt**.

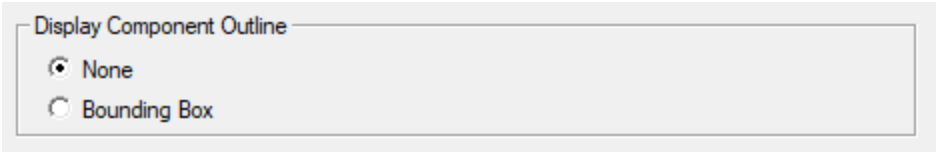
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Allow 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 Edit** – allows you to choose between never allowing edits to the representation 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 representation use and representation edit, the password does not have to be the same.

Hiding Representation Content

The **Hide Selected Contents** option allows selected parts of the representation to be displayed either as a **Bounding Box** or without an outline (**None**).

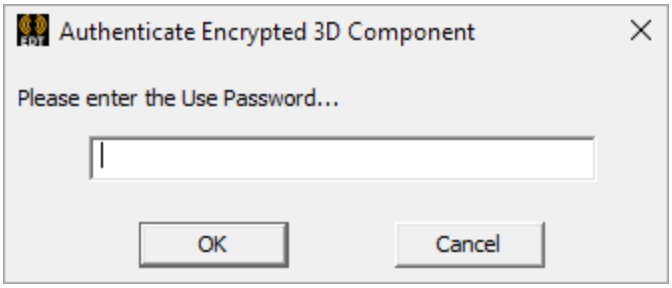


Once encryption is set, visibility can be set on a per-object basis using the **Model** tab. Preview the representation using the**Image** tab.

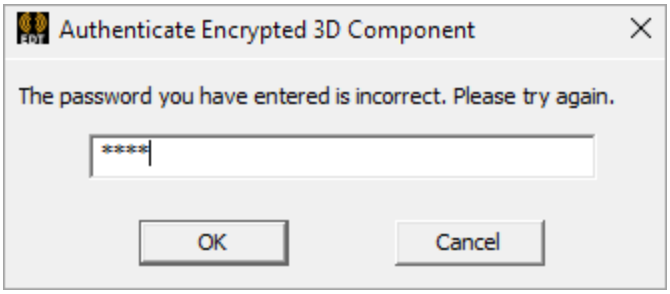
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inserting Encrypted Representations

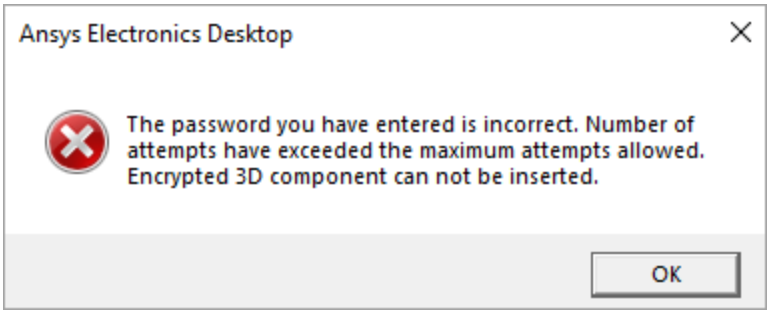
When an encrypted 2.0 component representation is added to a design, a prompt will appear asking for the password.



If the password is incorrect, a warning appears:



There is a three attempt maximum, after which a warning will appear and no further attempts may be made:



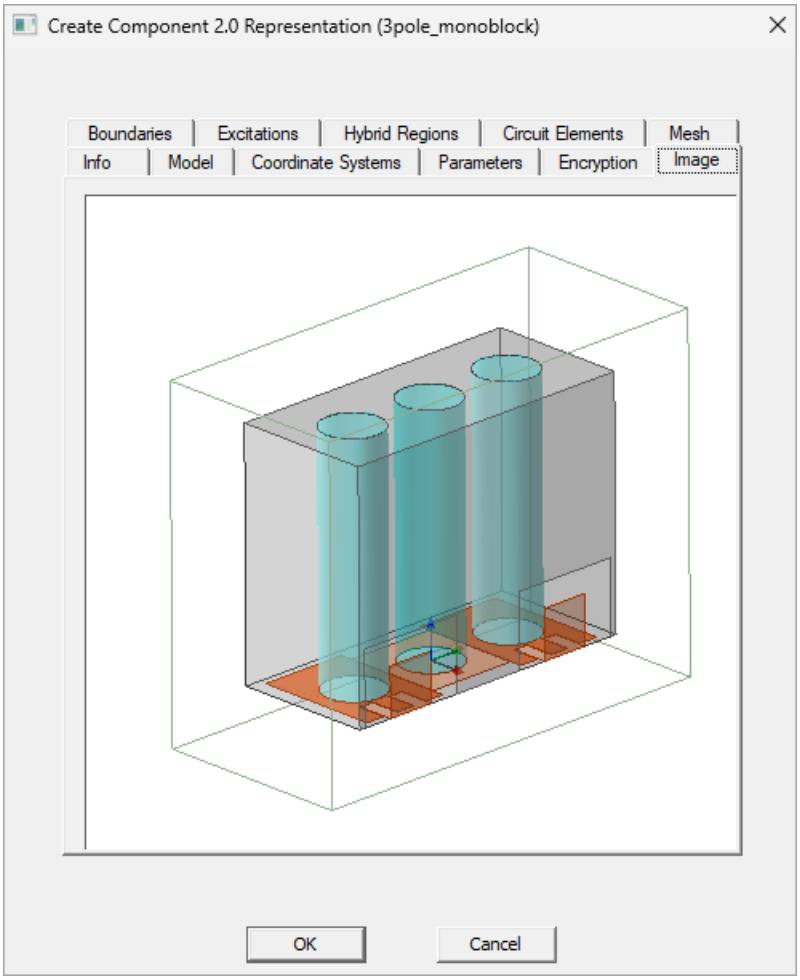
Copying and Pasting Encrypted Representations

An encrypted representation cannot be copied/pasted. They can be moved, and multiple instances of the representation can be added to a project design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Component 2.0 Representation Image Tab

The **Image** tab shows the preview image for the representation, based on the reference coordinate system. The image also responds when you **Include** or **Exclude** objects from the **Model** tab.



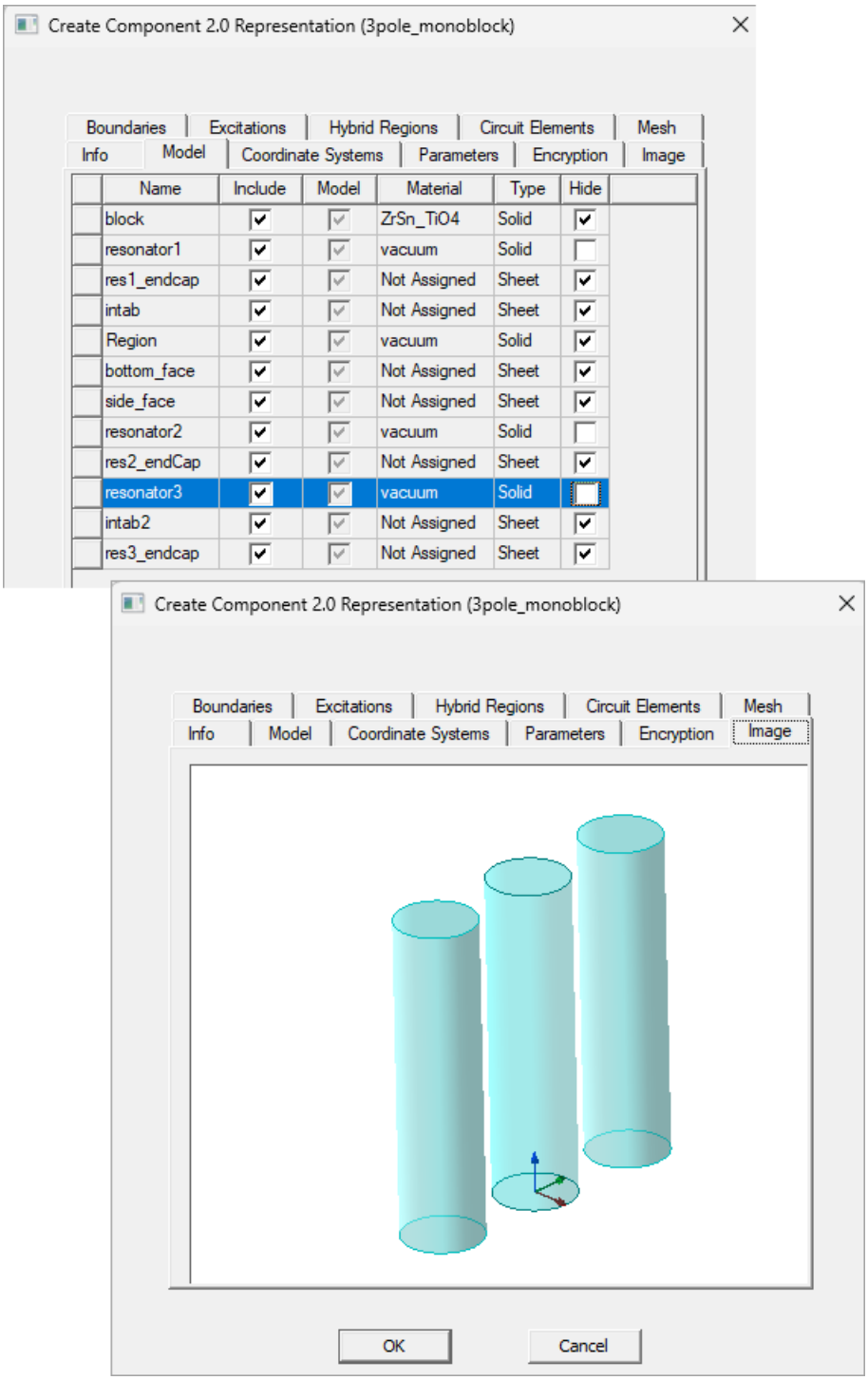
To move the model in the Image window, use the following mouse combinations:

- Alt + left click to rotate
- Shift + left click to move
- Middle button in mouse to scale

These changes will be saved to the component representation after you click **OK**.

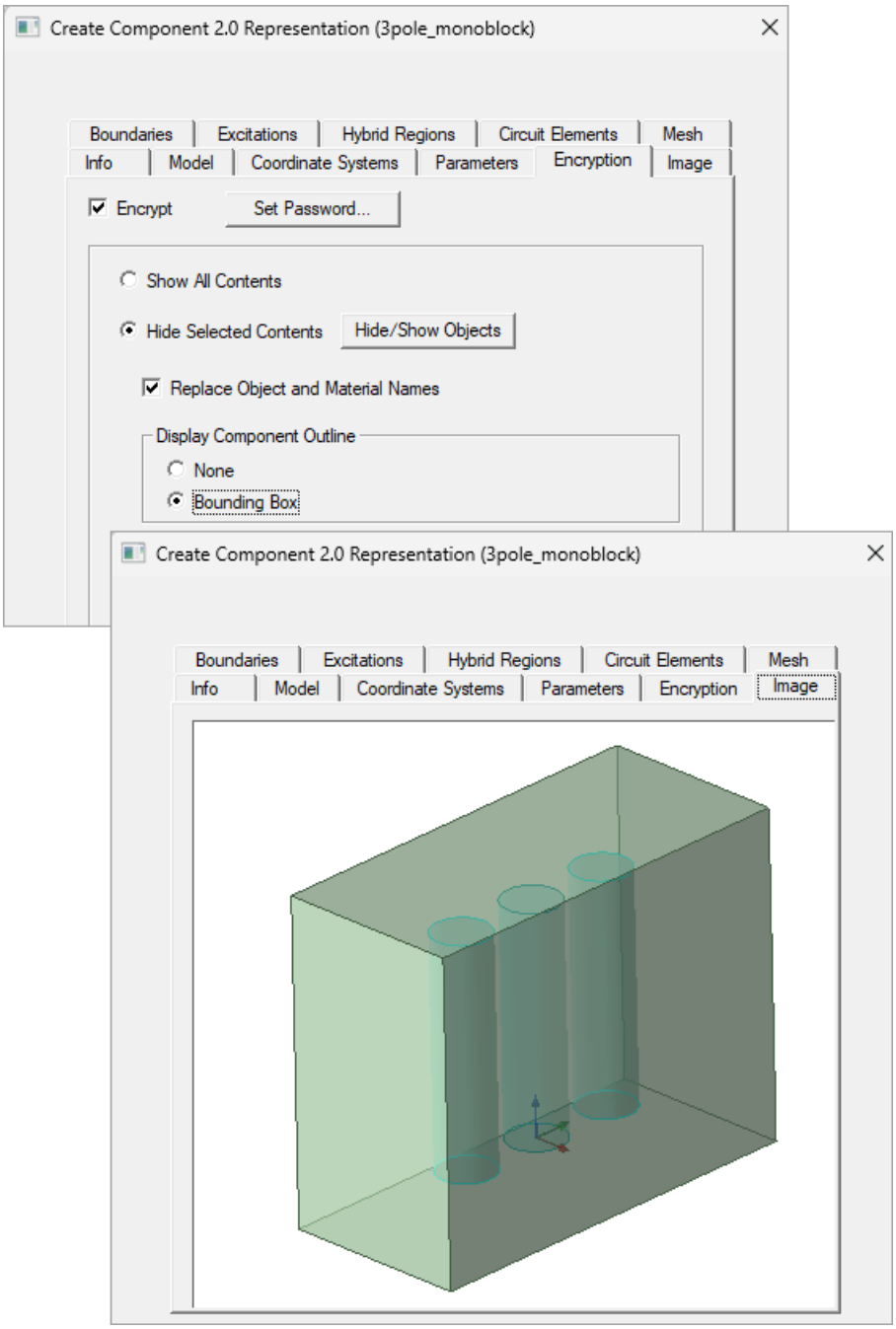
If you use the **Encryption** tab and **Model** tab settings to hide/show the model or specified model objects, the **Image** tab updates accordingly. You must use encryption to enable the **Hide** check boxes on the **Model** tab.

Ansys Electromagnetics Suite 2026 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 an object outline, the **Image** tab responds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

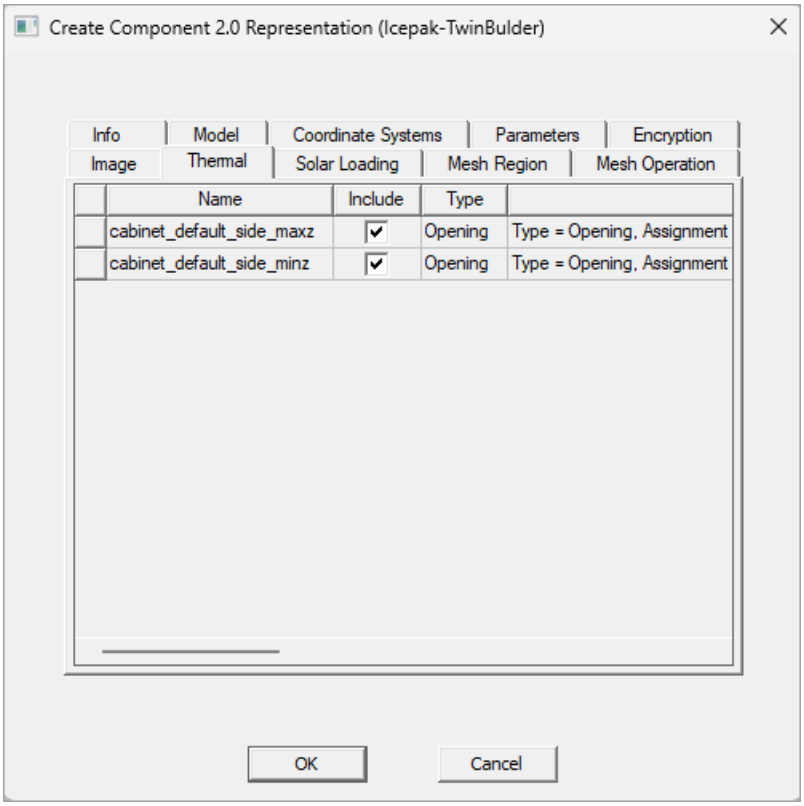


Create Component 2.0 Representation Thermal Tab

On the **Thermal** tab, you can view a list of thermal boundary conditions assigned to the geometry selected to create the 2.0 component representation. Use the **Include** check box to include or exclude the individual boundaries.

The Type and Description columns are read only.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

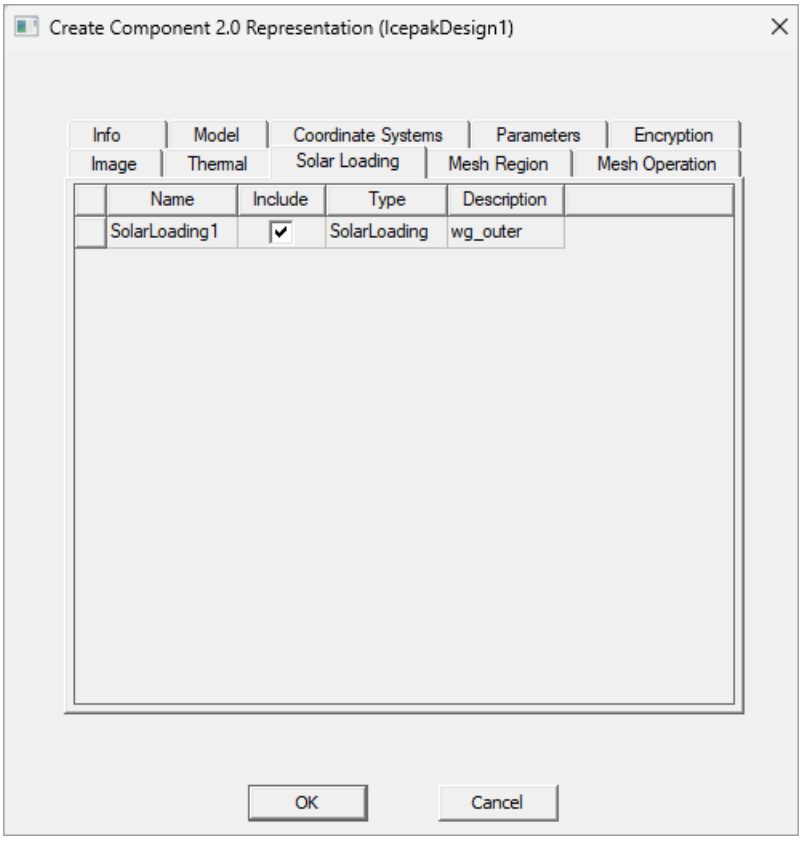


Create 2.0 Component Representation Solar Loading Tab

On the **Solar Loading** tab, you can view a list of solar loading boundary assigned to the geometry selected to create the 2.0 component representation. Use the **Include** check box to include or exclude the individual boundaries.

The Type and Description columns are read only.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

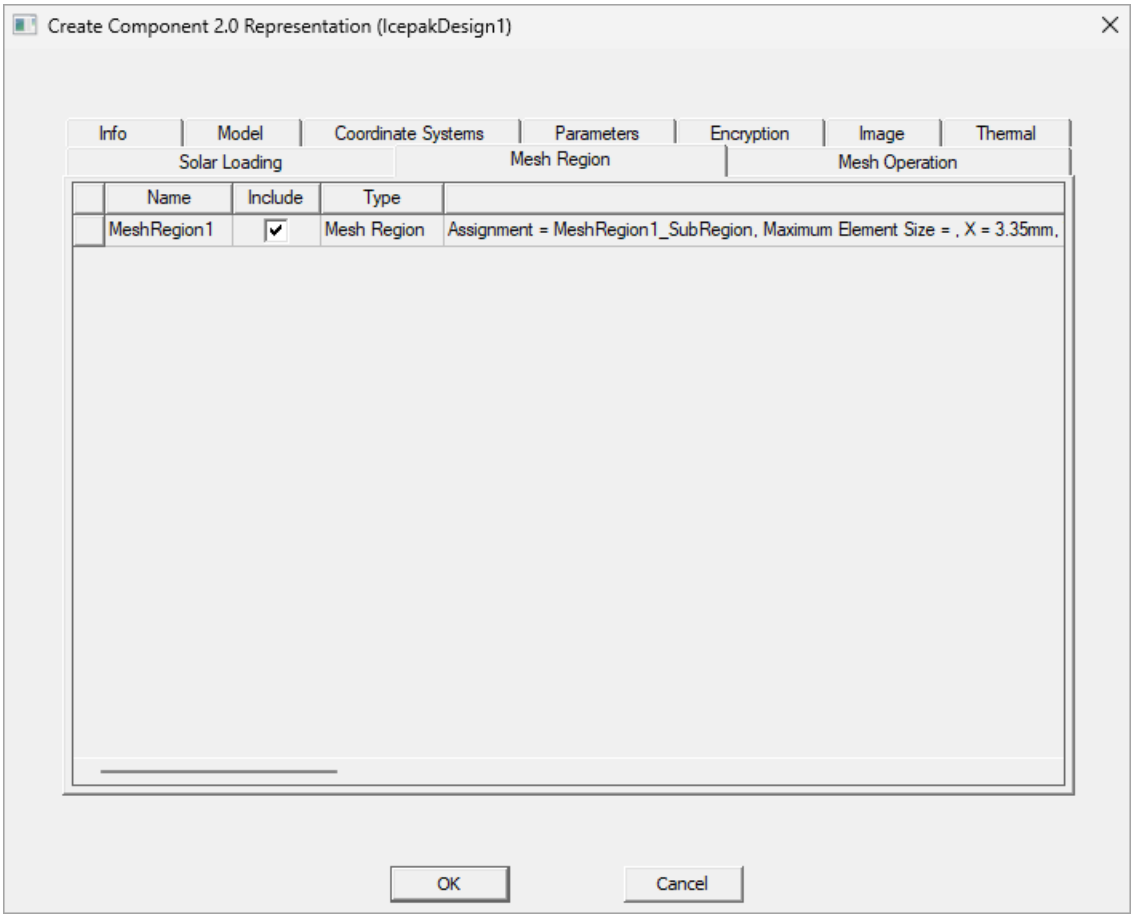


Create Component 2.0 Representation Mesh Tabs

For Icepak, the Create Component 2.0 Representation has two mesh tabs: **Mesh Region** and **Mesh Operation**.

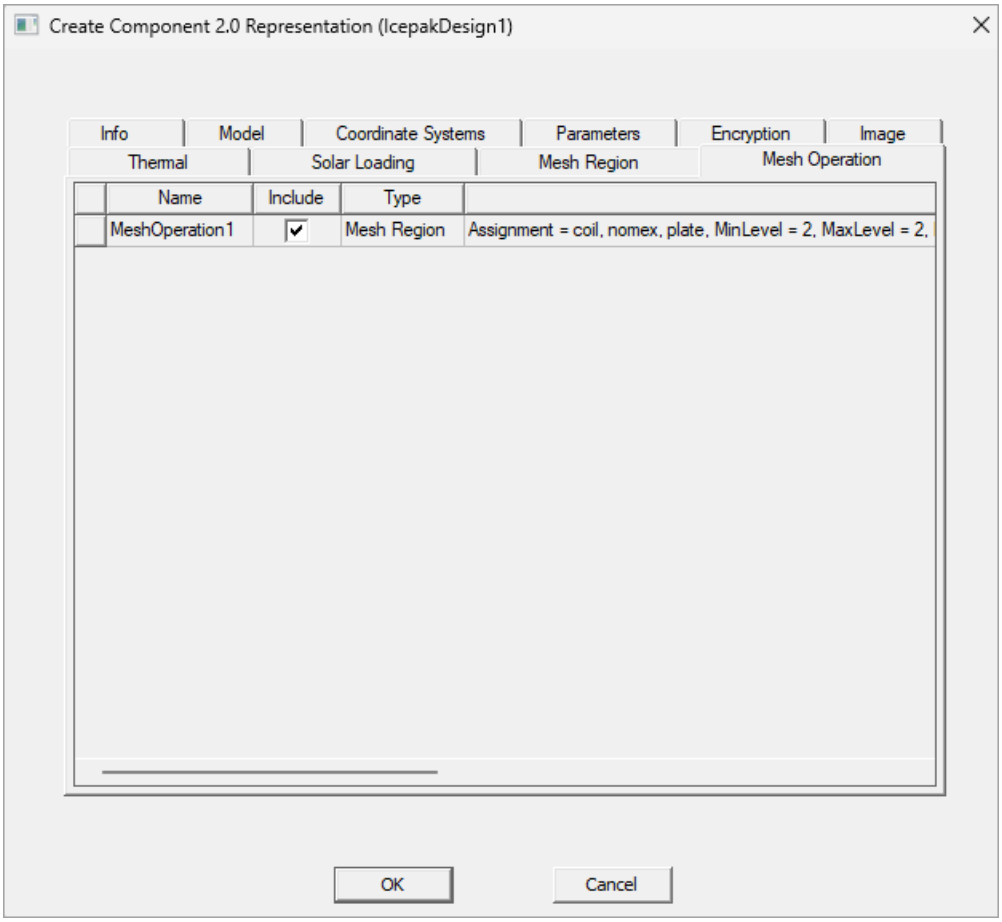
The **Mesh Region** tab has a list that is populated with all the mesh regions on the selected design:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Mesh Operation** tab has a list that is populated with all the mesh operations on the selected design:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



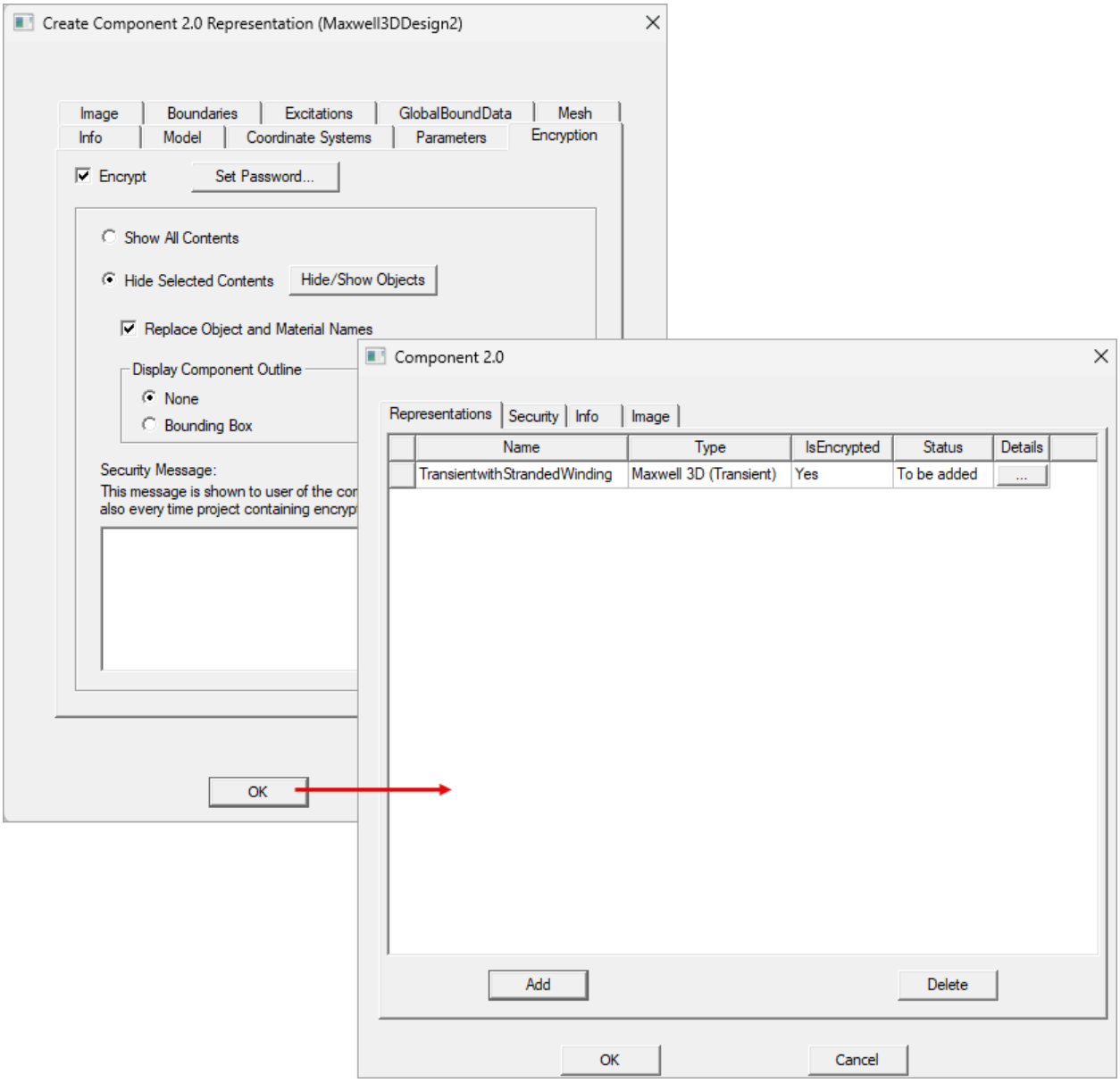
For both tabs:

- Use the **Include** check box to include mesh operations in the 2.0 component representation (selected), or to exclude them from the 2.0 component representation (cleared).
- *Type* and *Description* columns are read only.

Component 2.0 Window

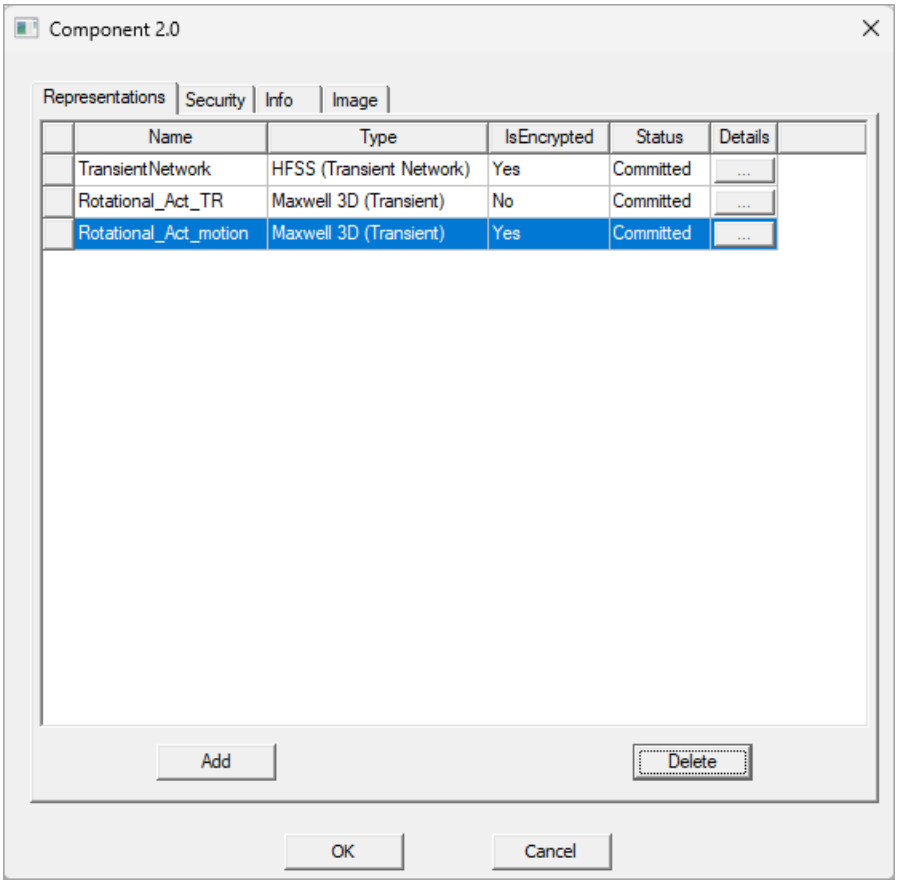
When you click **OK** in the Create Component 2.0 Representation window, the Component 2.0 window opens. This window gives user access to editing the component, to securing the component with encryption, and to entering info for the component. It can also be used to view images of all the representations in the component. This window does not exist until the user defines the first representation for the component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The same window can be opened from the **Edit Component 2.0** menu:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



It will also open if you select Insert 2.0 Component and specify a .acmp2 file.

This window includes the following tabs:

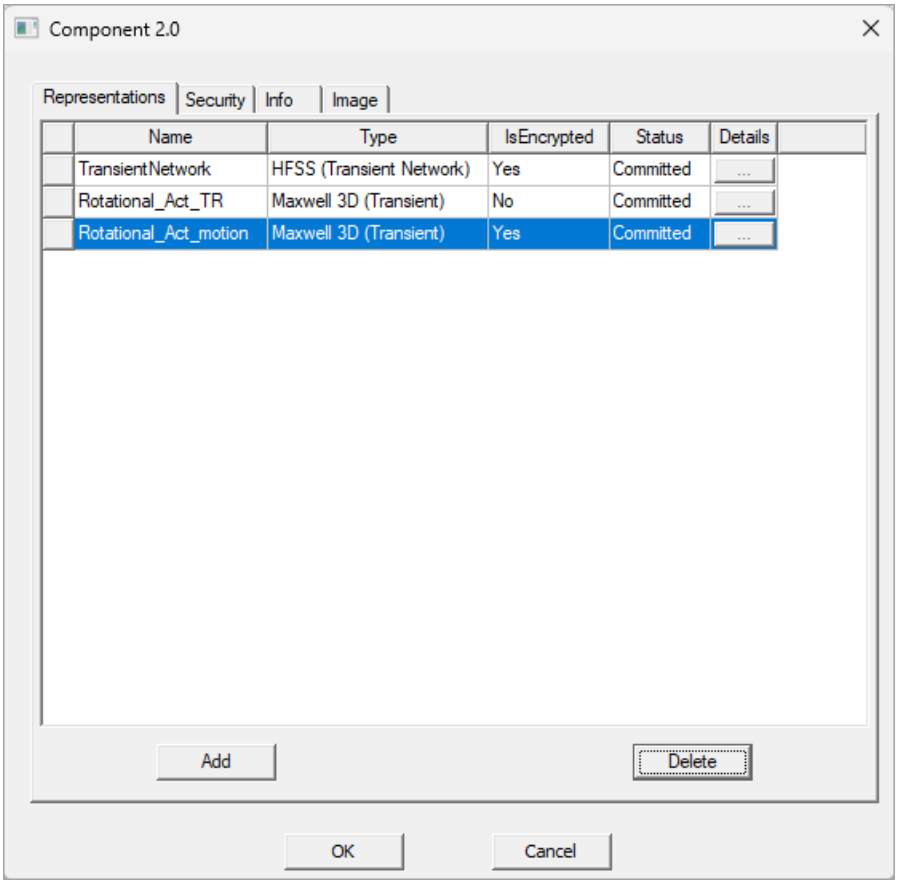
- Representations
- Security
- Info
- Image

These tabs are explained in the topics that follow.

Representations

The **Representations** tab lists all the design representations that have been included in the 2.0 component:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

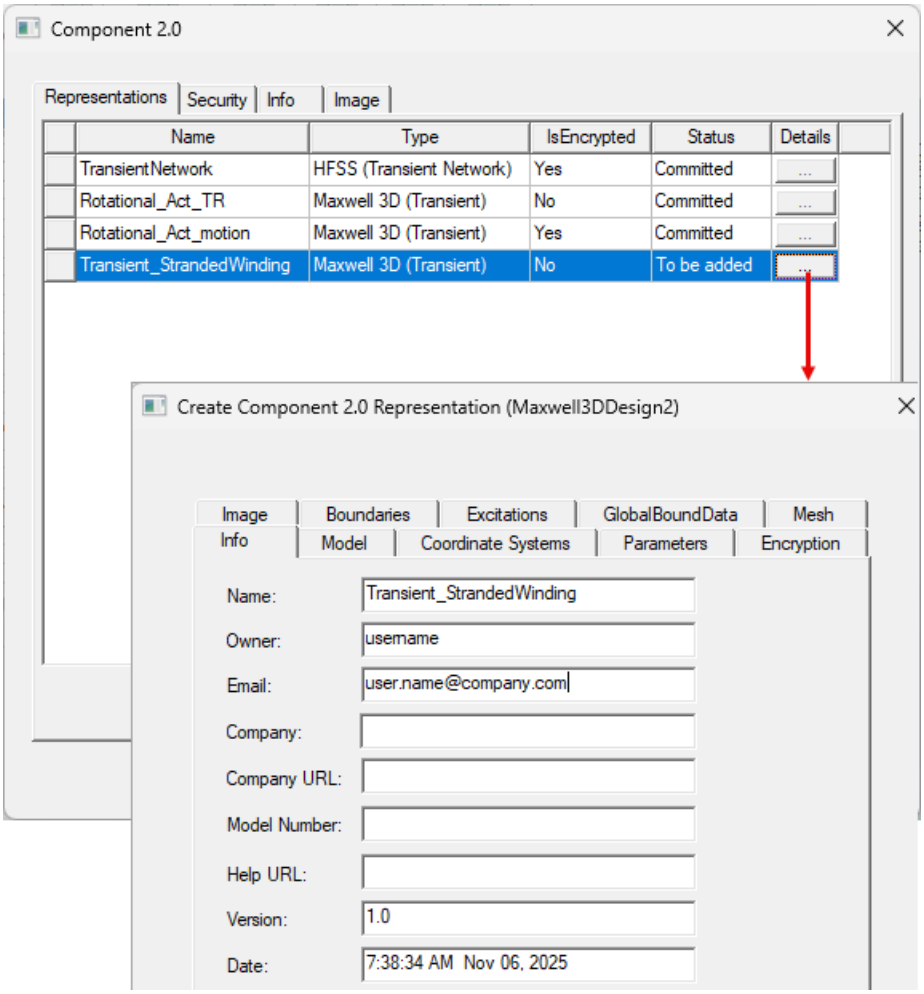


For each representation, it lists the following:

- **Name**: assigned in the Info tab of the Create Component 2.0 Representation window when the design was added.
- **Type**: solver type that was used in the creation of the original design.
- **IsEncrypted**: encryption status as assigned in the Encryption tab of the Create Component 2.0 Representation window when the design was added.
- **Status**: lists whether or not the representation has been saved to the 2.0 component (**Committed**). If the representation has not been added (by clicking on **OK** in the window), the status will be **To be added**.
- **Details**: If a representation has a **To be added** status, you can click on the button in the Details column to open the Create 2.0 Component Representation window for that representation; changes can be made to the representation from the window. Once a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

representation's status has changed to **Committed**, its Details button will be inactive.



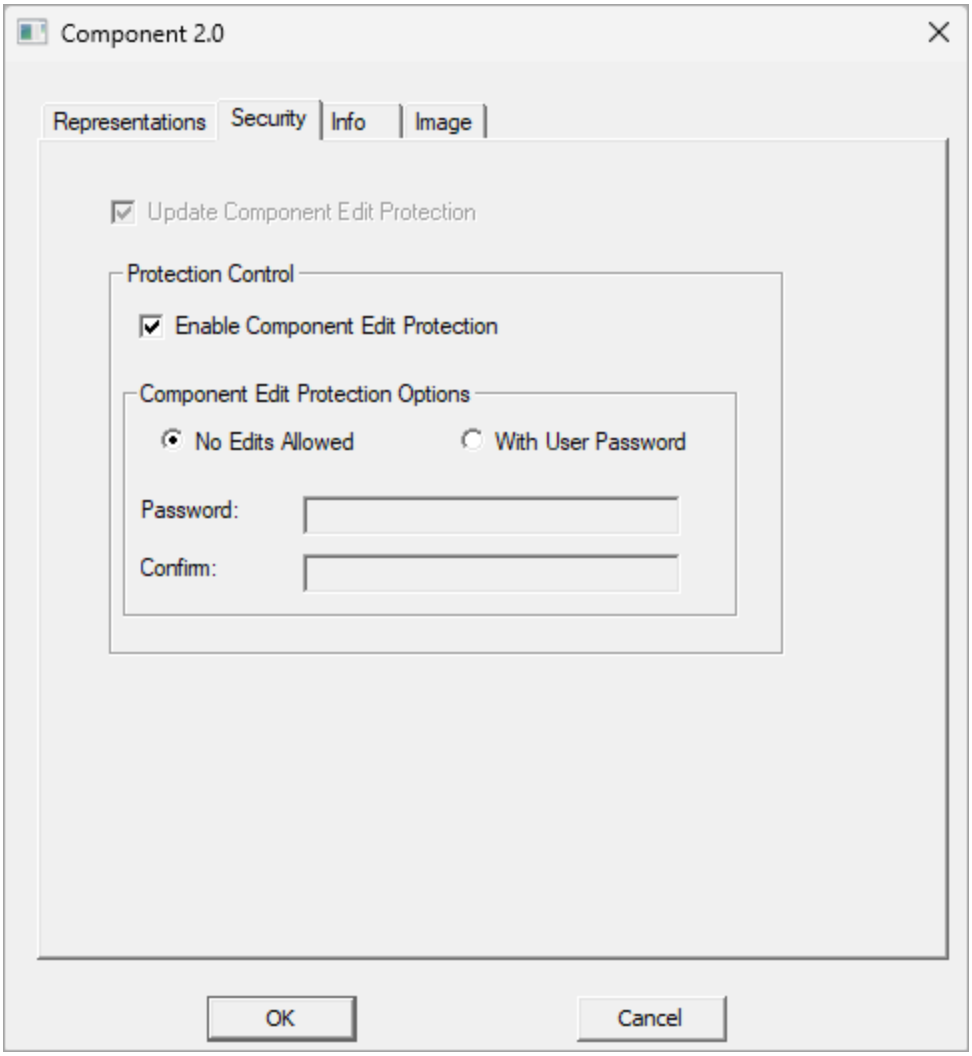
The **Add** and **Delete** buttons can be used to modify the list of representations; see Editing a 2.0 Component for more details.

Adding Security to a 2.0 Component

The **Security** tab of the Component 2.0 window is used to encrypt the component. In contrast to the Encryption tab in the Create 2.0 Representation window, this encryption applies to the entire 2.0 component, not to an individual representation.

If **No Edits Allowed** is selected, the user will not be able to add or delete representations in the component file. If **With User Password** is selected, the user can add or delete if the password is provided.

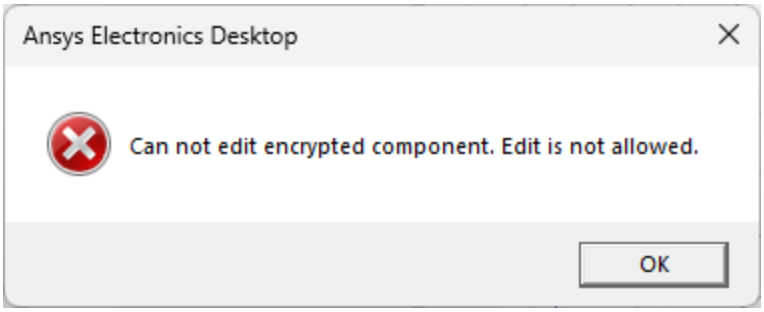
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



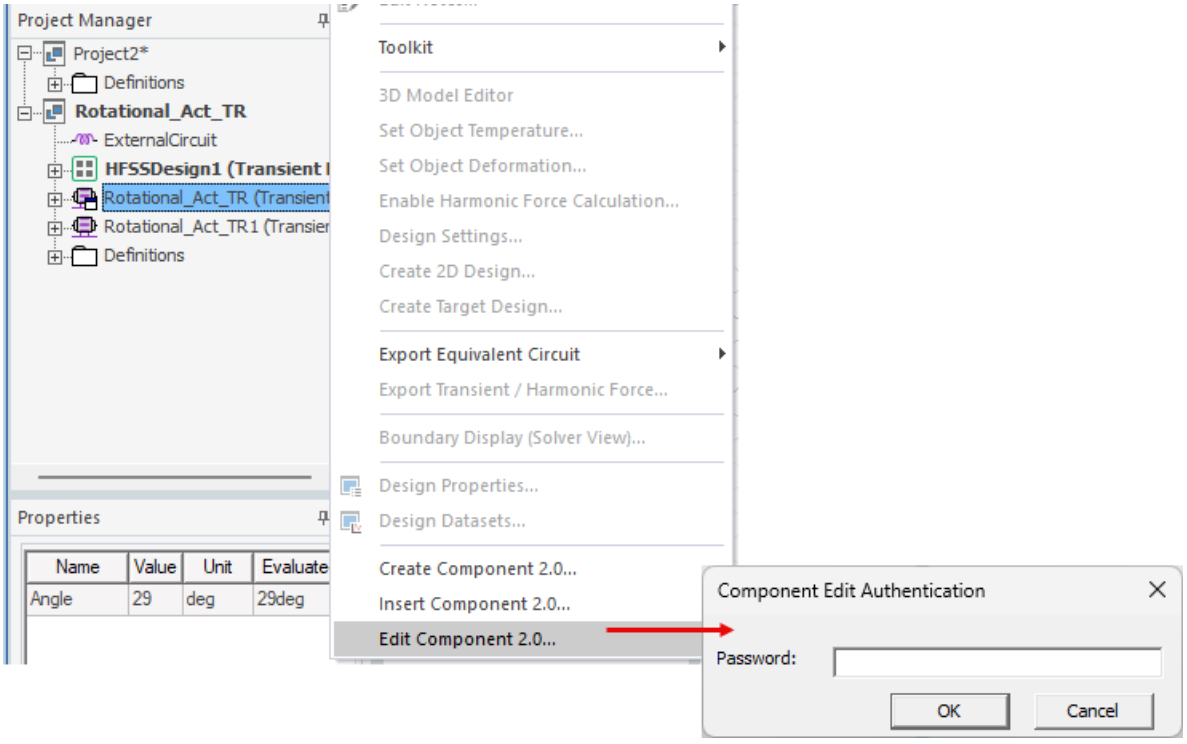
To add edit controls to the file, check the **Enable Component Edit Protection** option so the edit controls become active. If the 2.0 component already exists and you would like to change the security, first check the **Update Component Edit Protection** option, which will not be grayed out for an existing component.

When the **No Edits Allowed** option is selected and saved with the component file, if the file is selected from the **Edit Component 2.0** menu, Desktop will display the following message:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the **With User Password** is selected for the Component Edit Protect Options, when you select the encrypted component file from the **Edit Component 2.0** option, you will be prompted for the password:



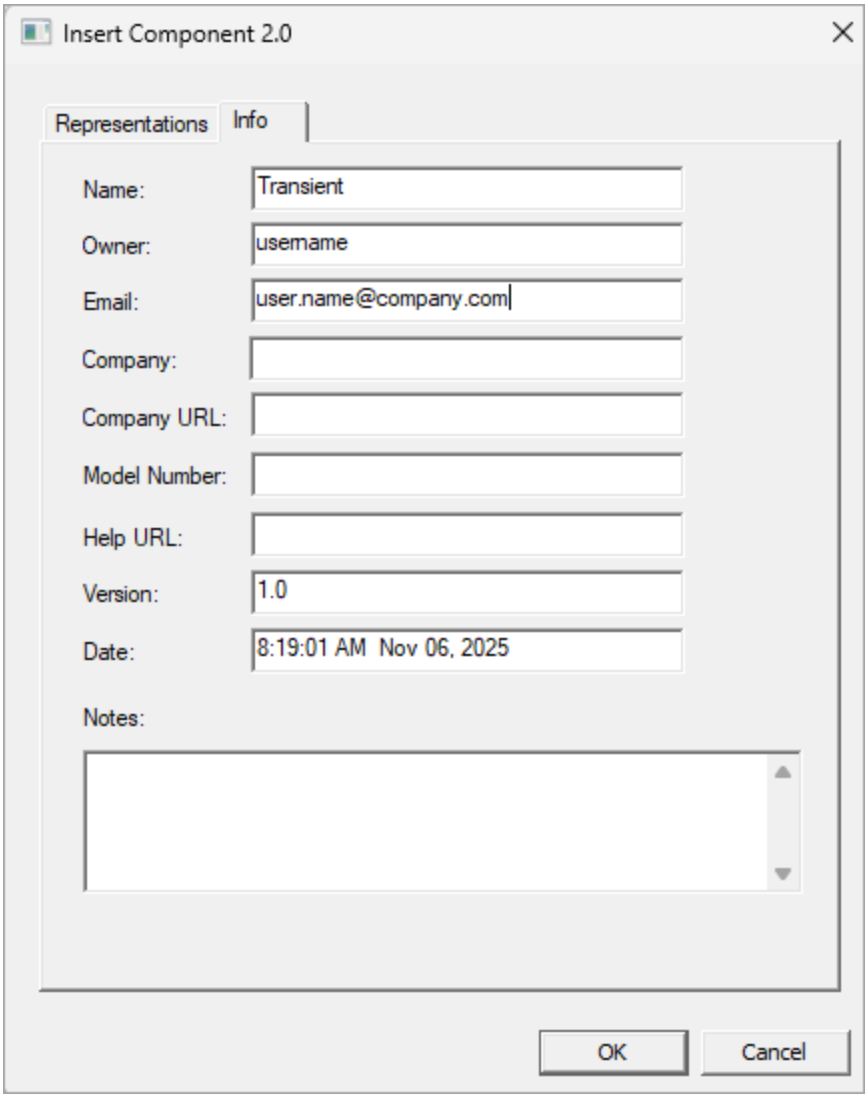
Important:

The component itself and its associated files will always be stored in an encrypted format.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

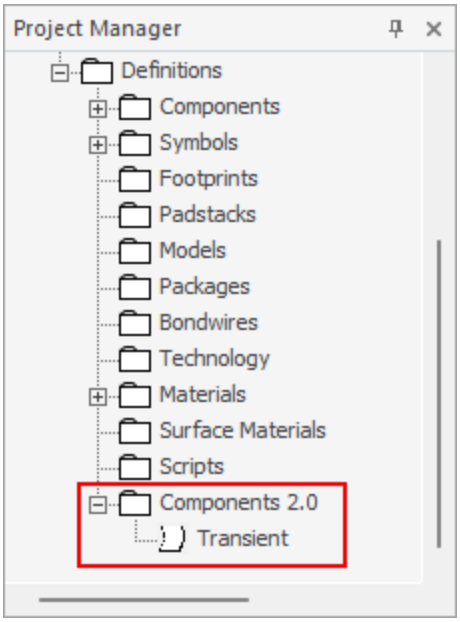
Info

The Info tab of the Component 2.0 window sets the information for the component file. From this tab, you can view and edit component information, as well as add notes. Fields include *Name, Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.



The name given in the **Name** field determines how the component is labeled in the Project Manager tree's **Definitions\Component 2.0** folder:

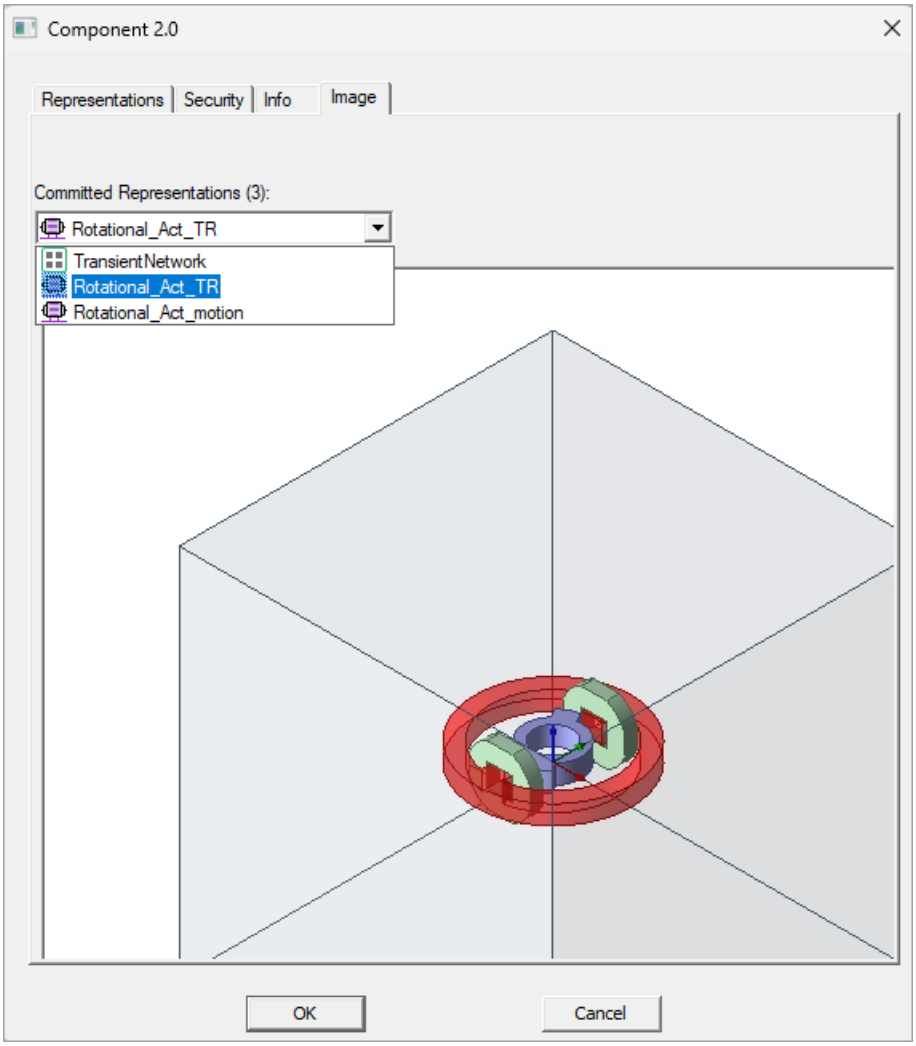
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Image

The Image tab of the Component 2.0 window offers a drop-down menu that includes all the representations in the component. Note that unlike the Image tab in the Create Component 2.0 Representation window, the representation image cannot be changed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Editing a 2.0 Component

After creating a 2.0 component, you can use the Edit function to add or delete design representations, change the component's information, or change its security.

Note that a representation cannot be edited once it is committed to a component.

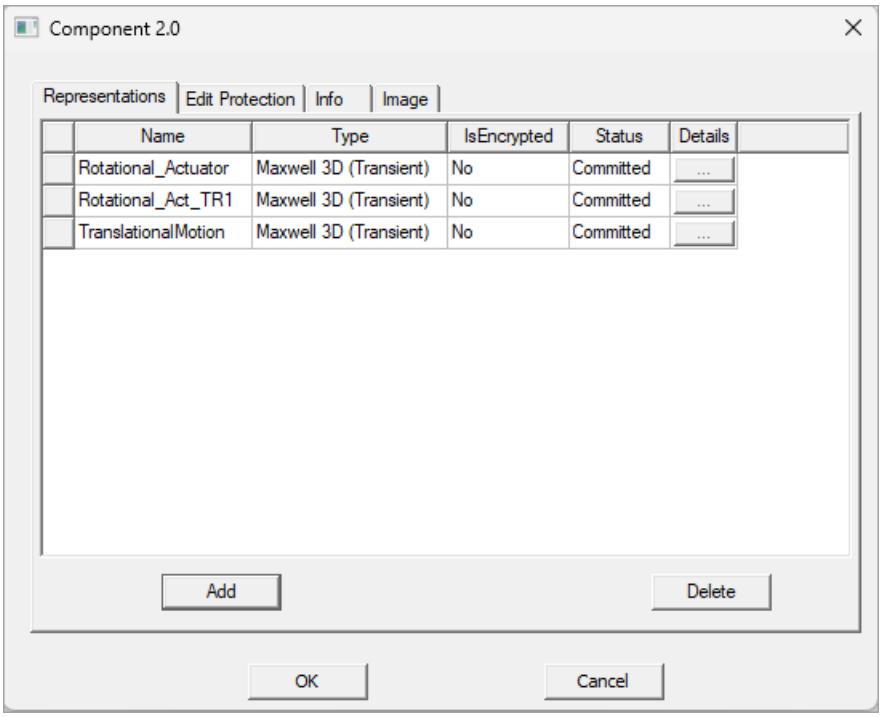
To edit a 2.0 component:

1. Right click on a design in the Project Manager tree, and select **Edit Component 2.0**.
2. In the file window that opens, navigate to the Component 2.0 file you would like to edit (it should have a .acmp2 file extension), and click **Open**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If the 2.0 component is encrypted, you will be prompted for a password. Enter the password to open the component for editing.

   The Component 2.0 window opens, which defaults to the **Representations** tab that displays the design representations that are in the component.



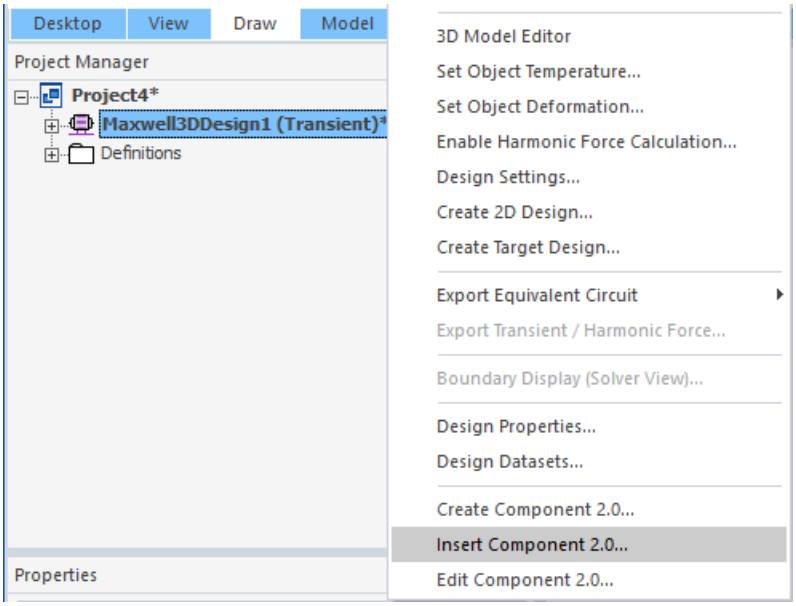
4. To add a representation to a component: select the design in the Project Manager tree, and then click **Add** in the Component 2.0 window. The Create Component 2.0 Representation window will open; see the Create 2.0 Component topic for more information.
5. To delete a representation from a component, select the design on the **Representations** tab, then click **Delete**. Note that if the representation is encrypted, you will be prompted for its password when you click **Delete**.

Inserting a Representation from a 2.0 Component

To insert a representation from a 2.0 component:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

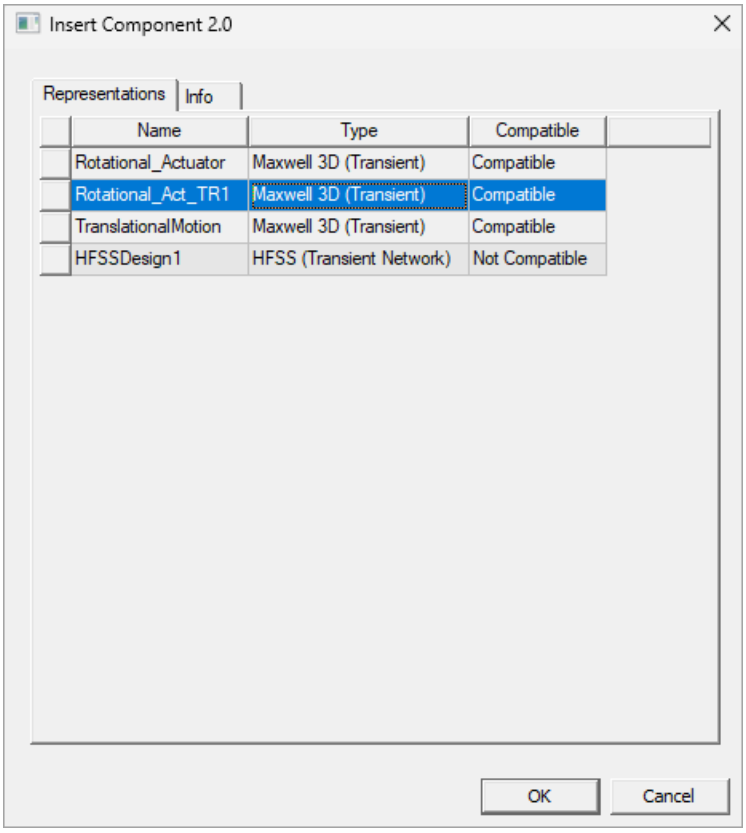
1. Right click on a design in the Project Manager tree, and select **Insert Component 2.0**.



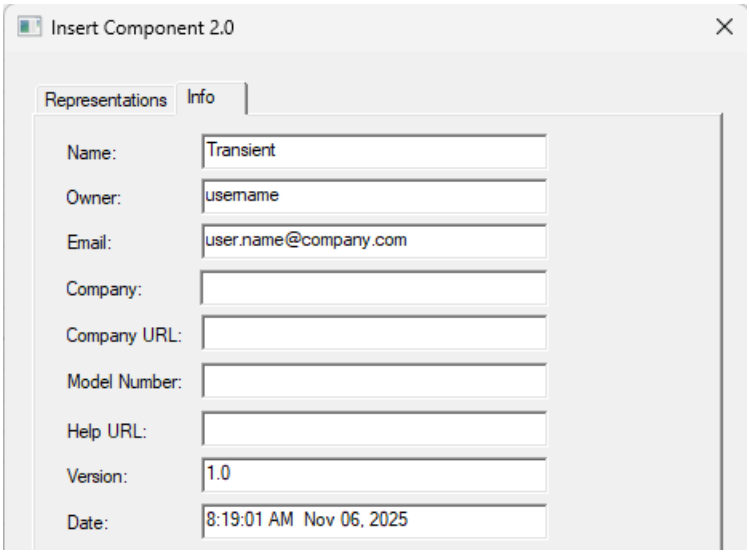
2. In the file navigation window that opens, find the ..acmp2 component file, and click **Open**.

   The Insert Component 2.0 window opens, as illustrated below. It opens to the **Representations** tab, which shows the representations contained in the 2.0 component. Each representation is listed by name and includes the type of design it is and whether or not it is compatible with the design you want to add the representation to. For example, if a Maxwell 3D transient design is active in the Project Manager tree, the Rotational_Actuator representation can be added to that transient design, but the HFSSDesign1 cannot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



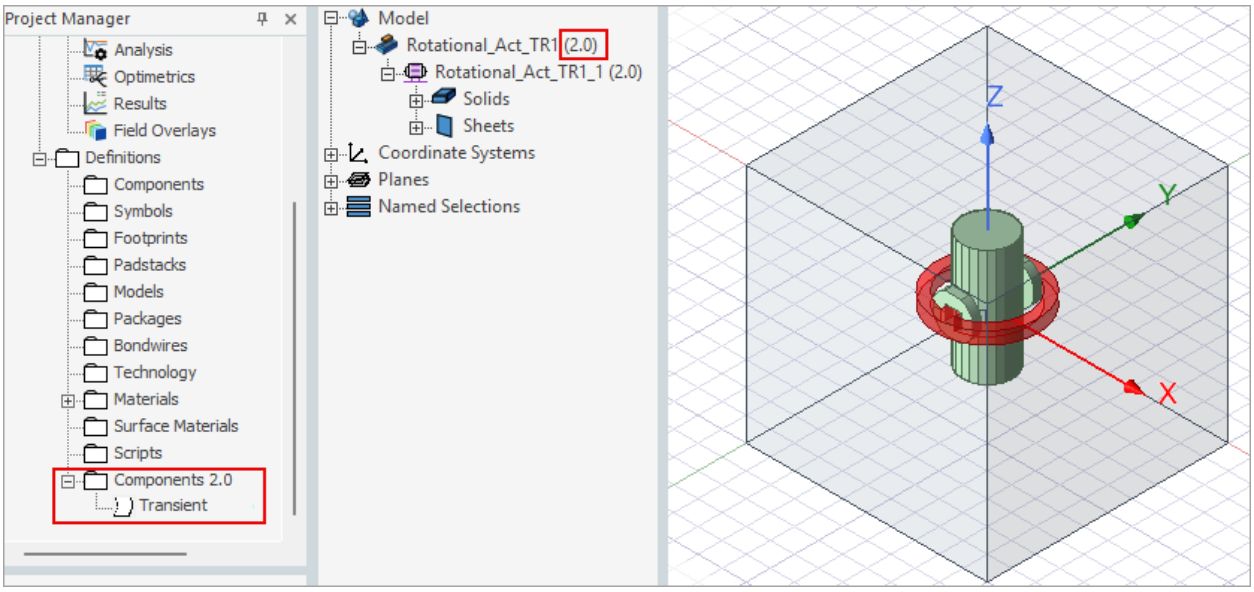
The **Info** tab lists the details for the 2.0 component, not the representation selected in the **Representations** tab:



3. Select the representation you would like to add to the active design, and click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The representation has been added in the Modeler window. In the history tree, it is listed with the 2.0 label, and the Definitions folder in the Project Manager tree lists the component the representation was taken from.



Once an instance from a 2.0 component has been inserted into a project design, other instances from that same component can be inserted in the project by right clicking on the **3D Components** label in the Project Manager tree or the model name in the history tree and selecting **Insert Instance**, which opens the Insert Instance window.

If a representation has not been encrypted, it can also be copied and pasted. If it has been encrypted, it can be moved, and multiple instances can be inserted, but it cannot be copied and pasted.

Layout Components [Beta] – Topic Index by Product

Support for Layout components is a beta feature available in HFSS, Maxwell, IcepakFEA, and Q3D Extractor 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 design 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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
- **IcepakFEA:**
  - Layout Components in IcepakFEA (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
- **Maxwell:**
  - 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
- **Q3D Extractor:**
  - 3D Layout Components in Q3D Extractor (Introduction)
  - Importing a Layout Component
  - Inserting an Existing Layout Component

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Net Visualization of Layout Components
- Layout Component Properties
- Editing Placement of a Layout Component

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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   > **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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

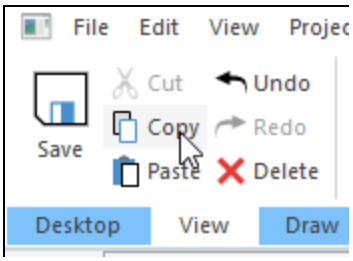
   Any objects you draw after this point will be assigned the default transparency you specified. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

   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.

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:



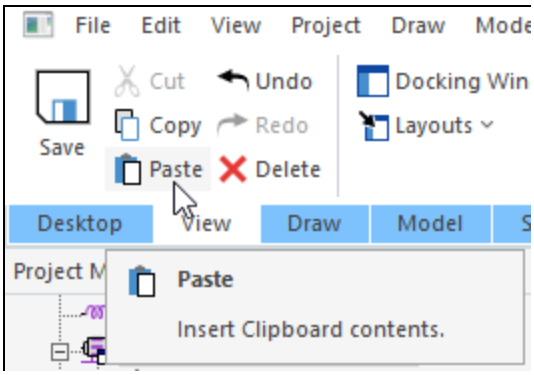
   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Modeler** > **Import From Clipboard**.

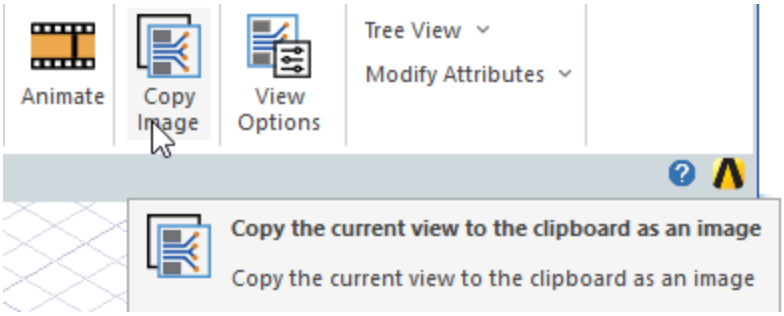
   The geometry is pasted from the Clipboard with the IDs preserved. Additionally, the source design's coordinate systems, materials, appearances, and other attributes are preserved and duplicated in the target design.

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.

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.

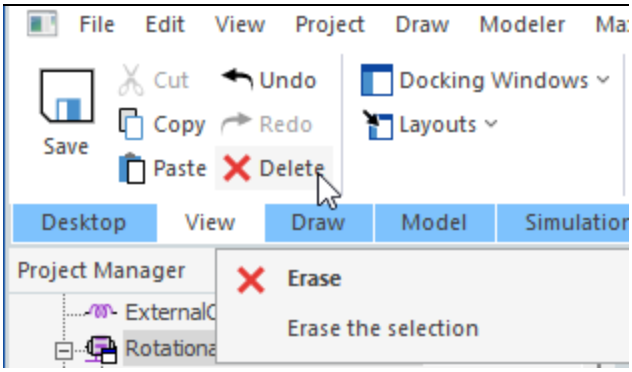
   An image of the report is copied to the Clipboard.

2. Open the application into which you want to paste an image of the report and follow that application's image pasting procedure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

- 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

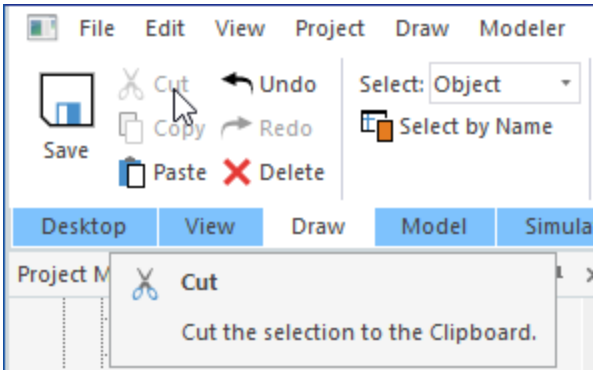
You can delete the last geometry operation performed on an object. For certain situations, this functionality can offer an advantage when compared with the *Undo* command. For example, if you performed several valid operations (such as assigning boundaries, excitations, or mesh operations) after operating 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.

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.

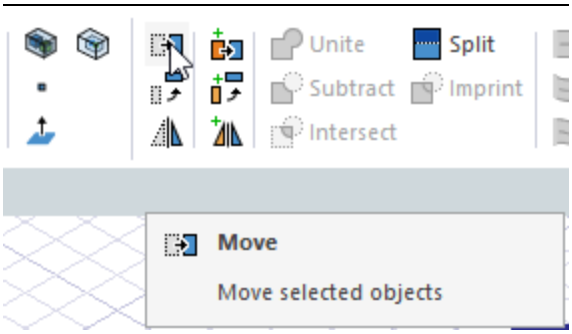
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

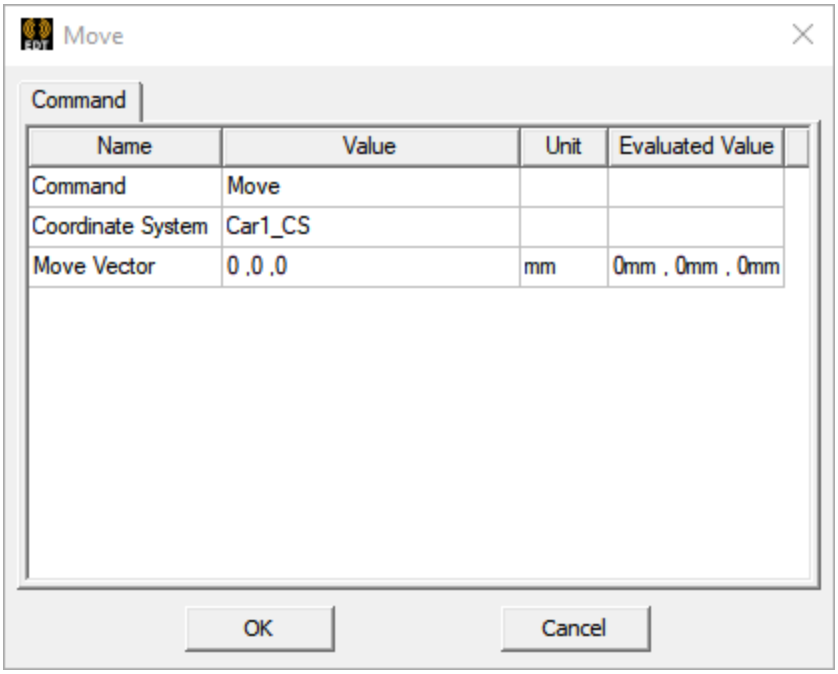
   All selected objects move the distance determined by the offset between the anchor point and the target point.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



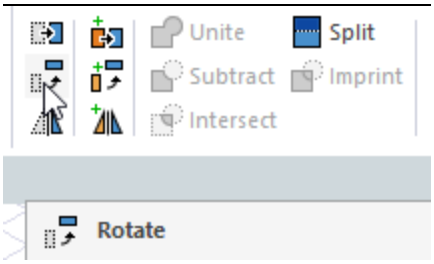
3. Enter the coordinates of the location to which the object should move and click **OK**.

Rotating Objects

Rotate objects about the x-, y-, or z-axis using the **Edit** > **Arrange** > **Rotate** command.

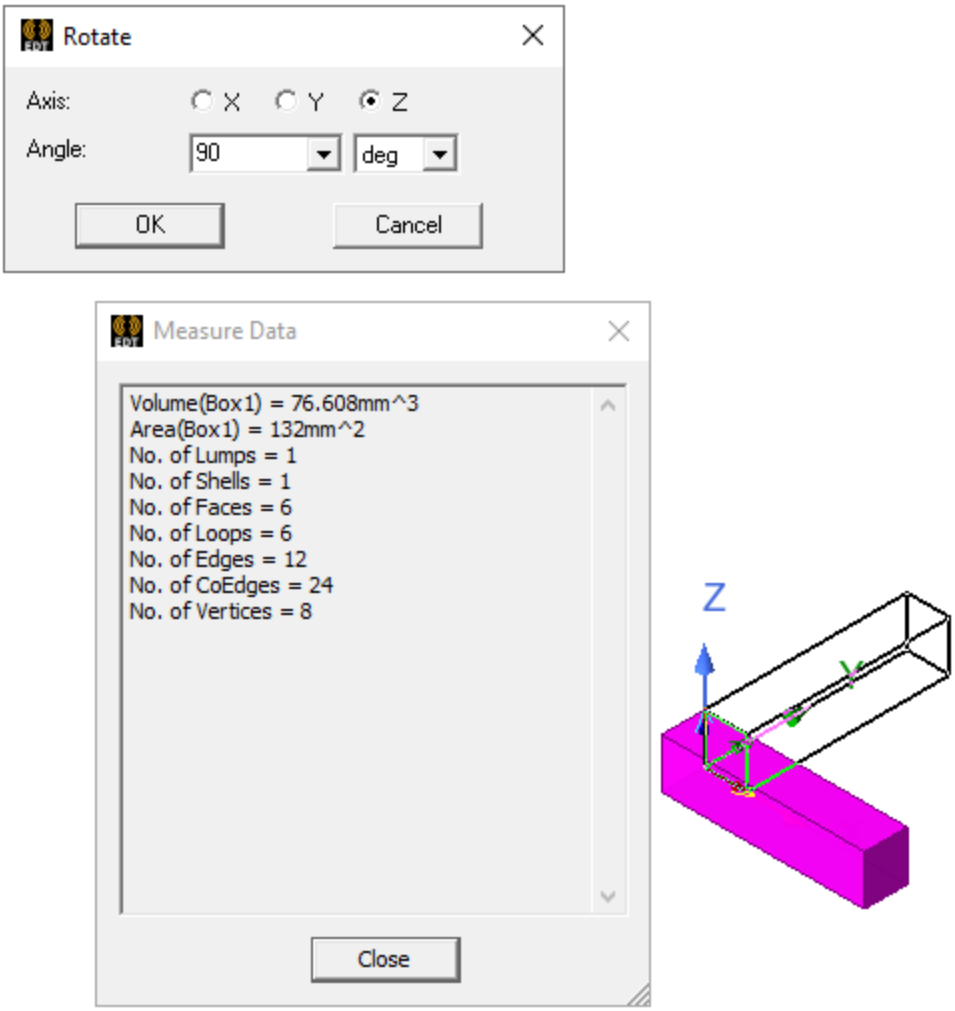
To rotate objects about an axis:

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:



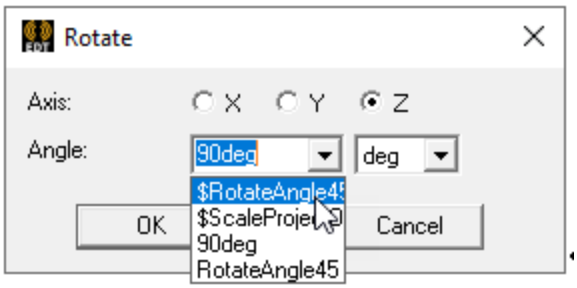
The **Rotate** dialog box appears along with the **Measure Data** window. A visualization of the proposed change appears in the Modeler window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

   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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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

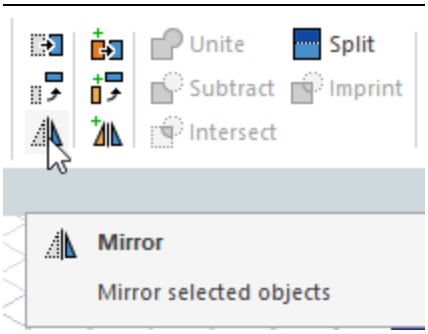
Mirror an object about a plane using the **Edit** > **Arrange** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to move an object and change its orientation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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:

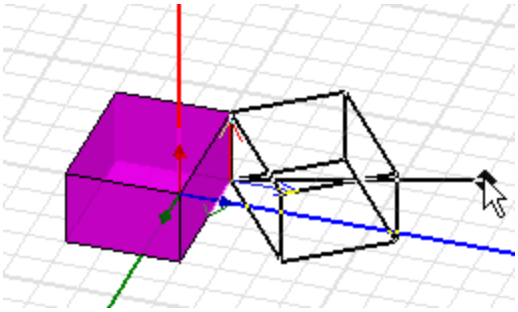
1. Select the object or objects that you want to mirror. You can select multiple objects.
2. From the menu bar, click **Edit** > **Arrange** > **Mirror** or, on the **Draw** ribbon tab, click the **Mirror** icon:



3. Select a point on the plane around which you want to mirror the object.

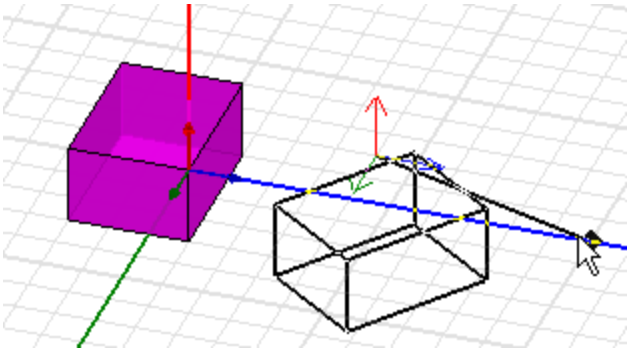
   You can do this by clicking a point, or typing coordinates in the X, Y, and Z boxes in the status bar.

   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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

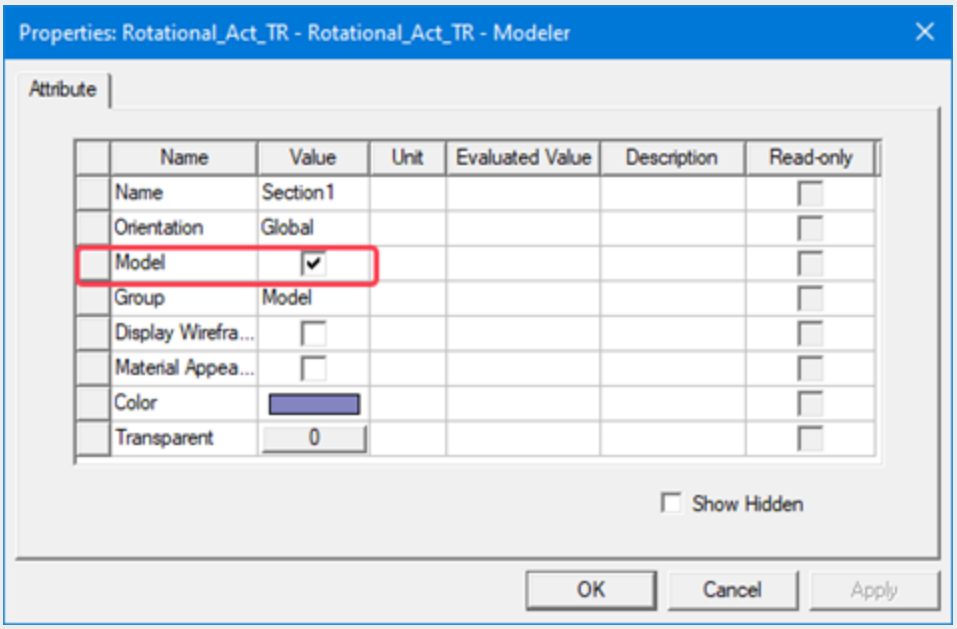
Duplicating Objects

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.

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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

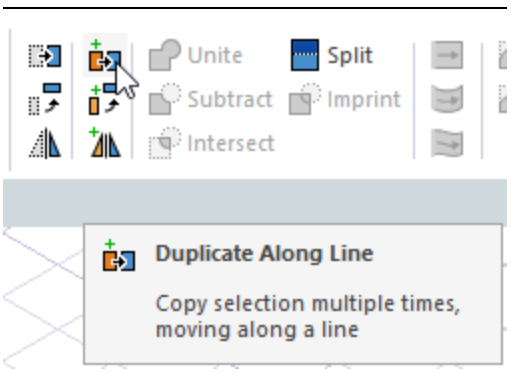
To copy objects to another design, use the **Edit** > **Copy** and **Edit** > **Paste** commands.

> **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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

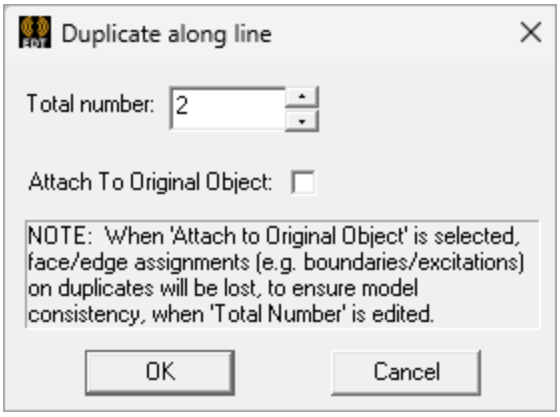
3. Specify the vector along which the object will be duplicated. The *Measure Data* window appears to help you choose points:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X, Y,** and **Z** boxes.

      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:



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. Click **OK**.

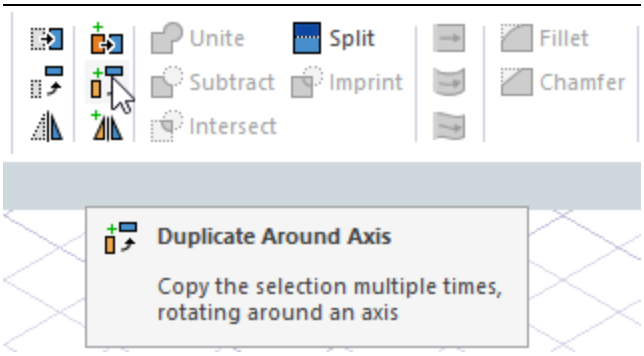
   The duplicates are placed along the vector you specified.

Duplicating Objects Around an Axis

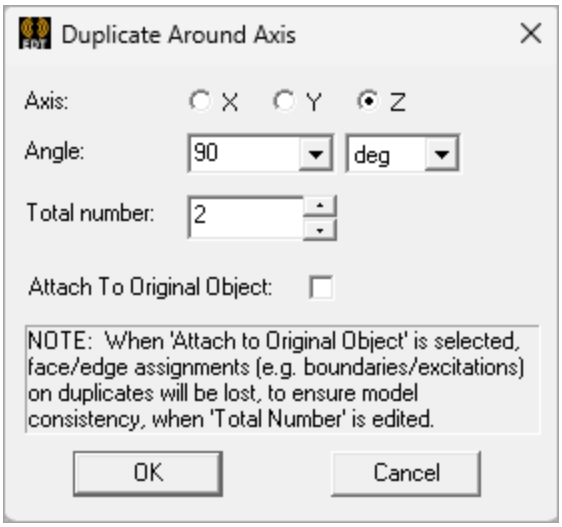
To duplicate an object around the x-, y-, or z-axis, use the **Edit** > **Duplicate** > **Around Axis** command.

Ansys Electromagnetics Suite 2026 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 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:



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.

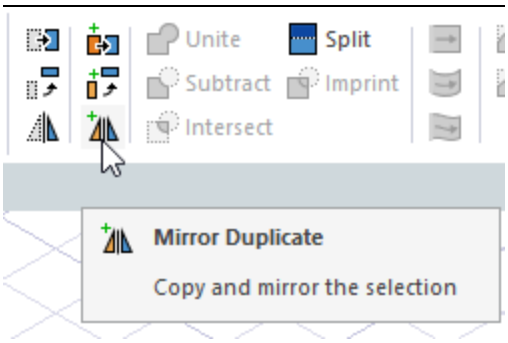
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:



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.

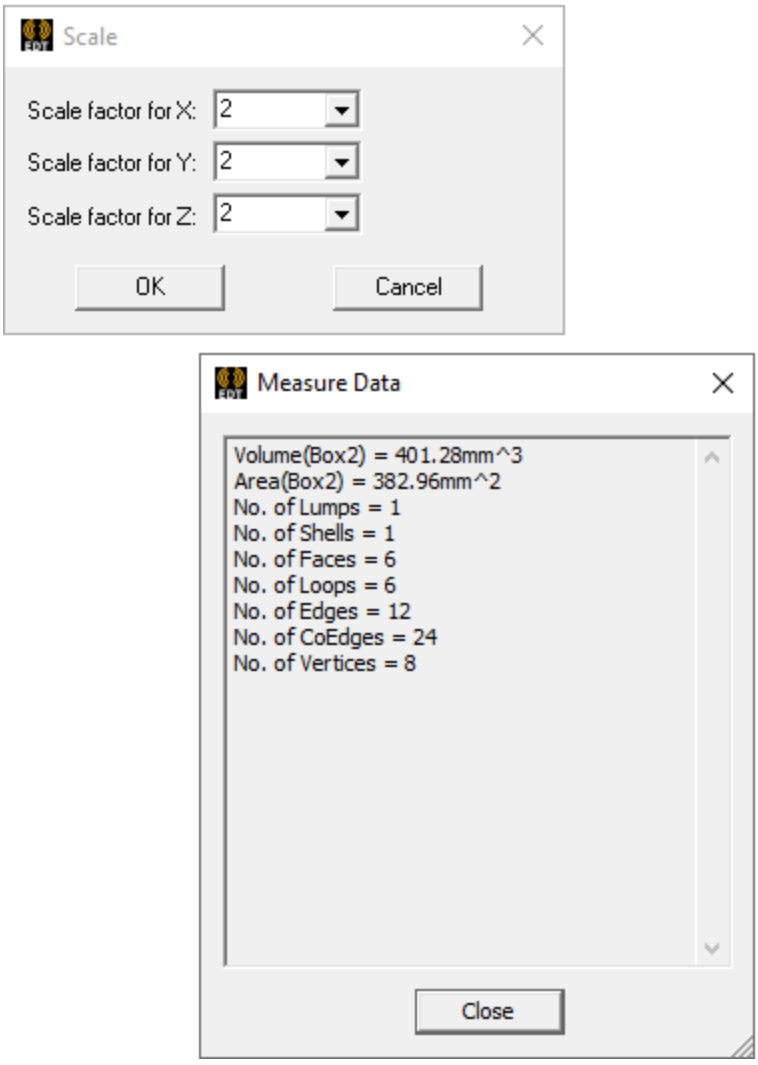
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
3. From the menu bar, click **Edit** > **Scale**.

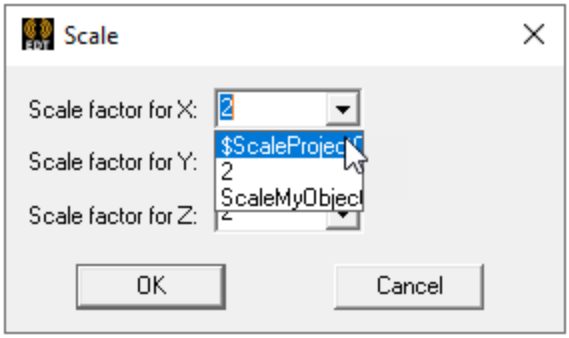
   The **Scale** dialog box appears with the **Measure Data** window.



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 2026 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.

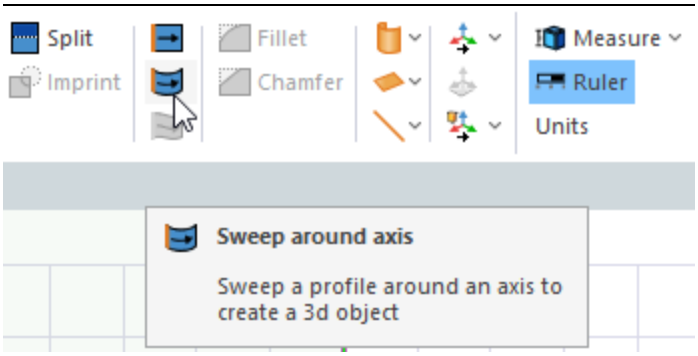
Before using this command, keep the following guidelines in mind:

- 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.

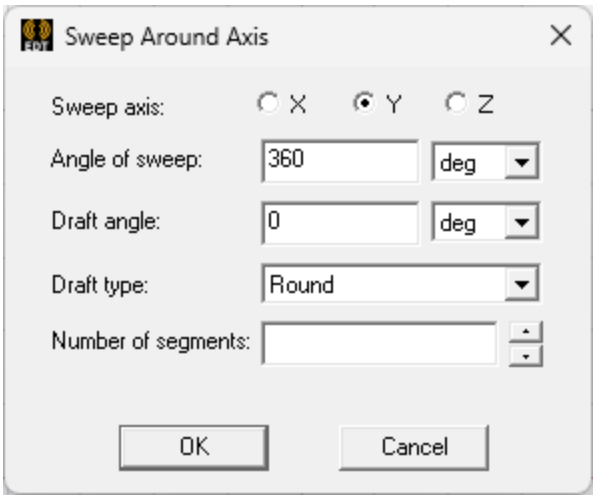
To sweep an object around an axis:

Ansys Electromagnetics Suite 2026 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with a *decreasing* radius (that is, the distance from the generated surface to the sweep 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.

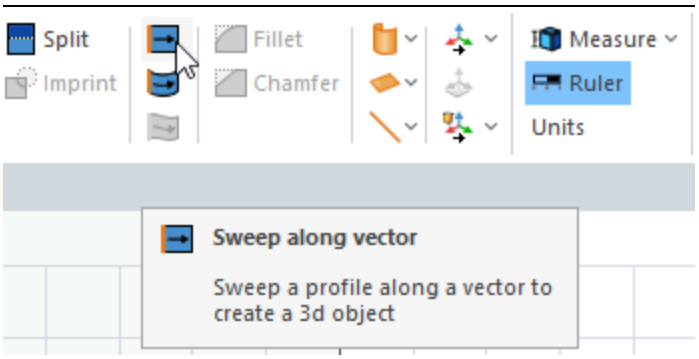
8. Click **OK**.

Sweeping Along a Vector

Sweep a 1D or 2D object along a vector using the **Draw** > **Sweep** > **Along Vector** command.

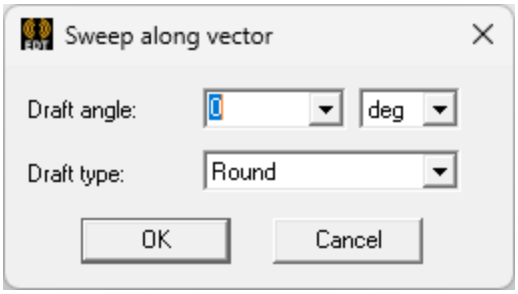
1. Select the object you want to sweep.
2. Click **Draw** > **Sweep** > **Along Vector** or in the **Draw** ribbon tab, click **Sweep along vector**:

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Extended**
- **Round**
- **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:

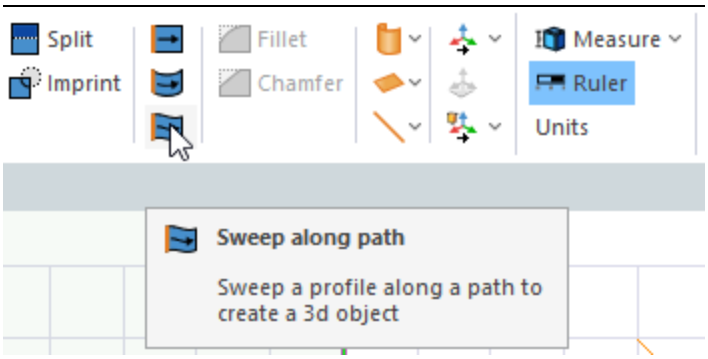
- When sweeping along a path that is not closed (i.e., the start and end point are not the same), ensure that the start point of the path is on the profile being swept.
- When sweeping a profile around a closed path, ensure that the path intersects the profile.
- Parasolid sweep operator preserves the 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.

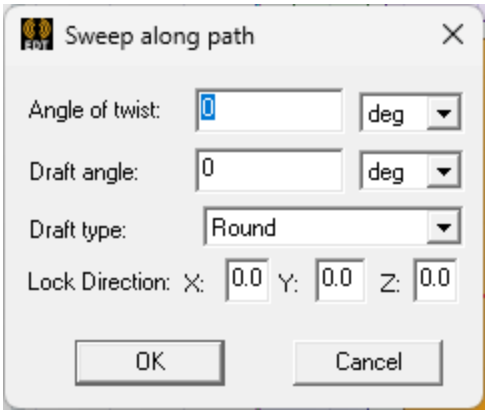
To sweep an object along a path:

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 2026 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**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

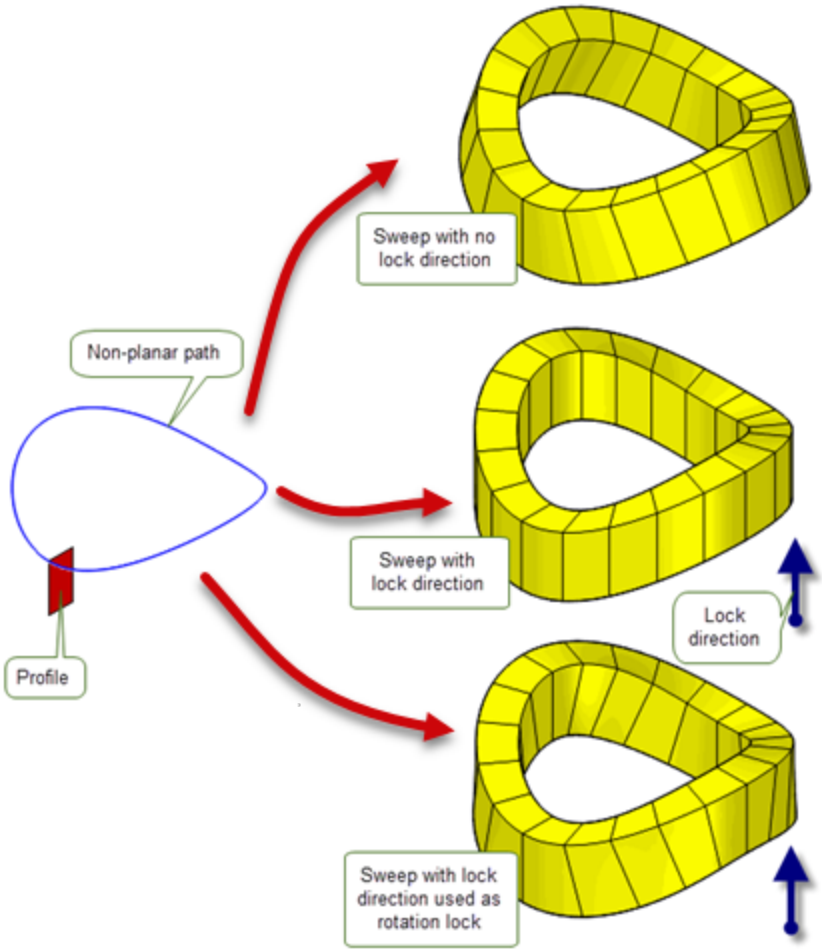
- **Natural**

For a complete description of the behavior of each of these three options, see Draft Types.

7. Define a **Lock Direction** to lock the orientation of the side faces so that they are normal to the selected direction.

   Without a lock direction, the profile's orientation will be relative to the normal plane of the path. When the path is not planar, the profile's orientation will change along the path during the sweep operation.

   With a lock direction set, the profile orientation will be relative to the lock direction (instead of the normal plane of the path). The sweep direction will be projected into the plane perpendicular to the lock direction, and the profiles of the sweep will maintain a fixed relationship to lock direction.

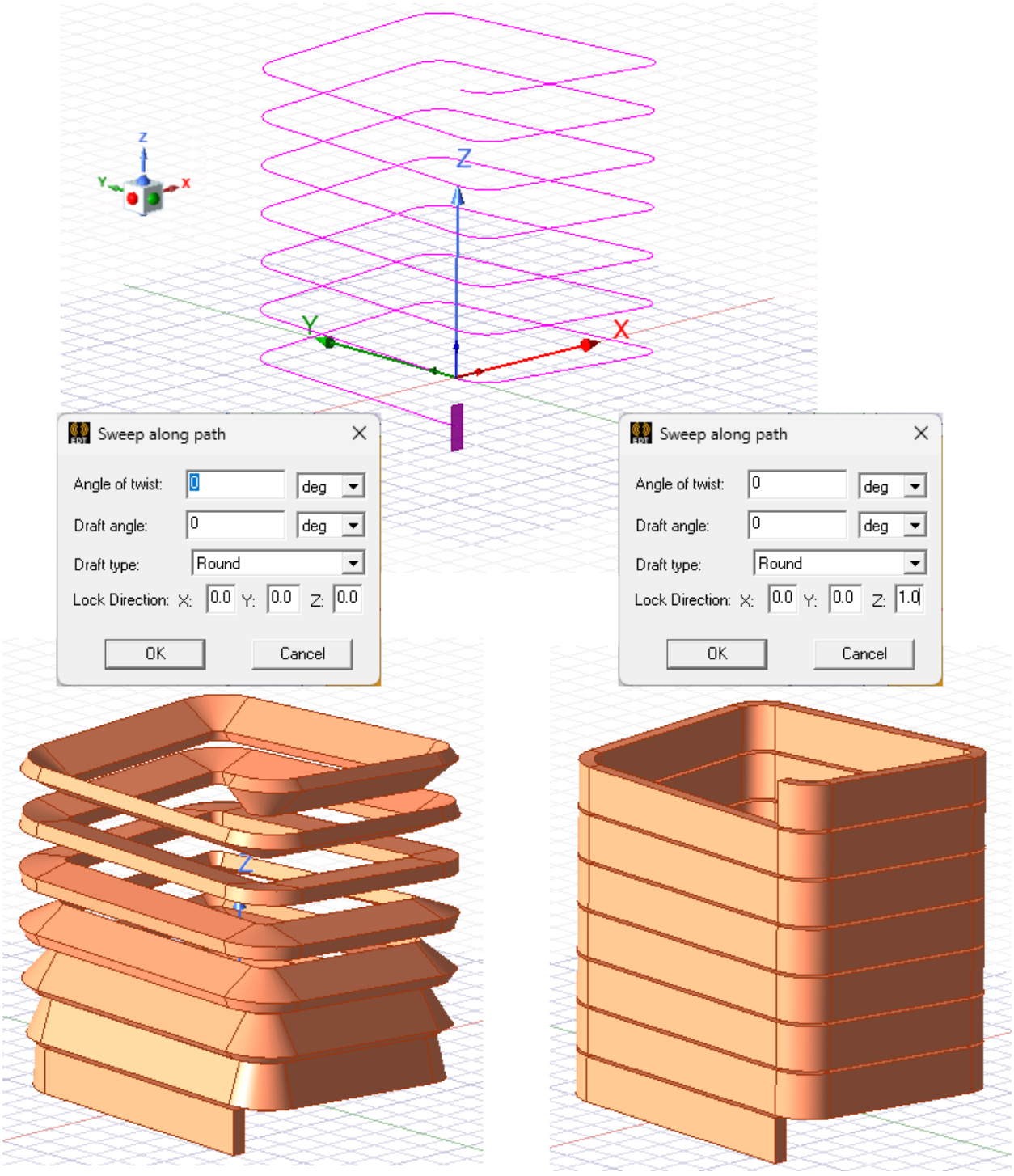


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If a Lock Direction is used with a Twist Angle, the axis of rotation of the twist angle follows the tangent of the path, but remains within the plane normal to the lock direction.

Below is a coil example of Sweeping Along a Path without and with a Lock Direction:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

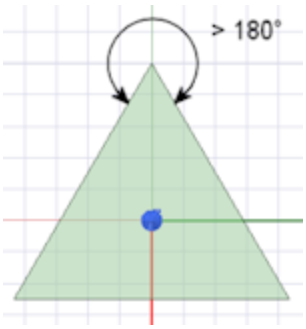
   The *Sweep Faces Along Normal* dialog box appears.

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

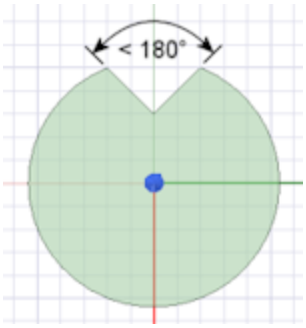
In many cases, the **Draft type** setting may have little to no effect on the resultant geometry of a swept object unless a *positive, non-zero draft angle* is specified. Consider a *sharp corner* feature. For the purpose of this discussion let's define a sharp corner as any protruding feature where the *external* angle between the adjacent faces is greater than 180-degrees:



Ansys Electromagnetics Suite 2026 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:

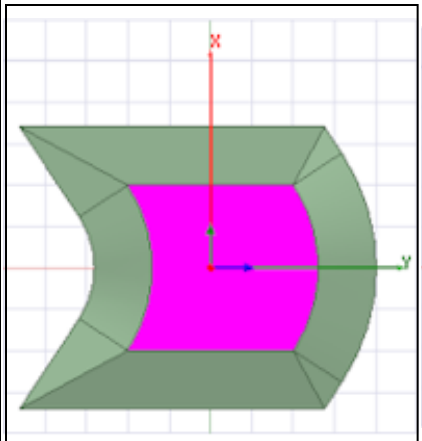
- **Natural:** This draft type generally maintains the shape of the original object as it is swept along the specified direction. That is, all sharp corners remain sharp, producing sharp edges in the sweep direction. There are exceptions to this general behavior. Rounding of certain corners may occur 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 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

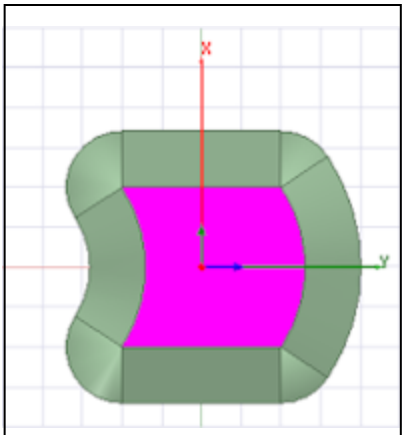
The images below compare the results of sweeping an object using each of the available **Draft type** options. The first six images are of an object swept with a positive draft angle defined. The final three images are of a different object swept with a negative draft angle defined. For all examples, the original 2D object is highlighted in magenta.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

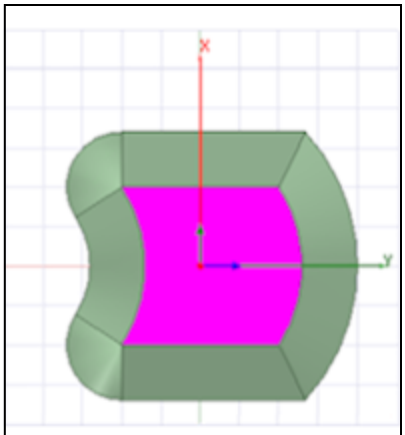
1. **Sweep Along Vector** (Positive Draft Angle):



1a: Extended

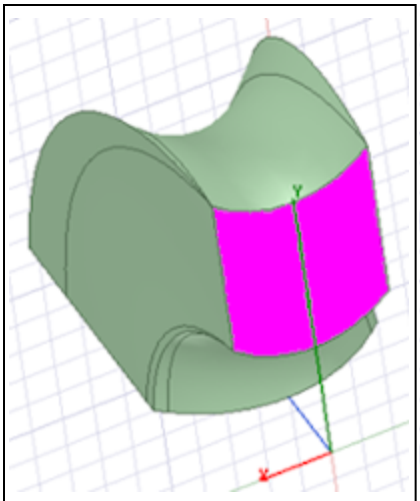


1b: Round

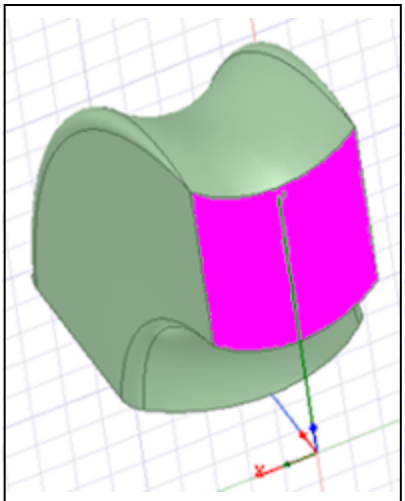


1c: Natural

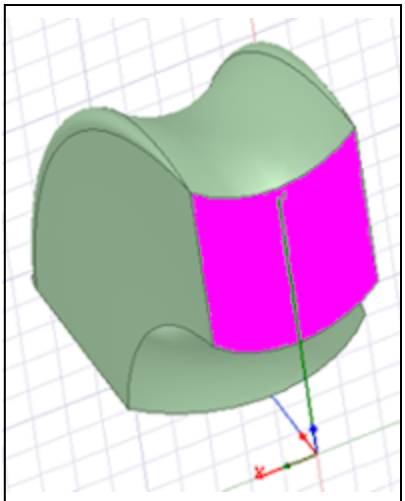
2. **Sweep About Axis** (Positive Draft Angle):



2a: Extended



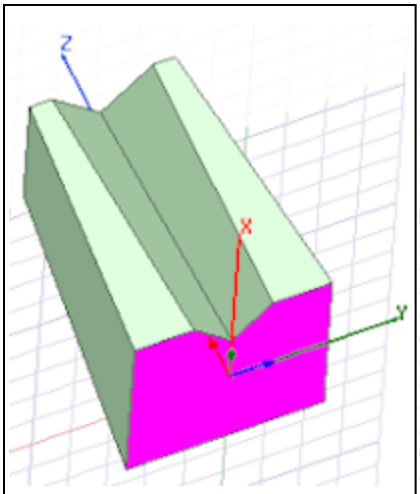
2b: Round



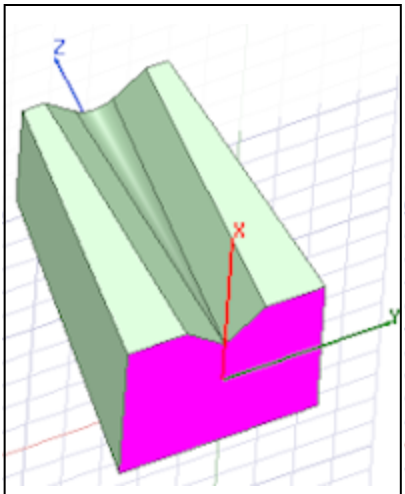
2c: Natural

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

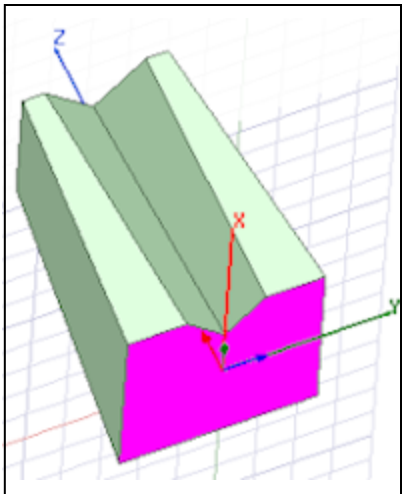
3. **Sweep Along Vector** (Negative Draft Angle):



3a: Extended



3b: Round



3c: Natural

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).

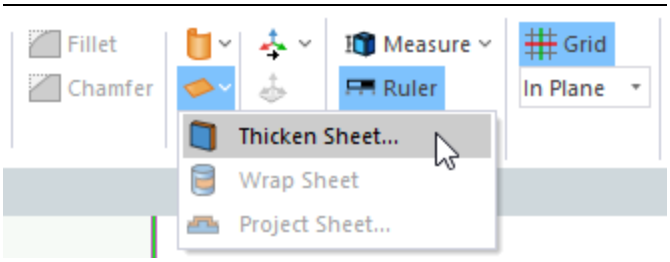
  In example **3c** (negative draft angle), no face is split, and no edge is rounded.

Thicken Sheet

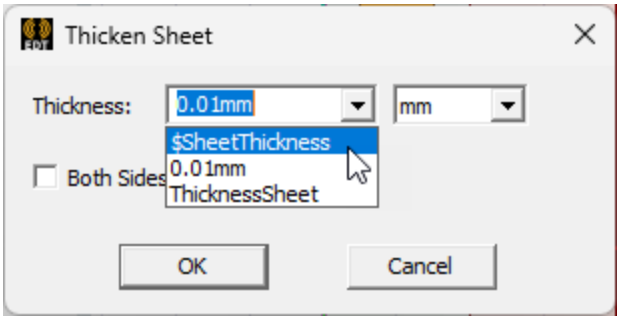
To thicken one or more sheet objects to make 3D objects:

Ansys Electromagnetics Suite 2026 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 drop-down menu.
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

You can use **Modeler** > **Surface** > **Wrap Sheet** function 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.

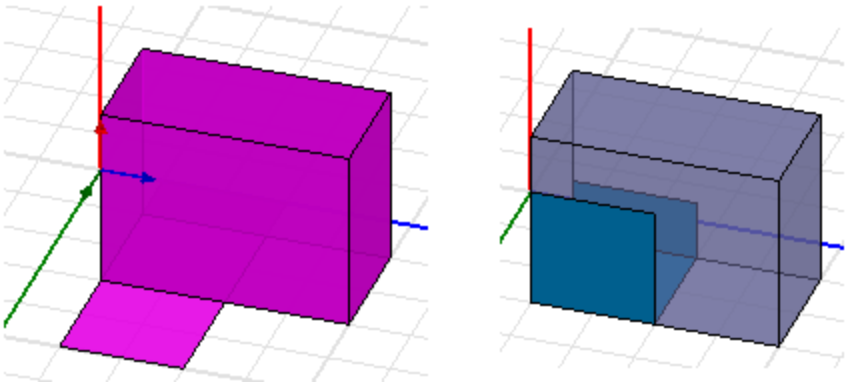
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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:



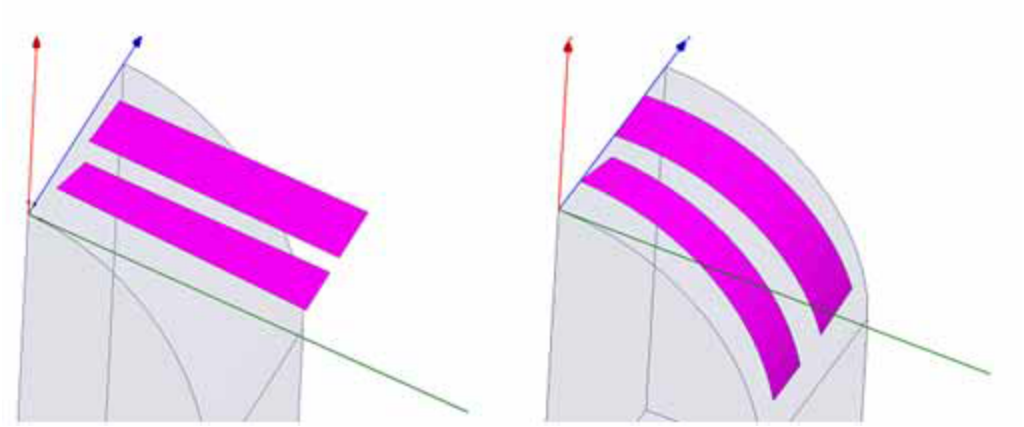
If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

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.

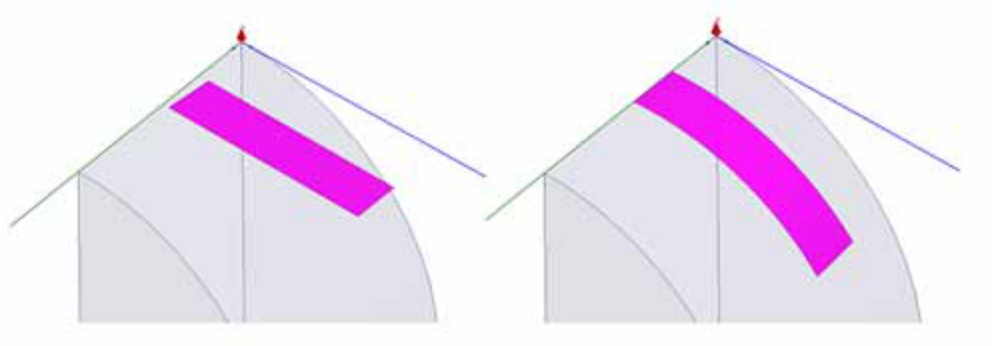
Examples:

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.

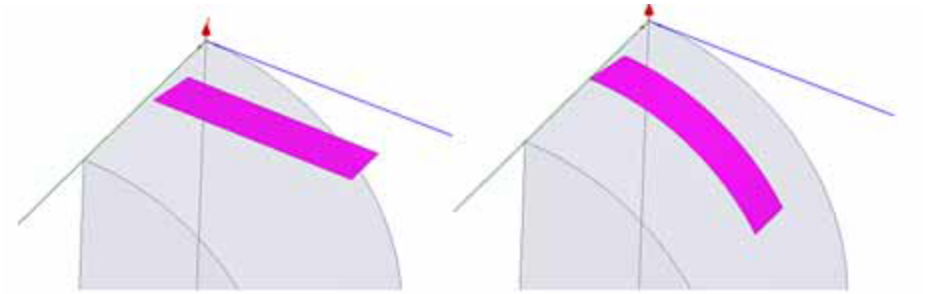
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move the edge to the edge before wrapping.

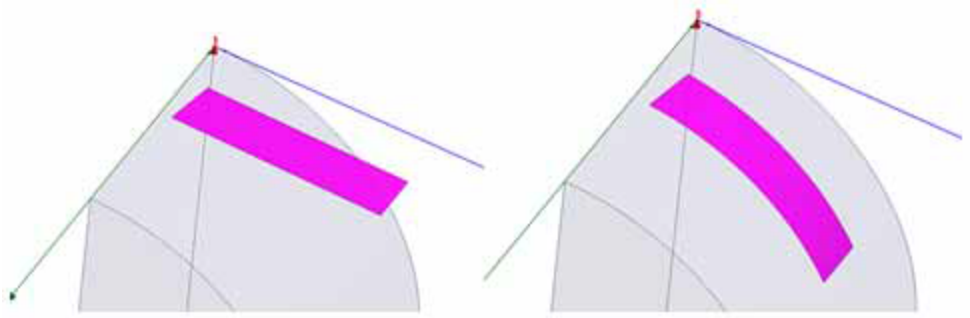


The following figure shows the situation when you move a vertex to an edge before wrapping.

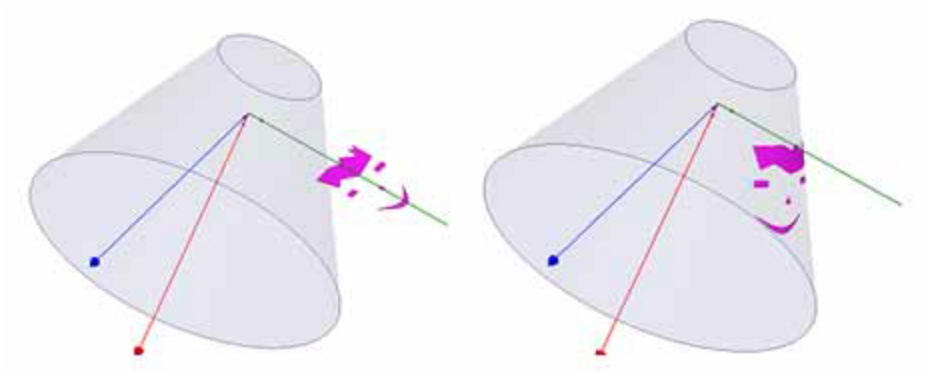


The following figure shows the situation where you move a vertex to a face before wrapping.

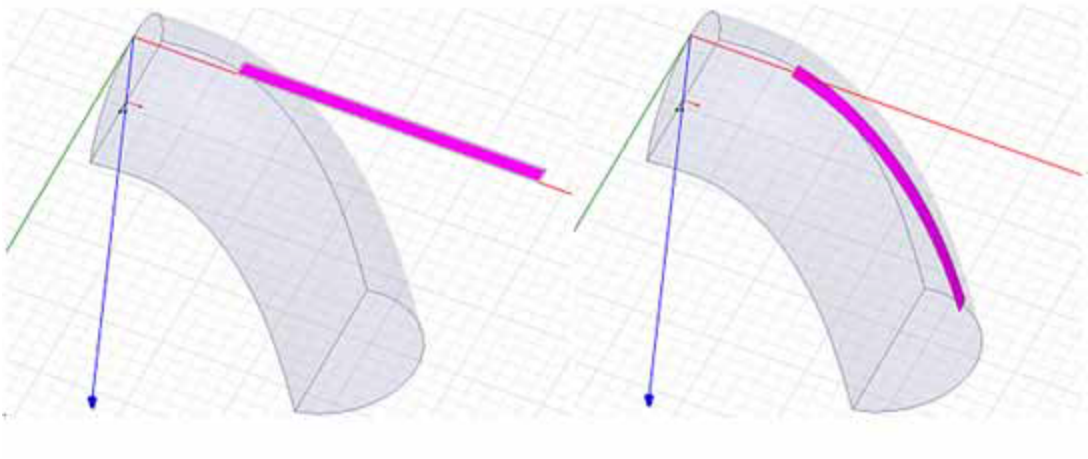
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

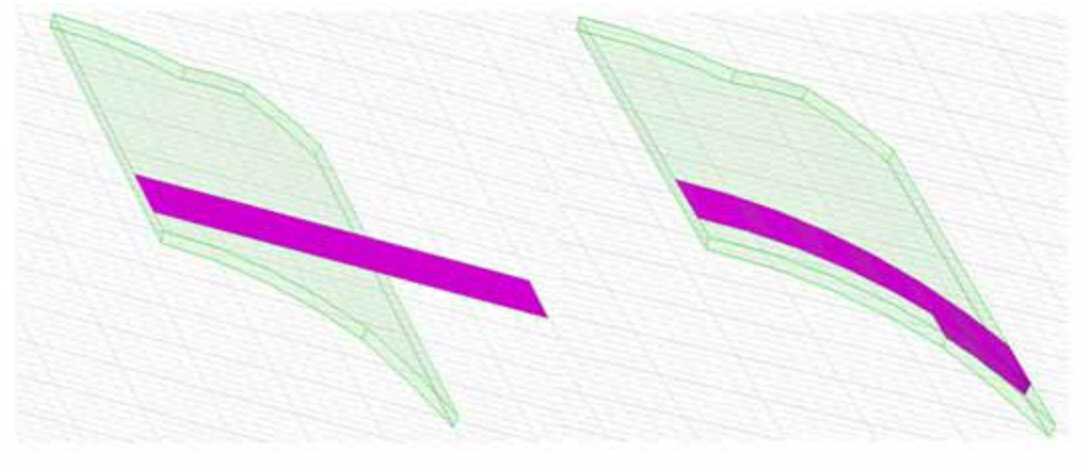


The next figure illustrates wrapping to a non-developable surface. The area of the sheet changed from 0.7000 mm^2 before wrapping to 0.6986 mm^2 after wrapping.



Ansys Electromagnetics Suite 2026 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:

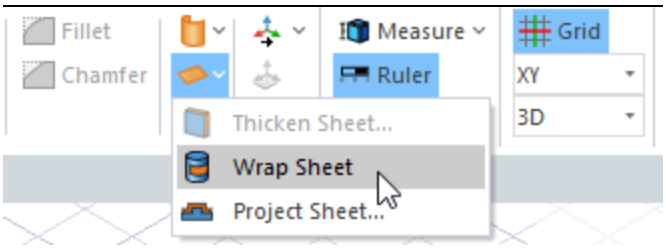
  WrapSheet: Wrapping sheet contacted pole of target surface. Recommend moving sheet and splitting out pole from target surface.

- 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 2026 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 2026 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.
3. 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.
3. 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

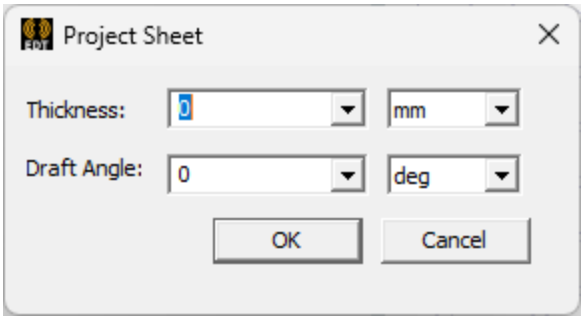
To connect objects:

1. Select the objects you want to connect. Select 3D objects as objects, not as faces.
2. Click **Modeler** > **Surface** > **Connect**.

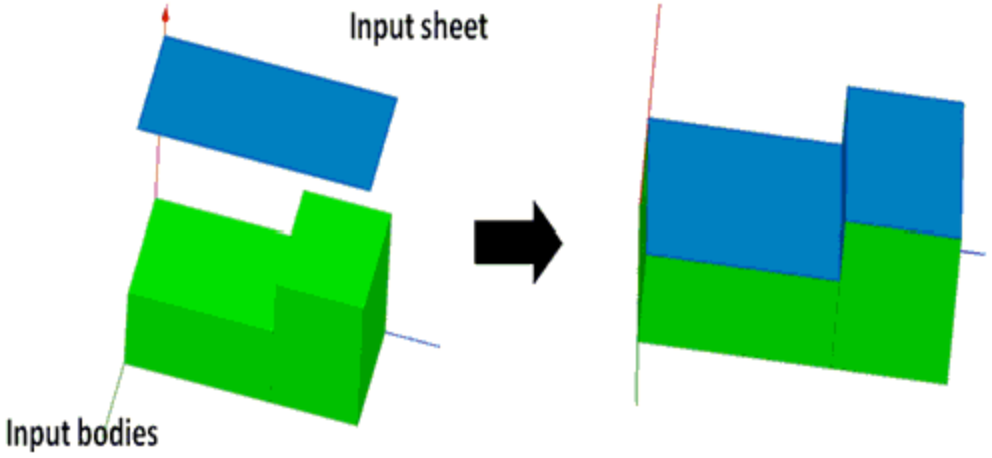
   A new object is created that connects the objects you selected. The first object you selected was modified to create the new object and all subsequently selected objects were deleted.

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.

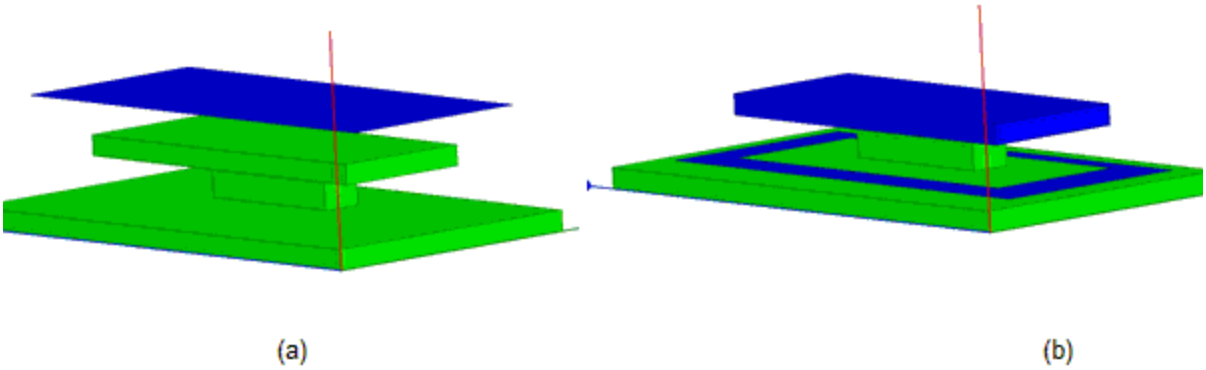


To use the **Project Sheet** command you must select one planar sheet and at least one solid body. If you select multiple solids, the sheet is project on all bodies as if they have been united. After the commend executes, a new sheet that lies on the surface of the selected bodies is created. The solid bodies are not changed.

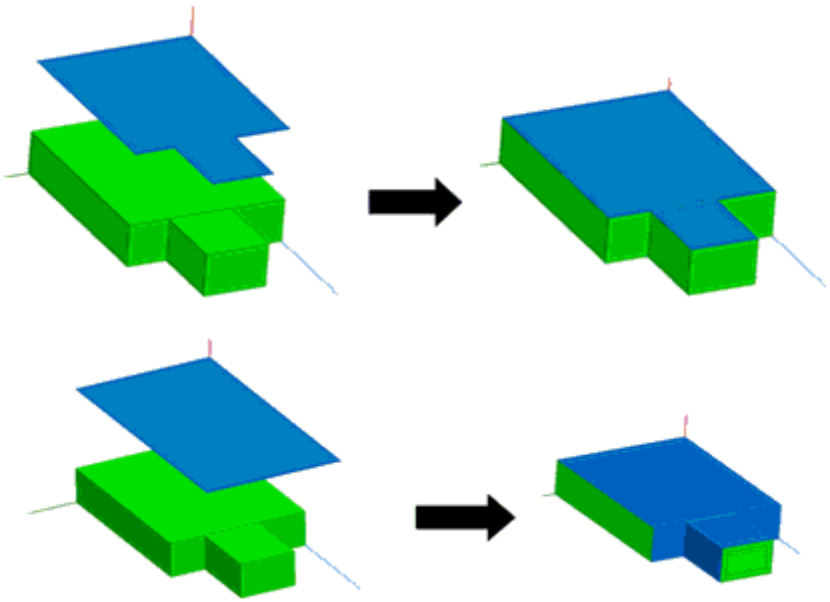


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faces that are hidden from the sheet being projected are not covered, due to the projected sheet.



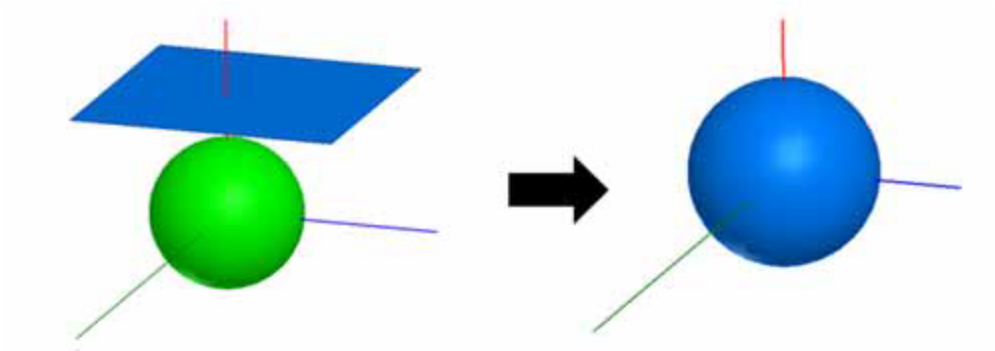
Only vertical faces (or faces parallel to the projection direction) that lie completely within the projection of the sheet will be covered by the projected sheet. Vertical faces that lie on the boundary of the projection are not covered.



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.

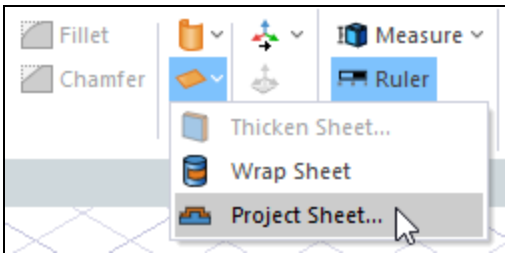
If a surface like a cylinder or sphere lies completely within the sheet projection, the entire surface will be covered, though there may be a warning that “Output may be incorrect because a self-obstructing face was found.”

Ansys Electromagnetics Suite 2026 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**:

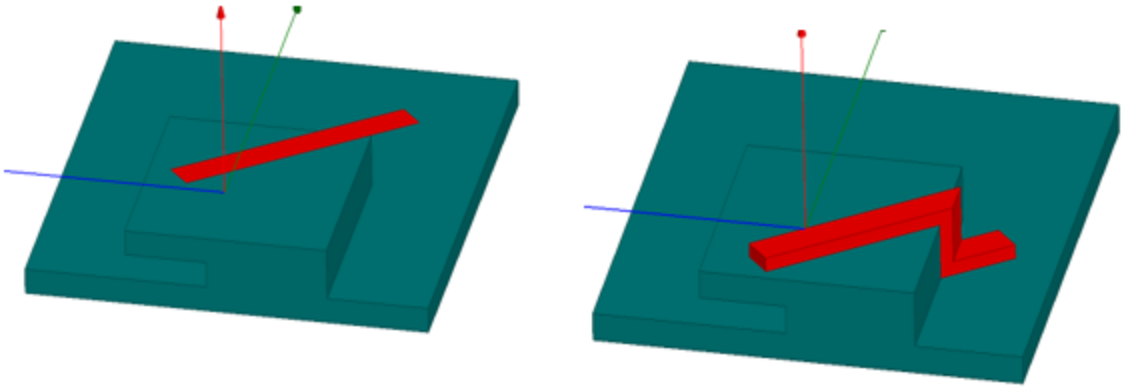


   For modeling conformal deposits, you can also specify a thickness and optional draft angle.

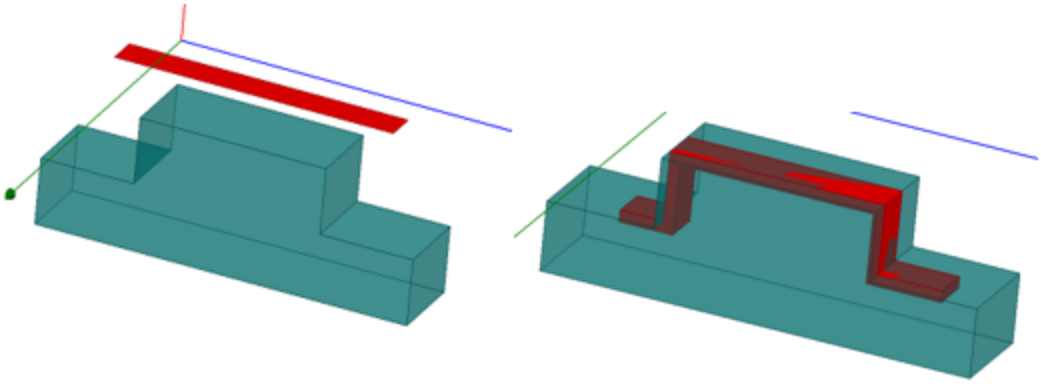
After you perform **Project Sheet**, the History Tree shows the **Project Sheet** command and the create command for the imprinted object. If you select the **Project Sheet** command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the Create <object> 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

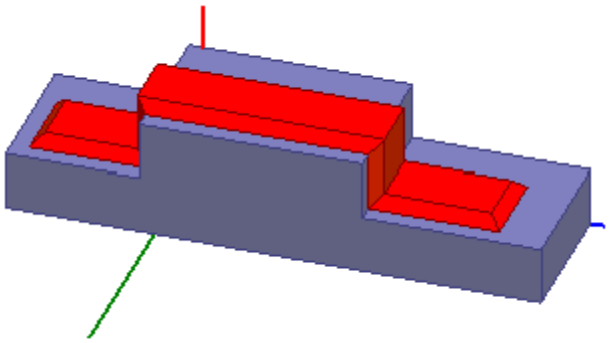


The following figure shows the effect of a negative value for thickness.



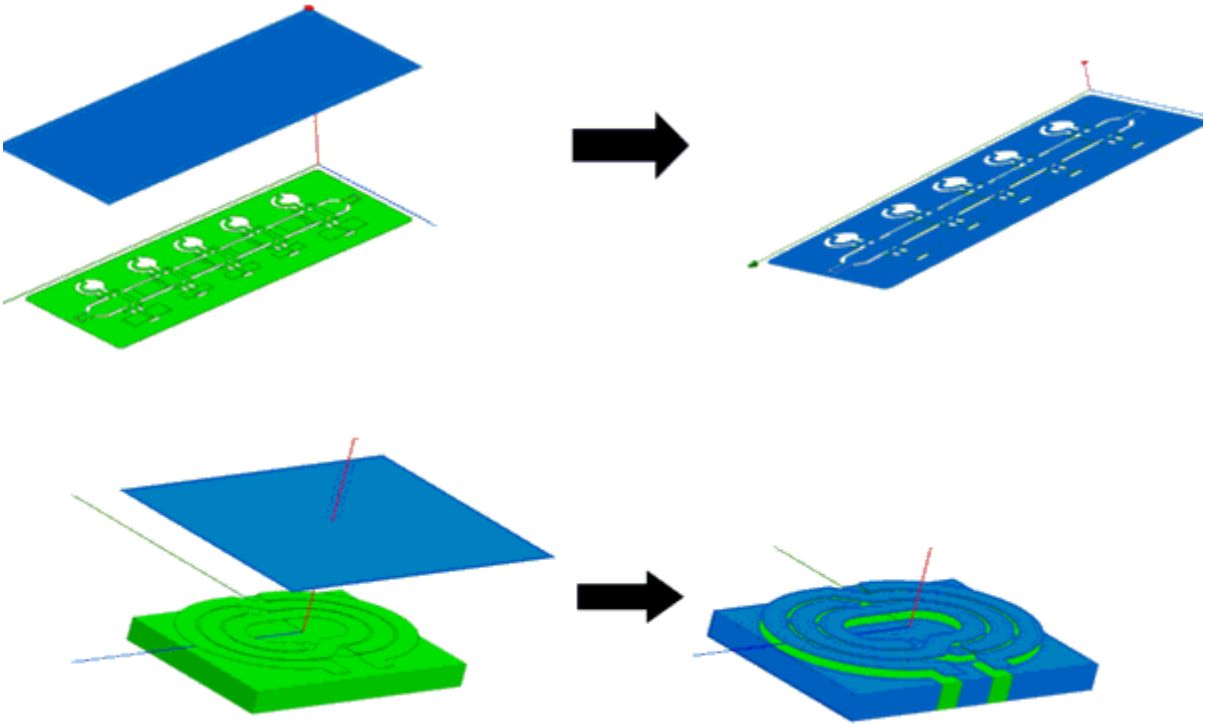
When a negative value is given the thickness will be limited by the target solid body. That is, the thickness can not be beyond the original solid body which was chosen. If you use a negative thickness it is also possible that not all overhangs will be eliminated.

The following figure shows the effect of a 45 degree draft angle.



Ansys Electromagnetics Suite 2026 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.

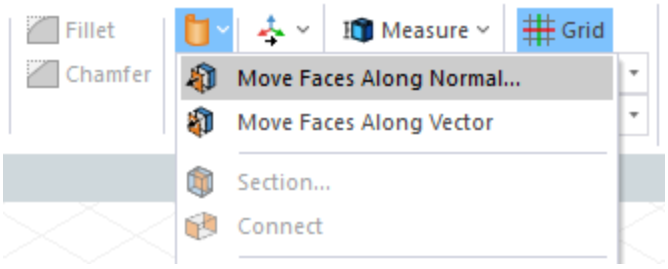
This command is useful for extruding faces, resizing holes, and removing rounded corners, as shown below.

To move an object face in a normal direction:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

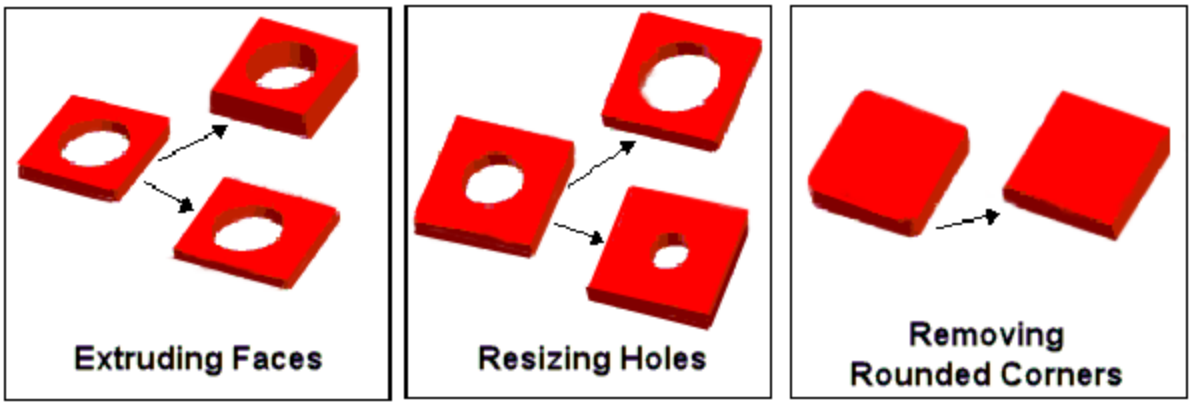
1. Right-click in the Modeler window and click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

   If you have created a suitable face selection, right-click the selection and click **Select Assignment** from the shortcut menu. You can operate on faces in the selection.
3. Click **Modeler** > **Surface** > **Move Faces** > **Along Normal**, or on the **Draw** ribbon tab, click **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

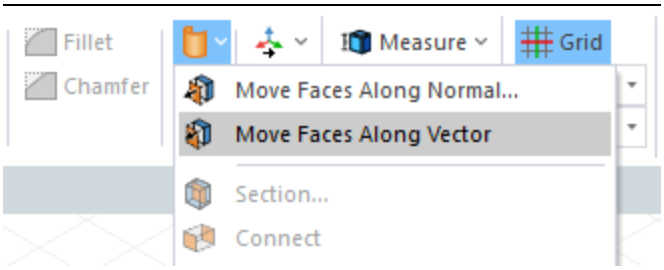
This command is useful for relocating holes in an object, as shown below.

To move an object face along a vector:

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 selection, 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.

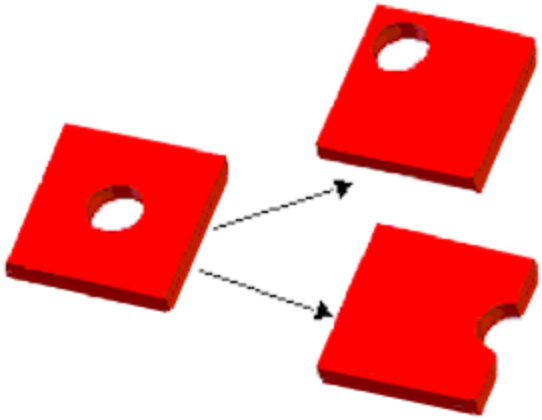
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the vector.

   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 2026 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.

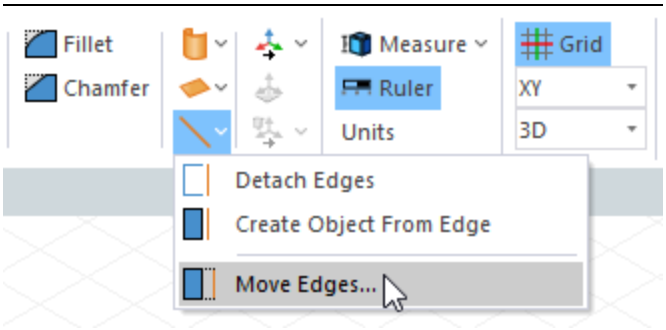
Moving Edges Along Normal

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

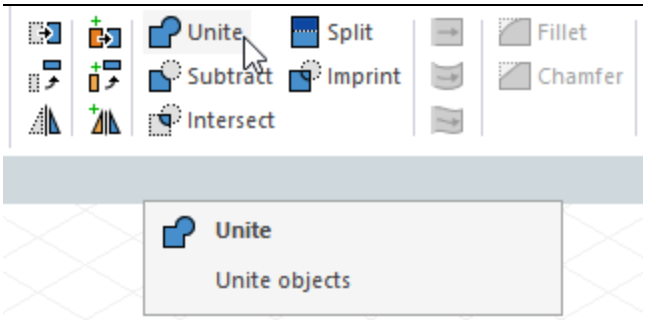
4. Type the distance you want to move the selected edge.
5. Click **OK**.

   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:



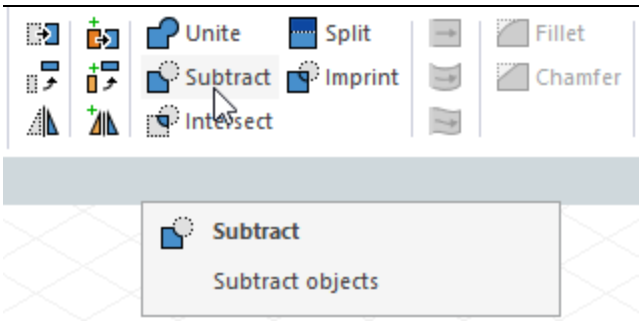
   The objects are united.

> **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Subtract* dialog box appears.

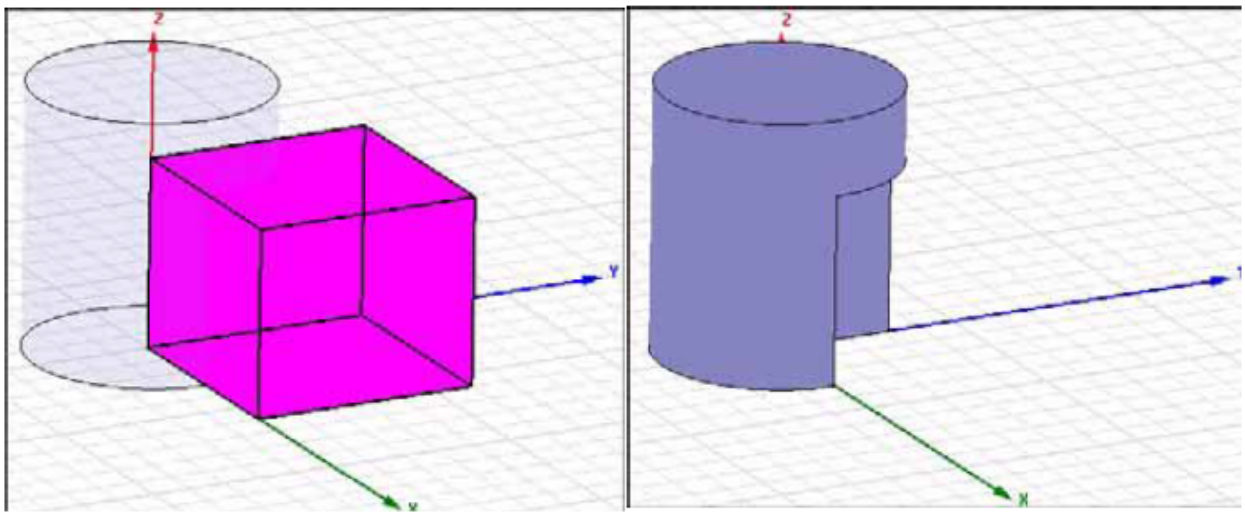
Objects listed in the *Tool Parts* list will be subtracted from the object or objects listed in the *Blank Parts* list.

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 2026 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:

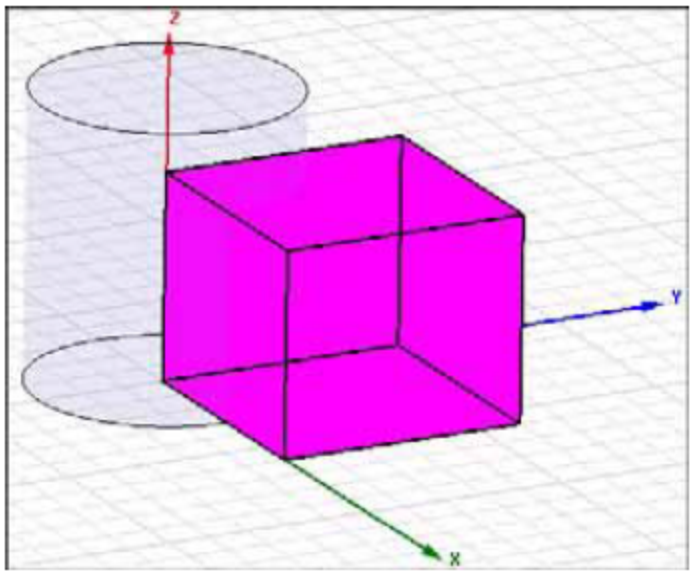
   If the objects you selected do not overlap, the result is a null object, and both objects vanish. When three or more objects have been selected, an intersection is created only if a common volume exists in which all the selected objects overlap. Otherwise, a null object is created and all original objects 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.

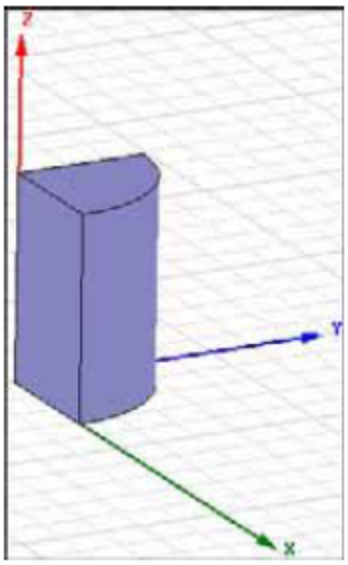
2. From the menu bar, click **Modeler** > **Boolean** >  **Intersect**. (You can also access this command from the **Draw** ribbon tab.)

   The original objects vanish, leaving only the new object that was formed from their intersection.

Ansys Electromagnetics Suite 2026 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.

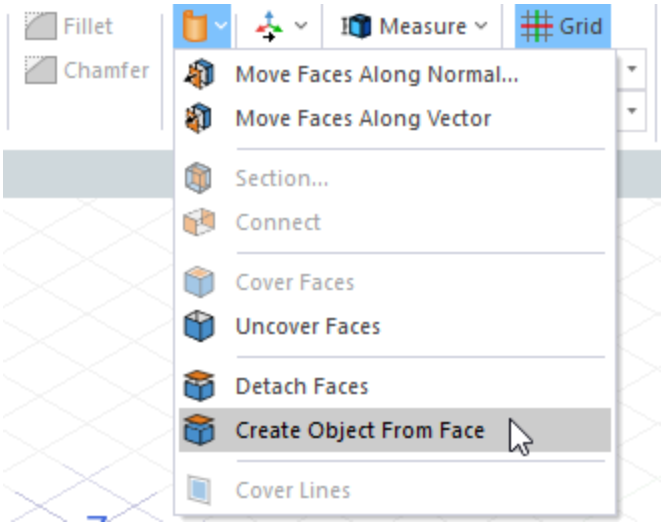
Creating an Object from a Face

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 2026 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

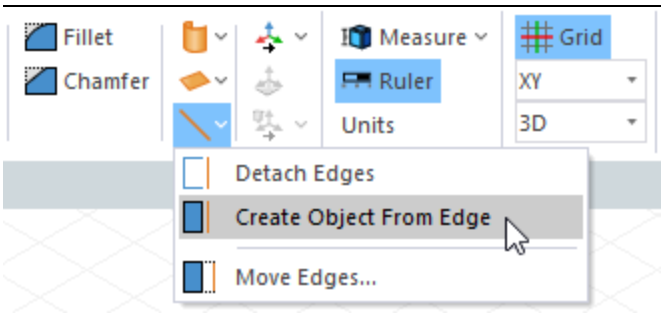
The **Modeler** > **Edge** > **Create Object From Edge** command copies a selected edge, resulting in a new line object.

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 2026 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**:

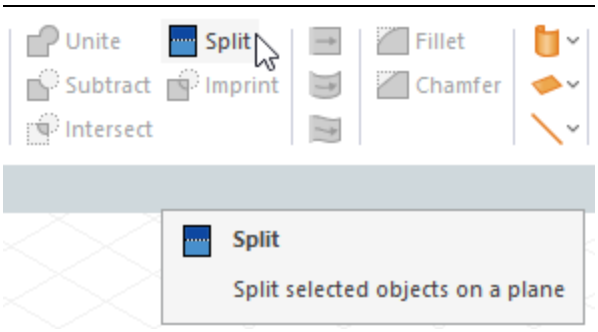


The edge is copied. The resulting object appears in the History Tree as a line object.

Splitting Objects

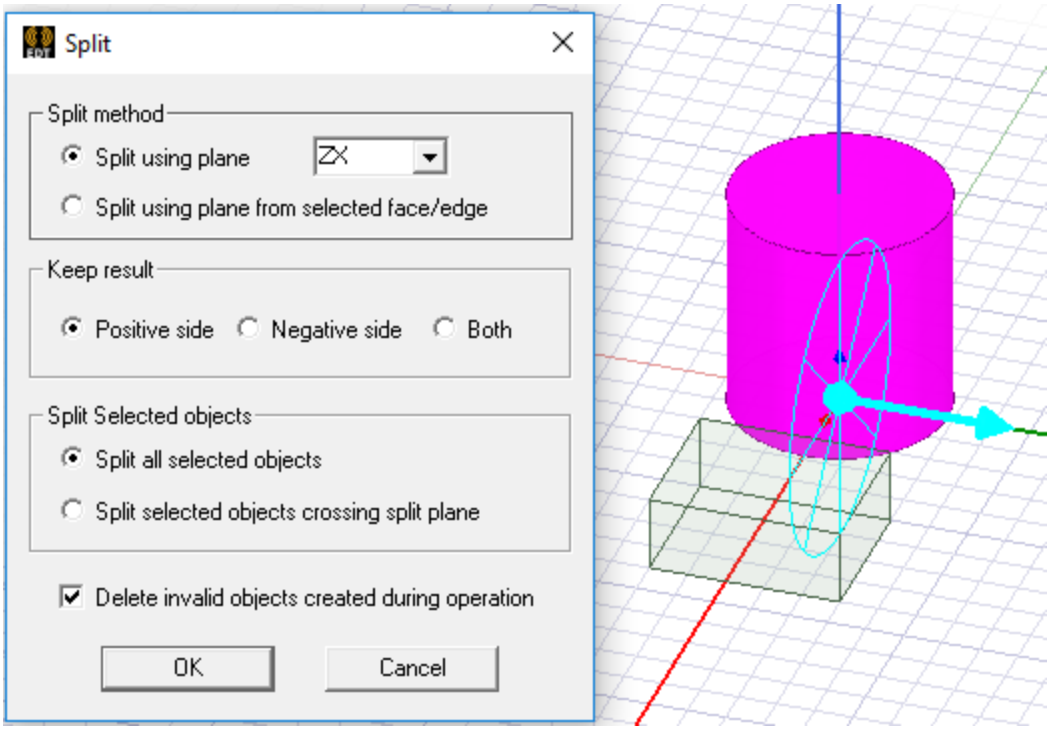
To split an object or several objects using one of the global planes (XY, YZ, or XZ) or a plane based on a selected face or arc, use the **Modeler** > **Boolean** > **Split** command.

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.

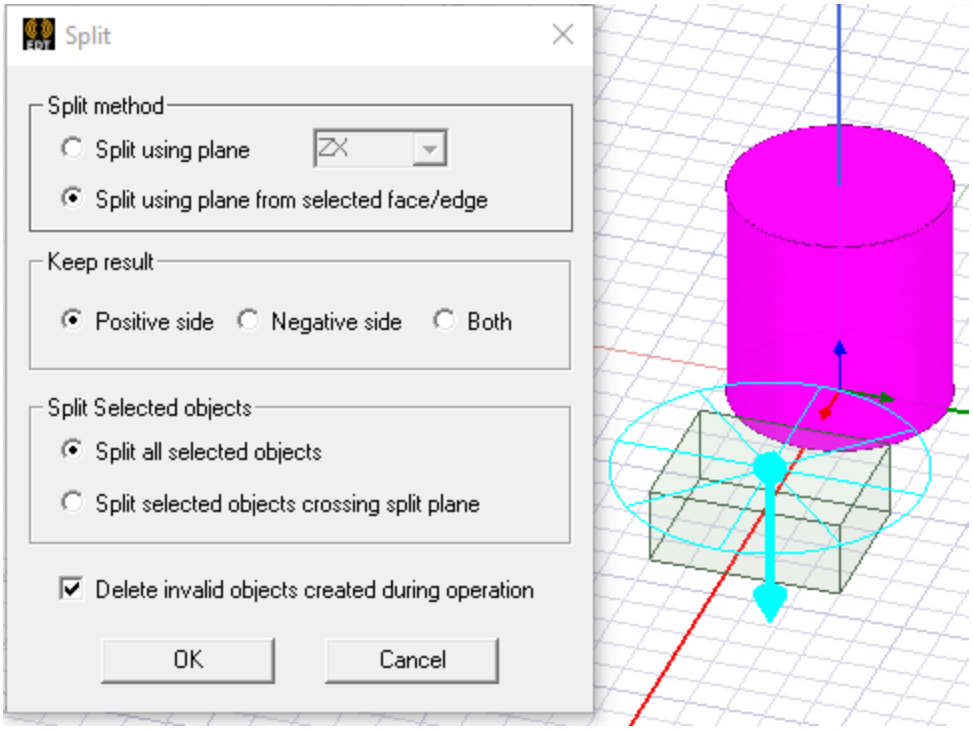
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

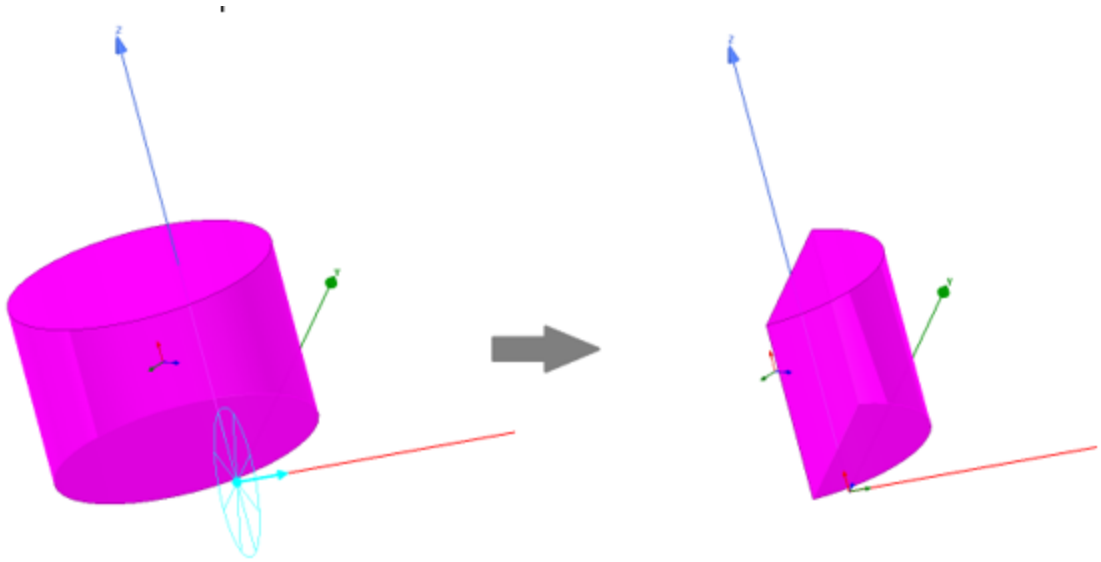


4. Select one of the following *Keep result* options to specify which object fragments you want to keep:
   - **Positive side:** Keep objects on the positive side of the splitting plane.
   - **Negative side:** Keep objects on the negative side of the splitting plane.
   - **Both:** Keep objects on both sides of the splitting plane.

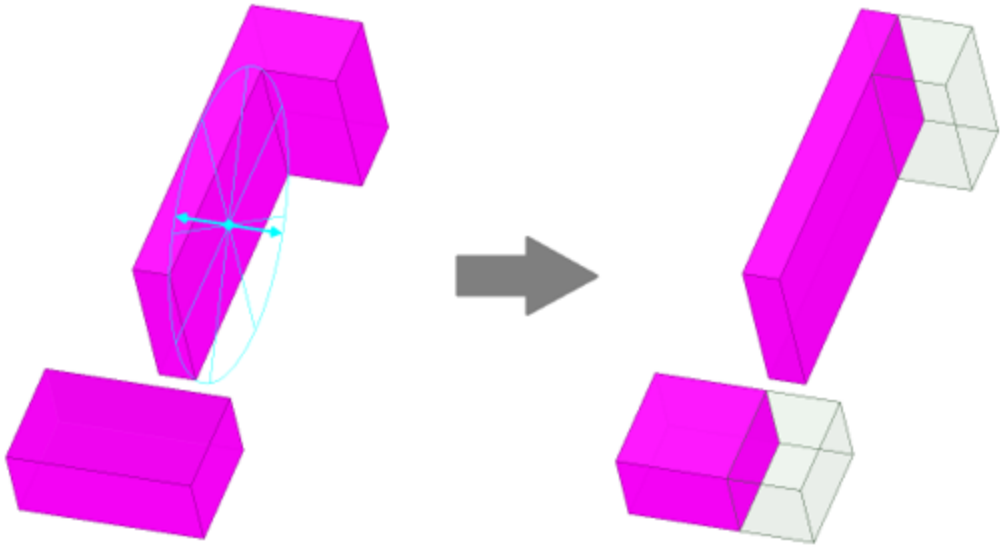
   The split plane is shown using a cyan colored wheel with spokes, as seen in the images above and below left. A cyan arrow indicates which side of the split plane is the positive side.

   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 2026 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 2026 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**.

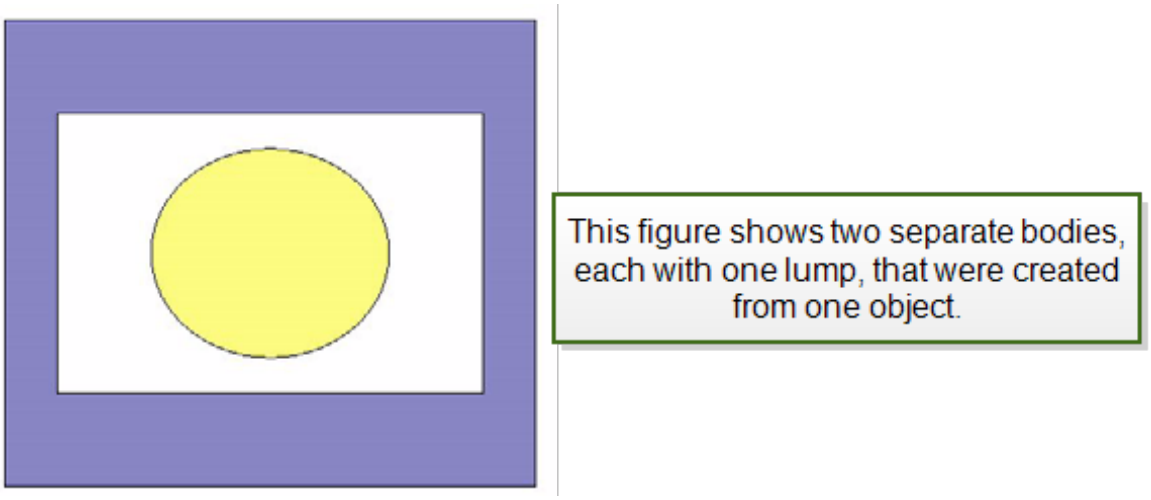
   The objects are divided and retained as specified.

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 2026 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.

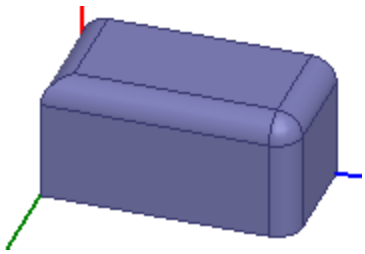
   > **Note:**
   >
   > Converting an arc or spline segment to a straight line segment results in two straight line segments; one segment is created between the start point and midpoint and one segment is created between the midpoint and endpoint.
4. By default, curved segments are generated as smooth (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 2026 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:

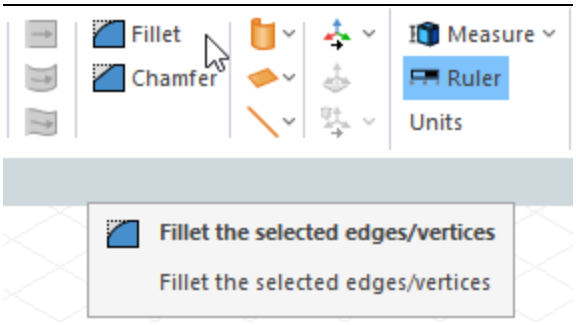
- Right-click the desktop, and choose **Select Edges** from the shortcut menu.

To round the edge of an object:

1. Select the edge you want to change.

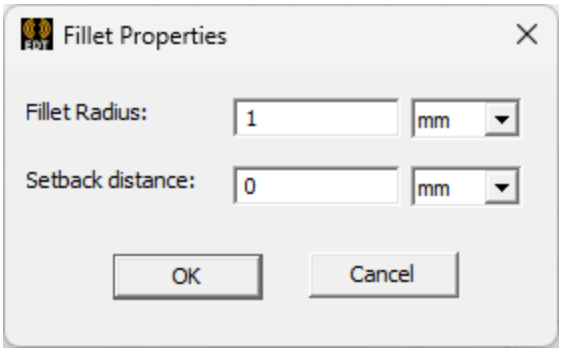
   The edge is highlighted and the *Fillet* command is enabled.

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 2026 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.

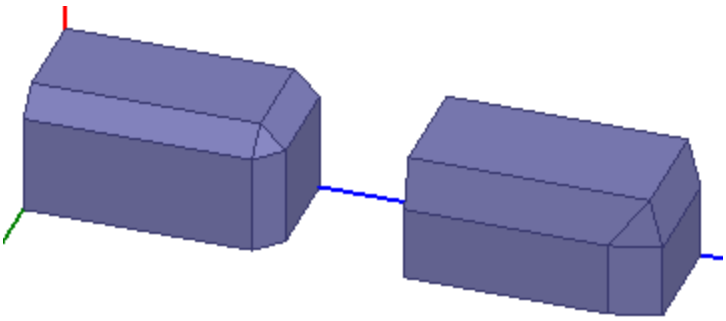
5. Click **OK** to apply the change to the edge.

   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.

Ansys Electromagnetics Suite 2026 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 flat faces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary).

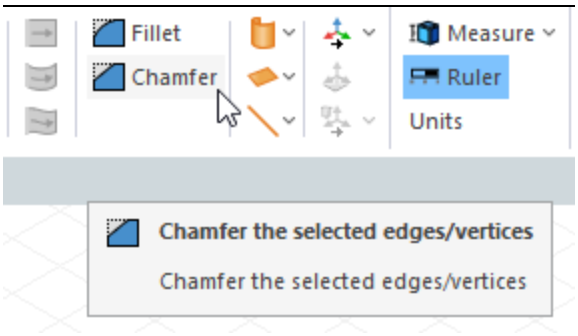
The chamfer command is disabled if an edge is not selected.

To flatten an object's edge for a symmetric chamfer:

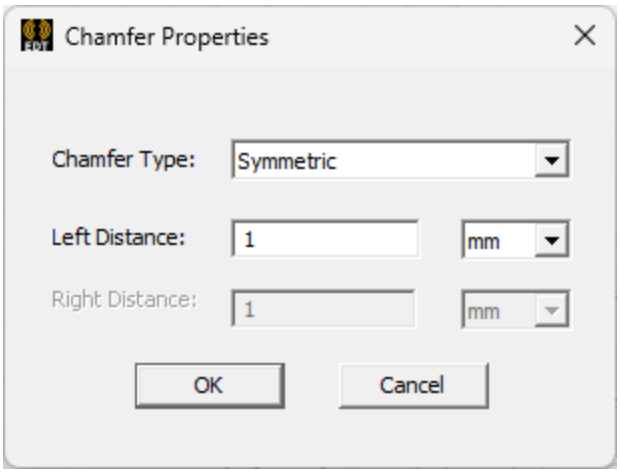
1. Right-click the desktop, and select **Select Edges** from the shortcut menu.
2. Select the edge (or edges) you want to change.

   The edge(s) is highlighted, and the **Chamfer** command is enabled.

3. From the menu bar, click **Modeler** > **Chamfer** or, on the **Draw** ribbon tab, click the **Chamfer** icon:



   The *Chamfer Properties* dialog box appears.



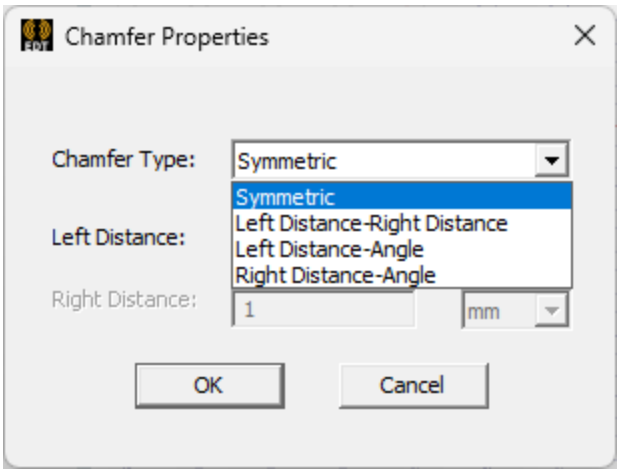
4. Type a **Left Distance** value and select the units from the drop-down menu.
5. Click **OK** to apply the change to the selected edges.

To flatten object's edge for an asymmetric chamfer:

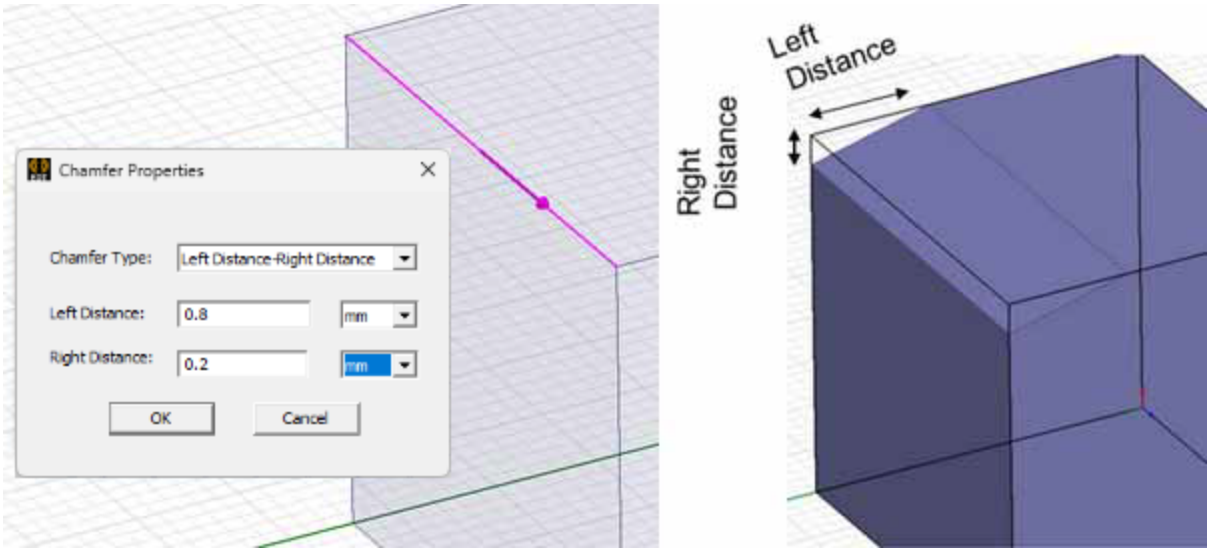
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

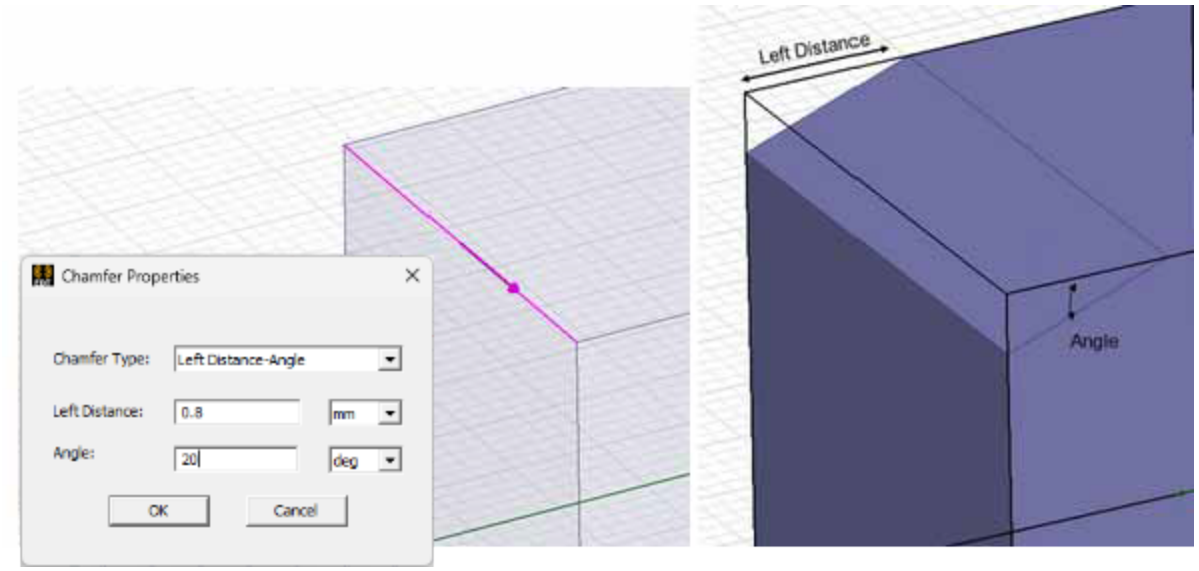


Example 1: Left Distance-Right Distance

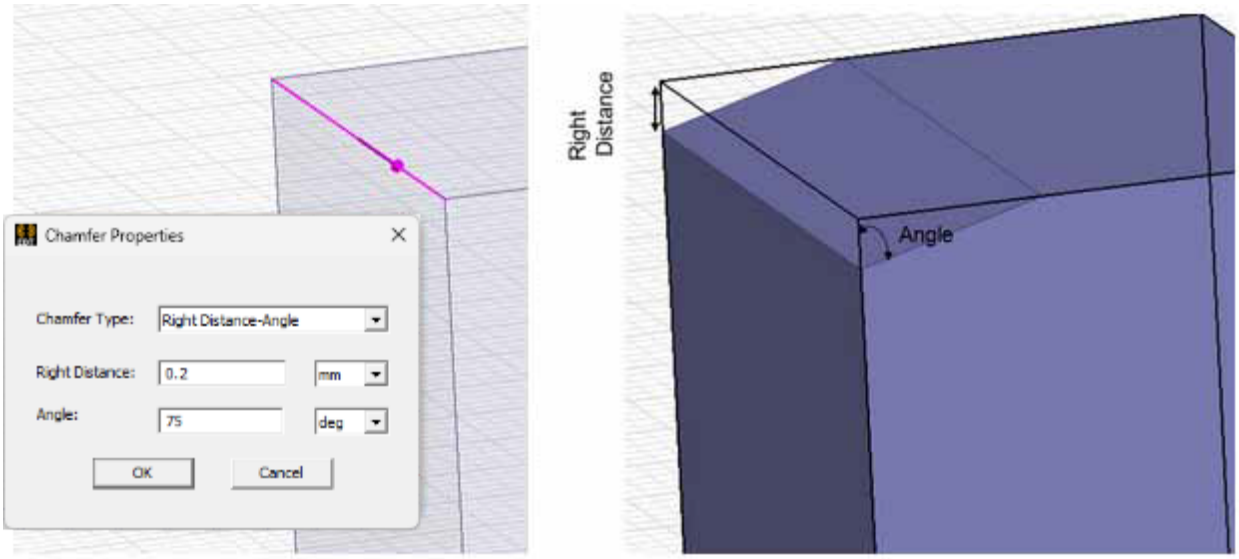


Example 2: Left Distance-Angle

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



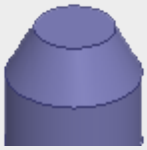
Example 3: Right Distance-Angle



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

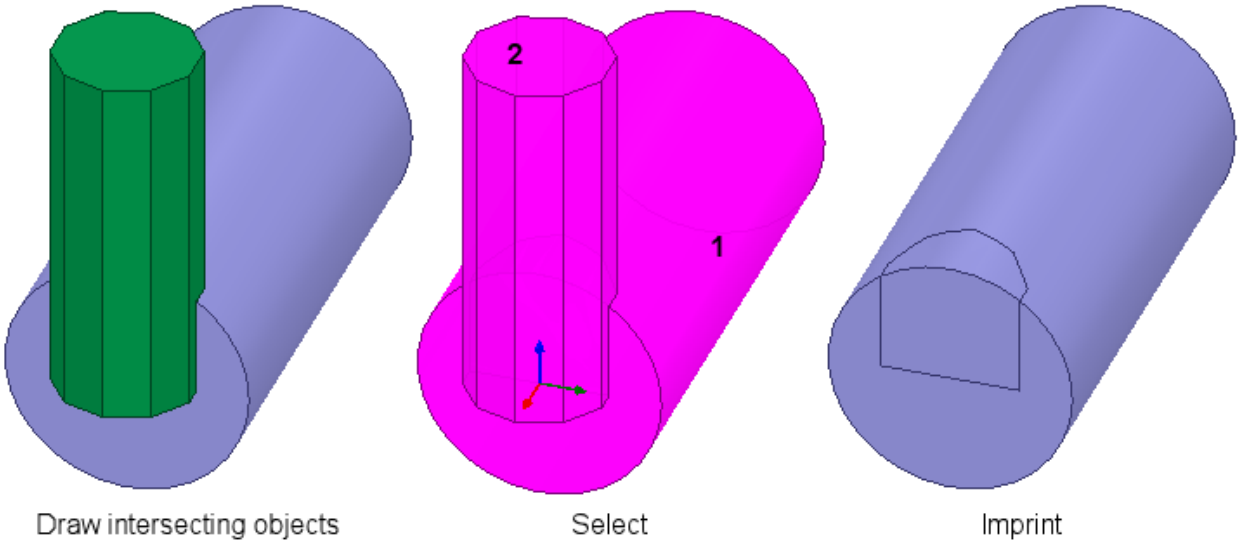
3. Click **OK** to apply the chamfer to the selected edges.

Imprinting an Object

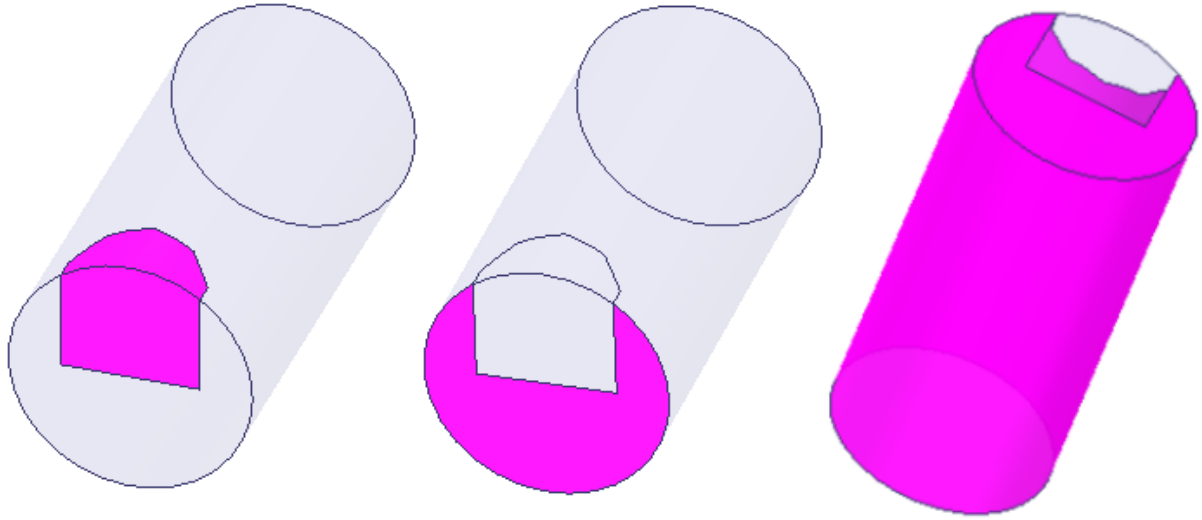
The **Boolean** > **Imprint** command lets you imprint the geometry of one object upon another. For example, you could draw a polyhedron intersecting a cylinder, and then imprint the intersecting lines on the cylinder. This process splits the larger face into two or more faces, with the intersecting 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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



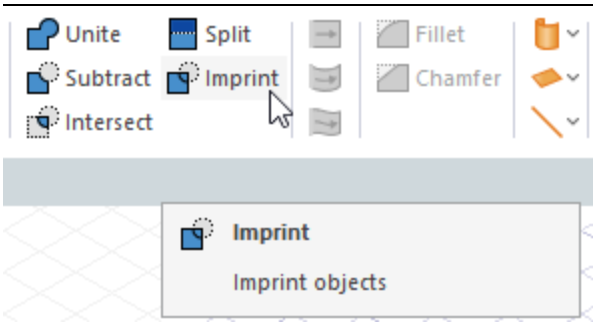
You can select the faces of the imprinted surface separately and assign properties as needed.



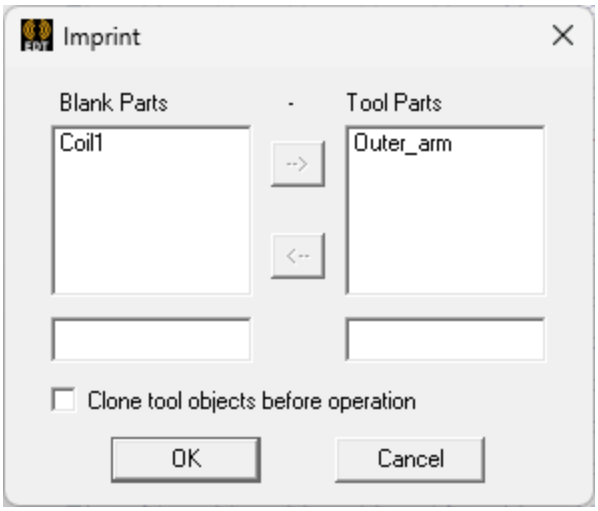
To imprint one object with another:

1. Select the intersecting objects.
2. From the menu bar, click **Modeler** > **Boolean** > **Imprint...** or, on the **Draw** ribbon tab, click the **Imprint** icon:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

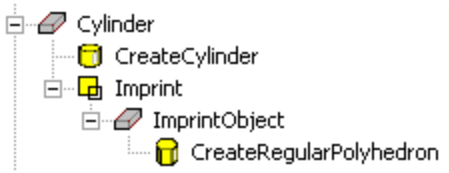


The **Imprint** dialog box displays in which you designate which objects are the *Blank Parts*, and which the *Tool Parts*. If necessary, you can select the objects in lists, and use the arrow keys to move them. If desired, you can clone the tool objects before the imprint operation.



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:



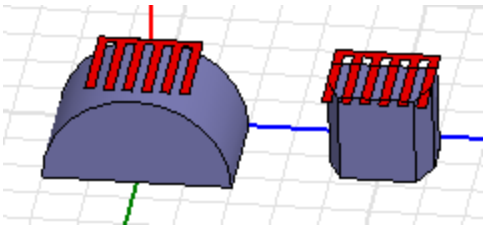
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

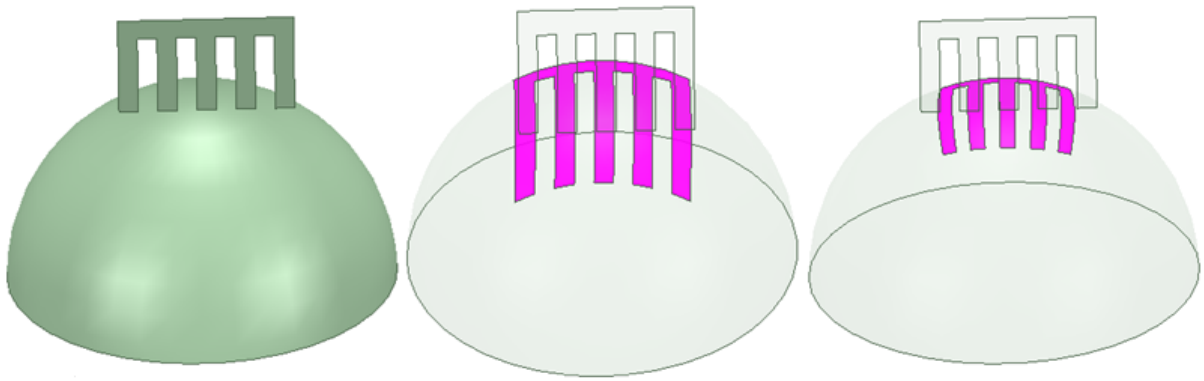
In the *3D Modeler* > *Operation* section of the *General Options*, you can also enable the **Automatically imprint wrapped sheets** option to automatically perform an *Imprint* operation command after executing the **Modeler** > **Surface** > **Wrap** command.

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.



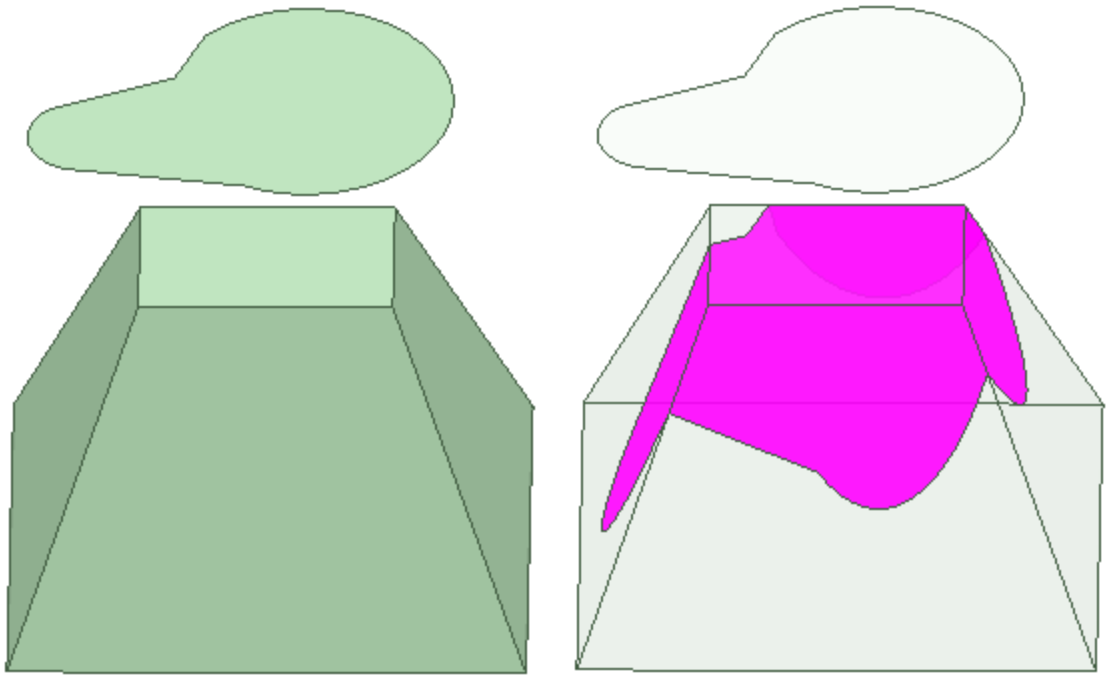
Comb & Hemisphere

Comb Projected Along Vector

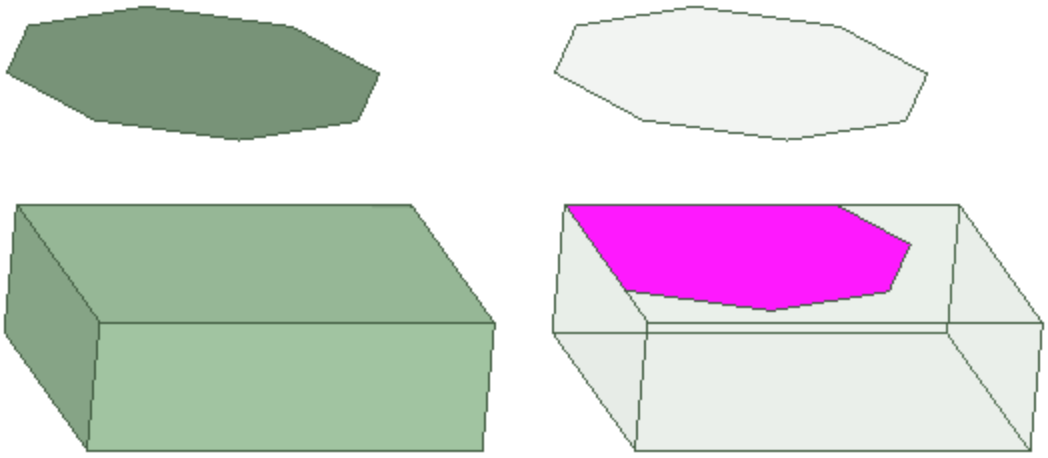
Comb Projected Along Normal (normal to hemispherical target object)

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 2026 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:

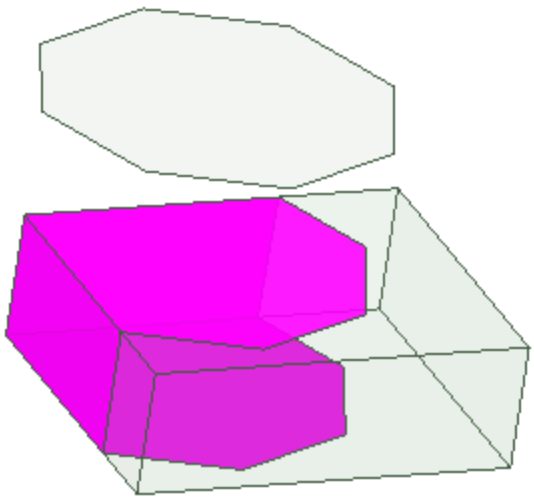


You can select the imprinted faces on the target object separately, and perform subsequent operations (such as creating sheet objects from the faces, assigning boundaries or excitations, or selecting for field overlays).

To perform an Imprint Projection:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

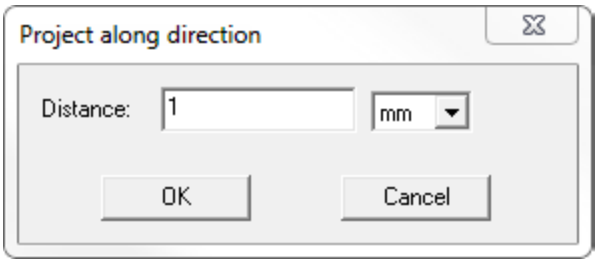
1. If you want to preserve the tool object that will be imprinted on the target object, use one of the following methods:
   - Copy the tool object to the clipboard so that you can paste it back into the model after the imprinting operation.
   - In the *3D Modeler > Operation* section of the *General 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 2026 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 2026 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 2026 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:

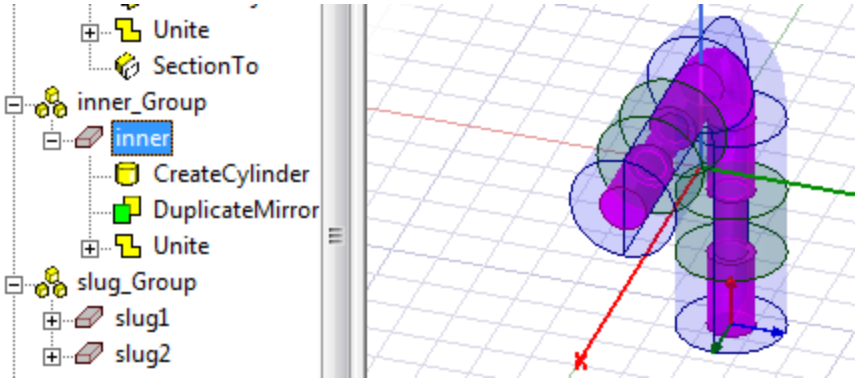
- Model
- Group
- Solids
- Selected Material, for example copper or air.
- Sheets
- Lines

> **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**.

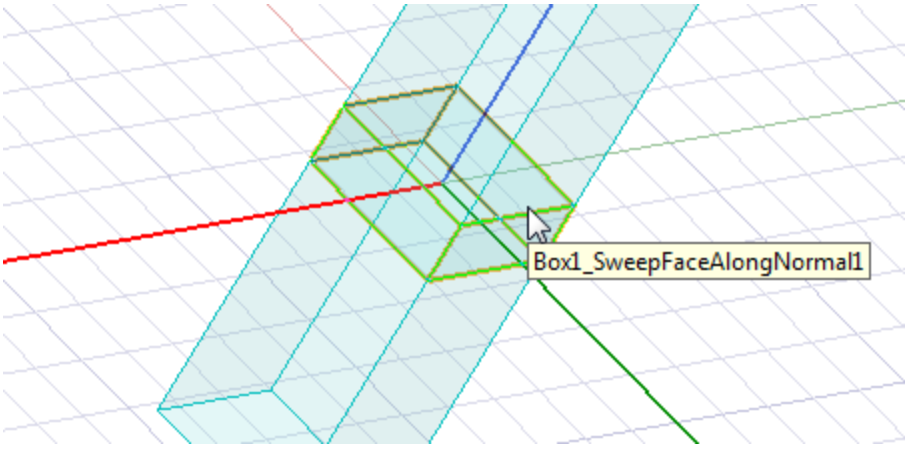
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

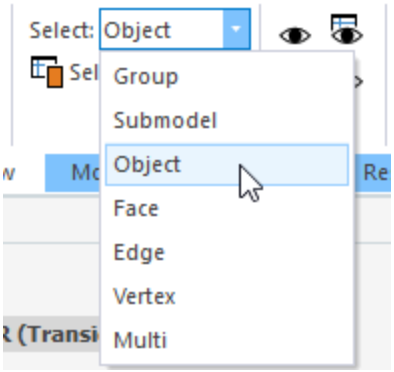


If the modeler is not currently in object selection mode, you can switch to it using one of the following methods:

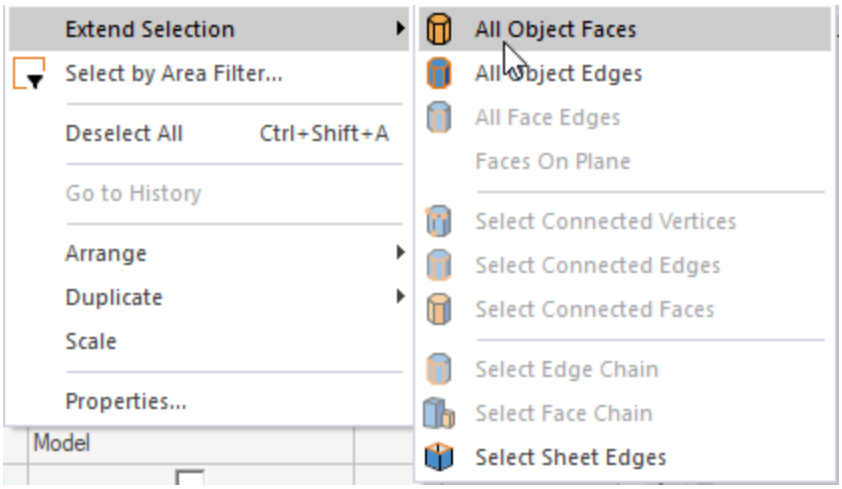
- Press the **O** shortcut key while the Modeler window is active.
- Right-click in the Modeler window and click **Selection Mode** > **Select Objects**.
- From the menu bar, click **Edit** > **Selection Mode** > **Objects**.

Ansys Electromagnetics Suite 2026 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:

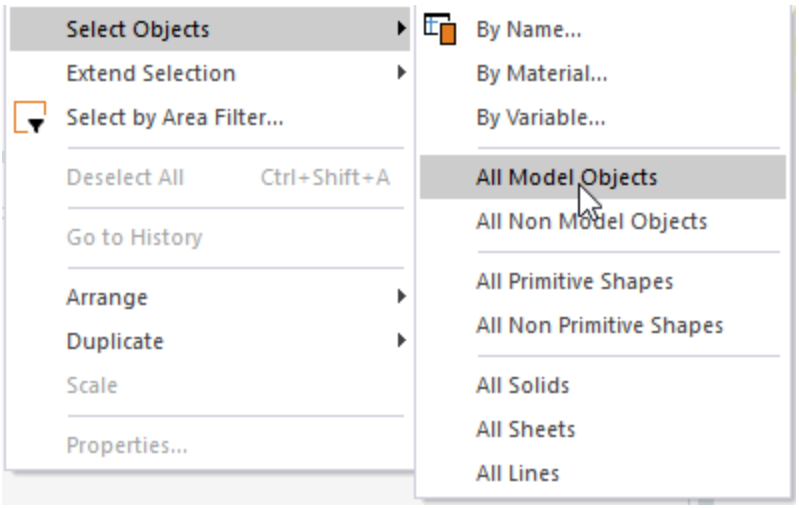


Selecting Several Objects

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 2026 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.

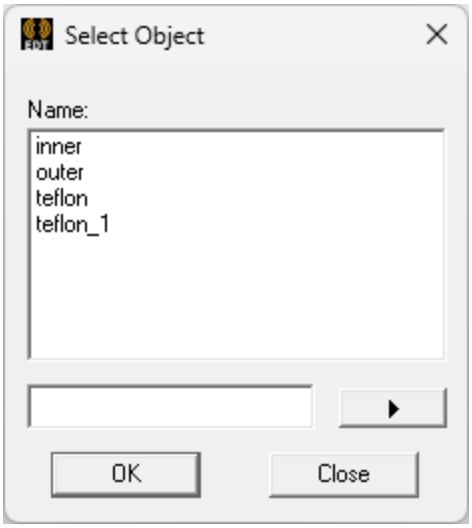
Selection by Object Name

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**.

Ansys Electromagnetics Suite 2026 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 **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**.

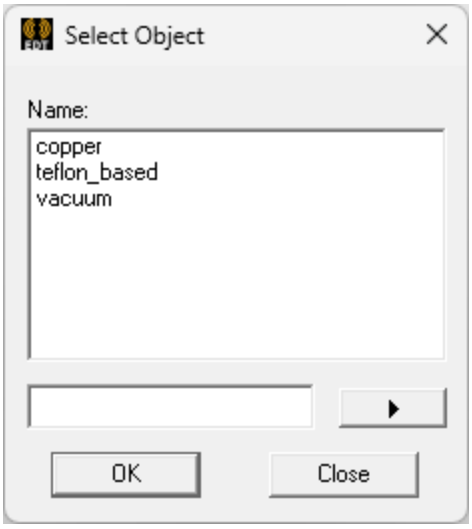
   The object or objects specified are selected.

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 2026 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.

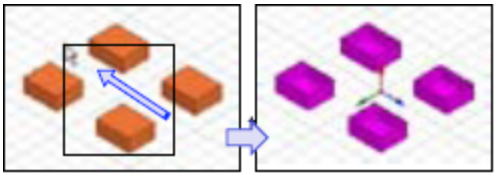
   Alternatively, type the name of a material in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

Select By Area

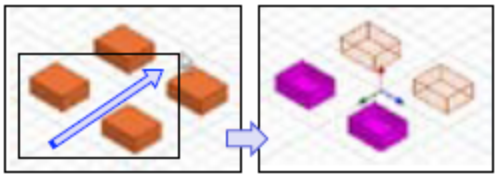
You can select items by area in the Modeler window by clicking and dragging a rubber band (selection box) around objects. Rubber band selection works differently depending on the direction you drag the mouse:

- **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.

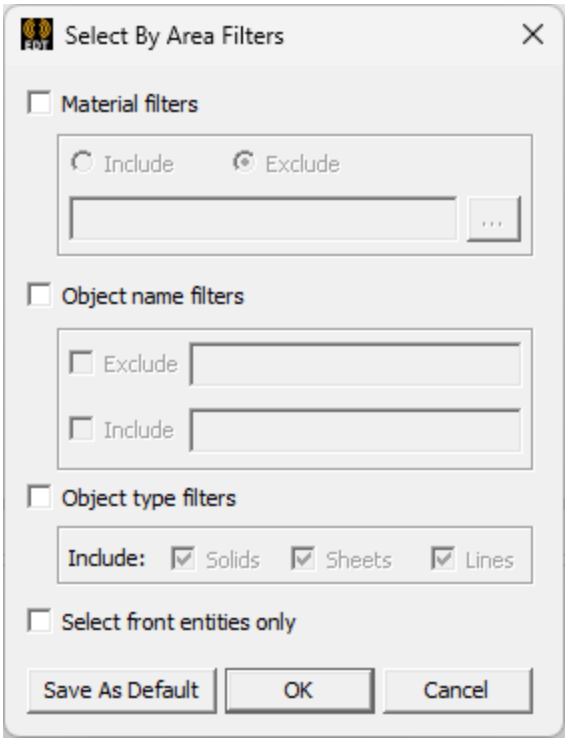
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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 Selection

Create an object selection when you want to group multiple objects for convenient selection and processing. Objects in a selection can still be treated as separate objects. The same object can be included in several different selections.

To create an object selection:

1. If you are selecting in the Modeler window, make sure that the modeler is in select object 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 selection.
3. Click **Modeler** > **Named Selection** > **Create** > **Object Selection**.

   The object selection is created with the default name ObjectSelection*n*. It appears in the History Tree under **Named Selection**. Selecting an object selection displays the properties of that selection in the docked *Properties* window. One of the properties is a list of objects contained in the selection.

   To rename the object selection, edit its Name in the docked *Properties* window. Object selections are sorted in alphanumeric order.

There is an automatically created selection called *AllObjects* included in the **Named Selection** branch. Click on this entity to select all objects in the model.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The object selection 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**.

If a selection 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 selection do not affect where the fields are saved. If all objects in the selection are lines, fields will be saved on all objects.
- To postprocess on line objects when saving fields on an object/face selection, include both the line and its containing object/face in the selection.

Example: To plot the E-field on a surface formed by the intersection of the XY-plane and several objects, first define a selection of these objects. Then, when plotting fields, select the object selection name from the **Geometry** window of the *Fields Calculator*. Fields will be plotted only at the intersection of the plane and the objects in the selection.

Reassigning Objects to Another Object Selection

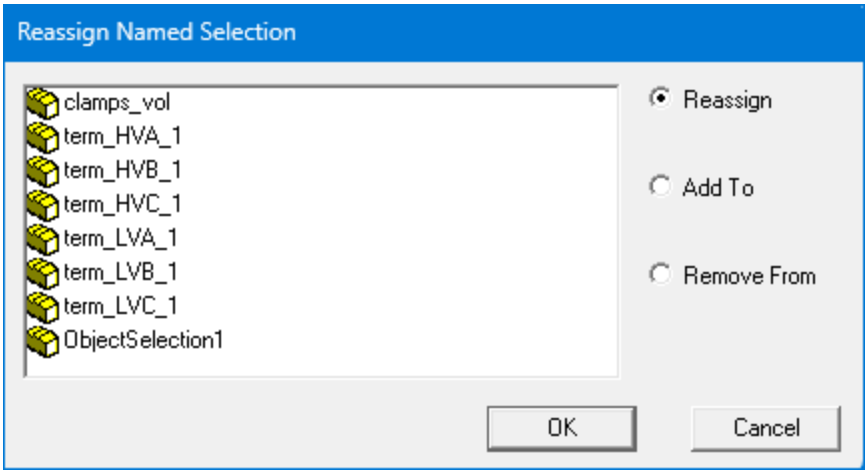
You can assign objects after you have created an object selection. Creating an object selection is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a selection can still be treated as separate objects. The same object can be included in several different selections.

To reassign objects to an existing object selection:

1. If you are selecting in the Modeler window, make sure that the modeler is in select object 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** > **Named Selection** > **Reassign**.

   A dialog box with the existing object selections is displayed. (They appear in the History Tree under **Named Selections**.)

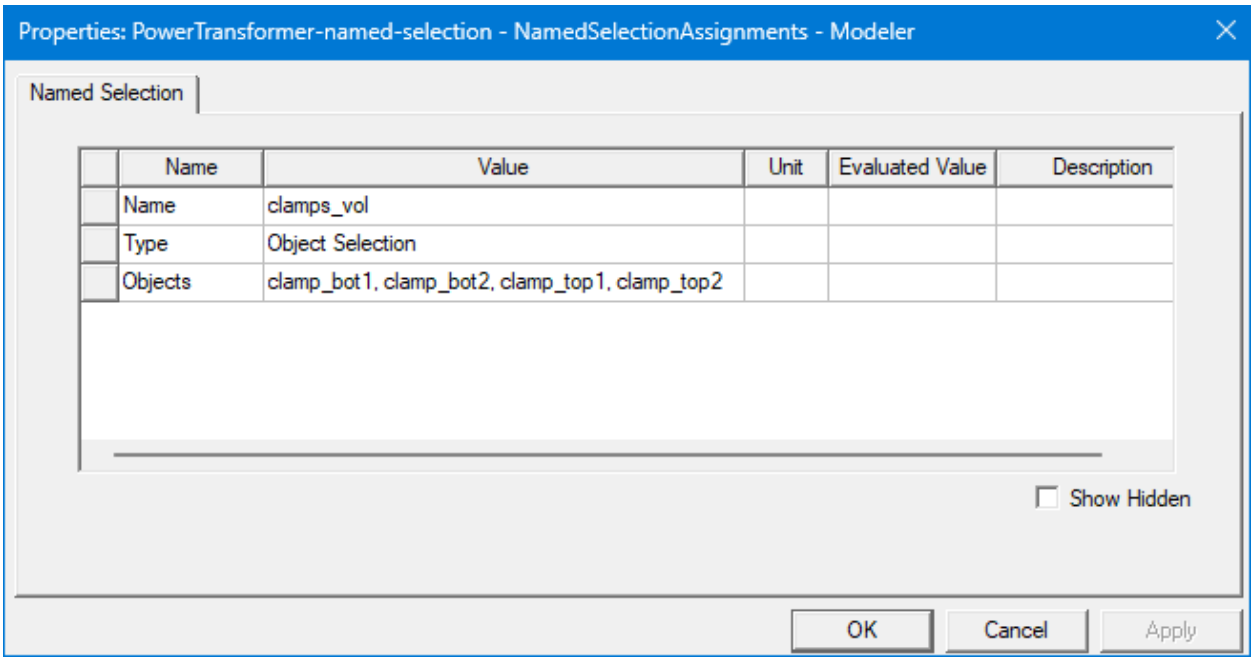
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click the named selection that you want to modify, and then select **Reassign**, **Add To**, or **Remove From**.
5. Click **OK**.

   If **Reassign** was selected, the object is reassigned to the active selection, **replacing** previous list members. The **Add To** and **Remove From** options act only on the selected objects.

The *Objects* property of the selection shows the objects contained in the selection, with each name separated by a comma.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The object selection 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 Selection Objects

To view the objects included in an object selection:

1. In the History Tree, expand the **Named Selection** branch.
2. Click on the desired selection to highlight the objects included in the named selection.

   or

   Right-click the desired selection, and click **Select Assignment** to highlight the objects in the named selection and switch the other design objects to wireframe. This option makes the object selection more visible.

The selection attributes appear in the docked *Properties* window.

To view the properties of the object selection (including a list of the objects in the selection):

1. In the History Tree, expand the **Named Selection** branch.
2. Right-click the selection you want to view and click **Properties**.

   The *Properties* dialog box appears. The objects included in the selection appear as comma-separated names in the *Objects* row.

3. Click **OK** or **Cancel** to close the *Properties* dialog box.

To use an object from a selection in another operation:

1. In the History Tree, expand the **Named Selection** branch.
2. Right-click the desired selection and click **Select Assignment**.

   The objects that are included in that selection are highlighted in the Modeler window, and the attributes appear in the docked *Properties* window.

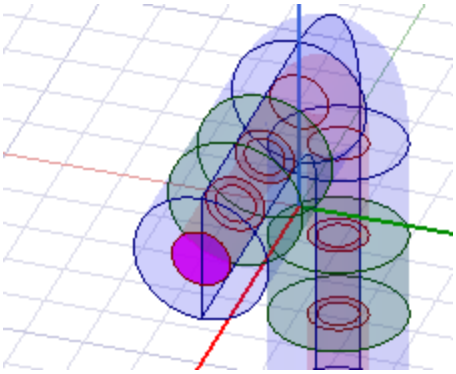
3. Holding **Ctrl**, select any additional objects you might want to use in the operation.
4. Complete the operation.

   For example, you could select an object selection 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.

Ansys Electromagnetics Suite 2026 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 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 selections, 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:

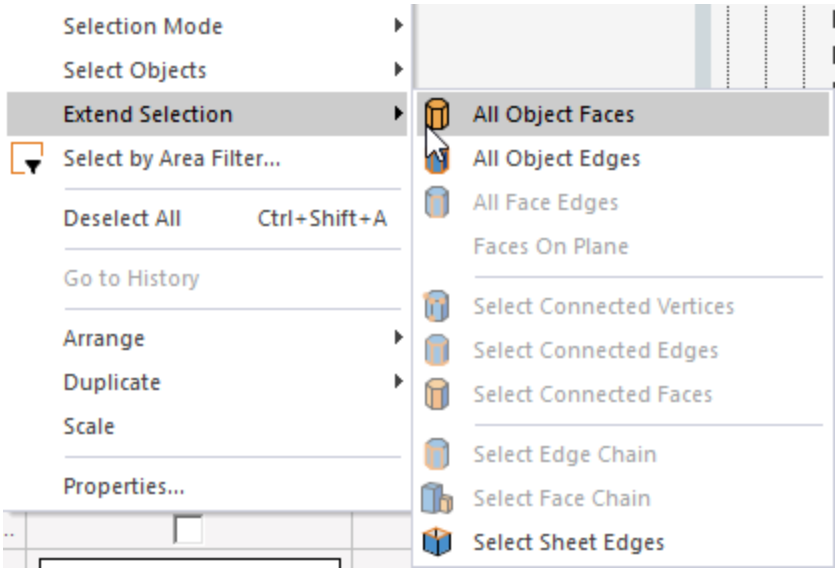
- From the menu bar, click **Edit** > **Selection Mode** > **Select Faces**
- Right-click in the Modeler window and choose **Selection Mode** > **Select Faces** from the shortcut menu.
- With the Modeler window active, press the **F** shortcut key.
- On the **Draw** ribbon tab, choose **Faces** from the **Select** drop-down menu.

  3.202

  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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Various tools are available for face selection, as described in the *Related Topics* listed below.

Selecting All Faces of an Object

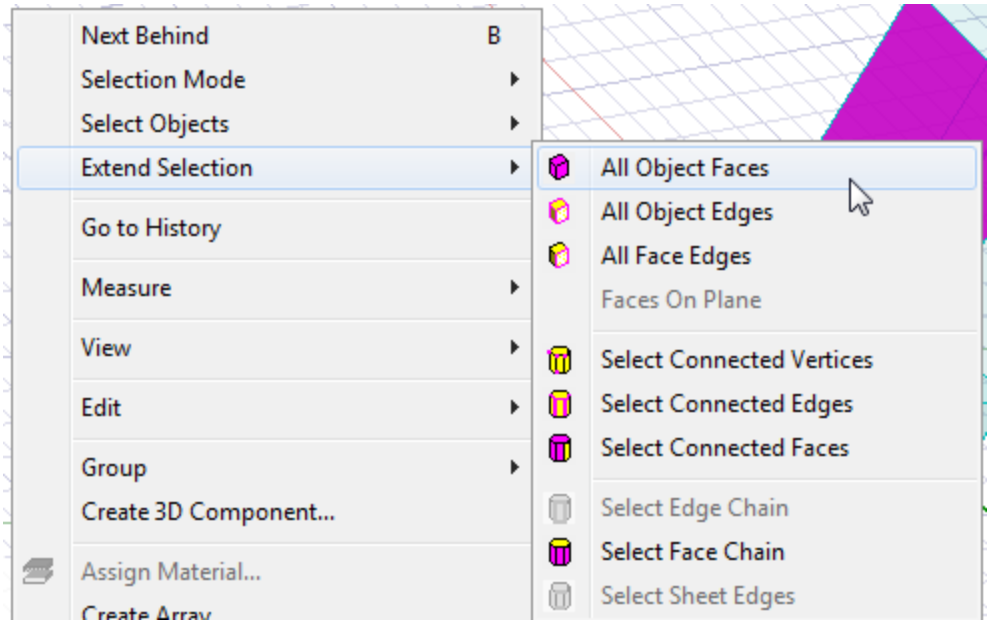
1. With the Modeler window active, switch to the face selection mode by pressing the **F** shortcut key.
2. Click a face on the object of interest to select it.

   You can select a face on each of two or more objects if desired.

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**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

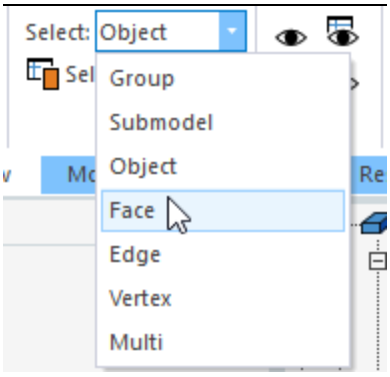
Faces from the shortcut menu:



The selection is extended to include all faces of each object that had at least one face selected originally.

Selecting Faces by Name

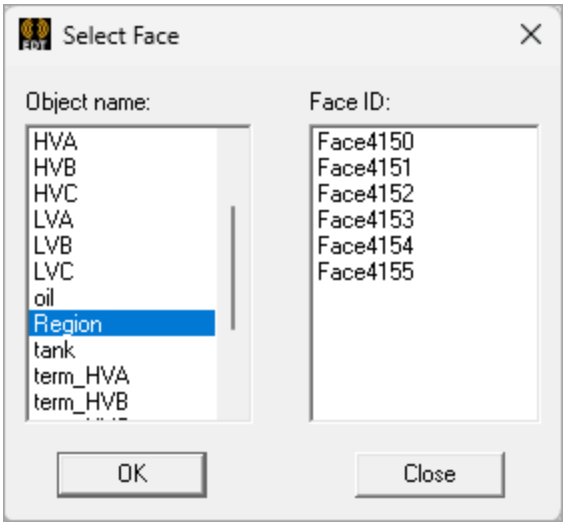
1. Make sure that the modeler is in face selection mode using one of the following methods:
   - With the Modeler window active, press the **F** shortcut key.
   - From the menu bar, click **Edit** > **Selection Mode** > **Faces**
   - On the **Draw** or **Model** ribbon tabs, select **Face** from the **Select** drop-down menu.



2. From the menu bar, click **Edit** > **Select Objects** >  **By Name,** or on the **Draw** or **Model** ribbon tabs, click  **Select by Name**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

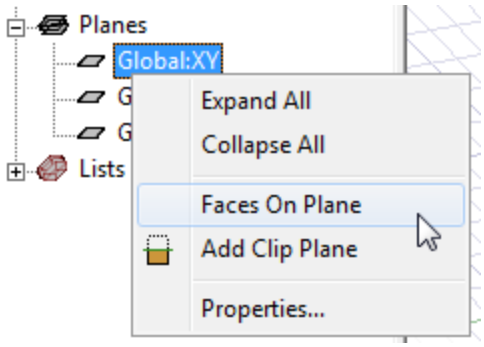
   All faces that lie on the chosen plane are selected.

Method 2:

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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the shortcut menu, click **Faces on Plane**.



All faces that lie on the chosen plane are selected.

Creating a Face Selection

Create a face selection when you want to group multiple faces for convenient selection and processing. Creating a face selection is a convenient way to identify and select a specific set of surfaces for boundaries, excitations or mesh, and for field plots and calculations. The same face can be included in several different selections.

To create a face selection:

1. Make sure that the modeler is in select face mode by pressing the **F** shortcut key.
2. Select the faces you want to include in the face selection.
3. From the menu bar, click **Modeler** > **Named Selection** > **Create** > **Face Selection**.

   The face selection is created. It is listed in the History Tree under **Named Selection**. The default name is FaceSelection*n*. The selections appear in alphanumeric order.

4. Optionally, to change the name of a face selection (for example, to a name describing the listed faces as ports or boundaries), select its entry in the History Tree and change the **Name** value in the docked *Properties* window.

   If necessary, the selections in the History Tree are resorted to restore the proper alphanumeric order.

The face selection will be treated as one group of surfaces when you are plotting and performing fields calculations. The face selection will be listed in the *Geometry* window of the *Fields Calculator*, when you select the *Surface* option.

- Line objects in the selection do not affect where the fields are saved. If all objects in the selection are lines, fields will be saved on all objects.
- For postprocessing on line objects, when saving fields on a object/face selection, include both the line and its containing object/face in the selection.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you right-click an existing face selection and choose **Select Assignment** from the shortcut menu, you can make a boundary assignment in the Modeler window to all faces in the selection, or you can execute the **Move Faces** geometry operations on the selection as a whole.

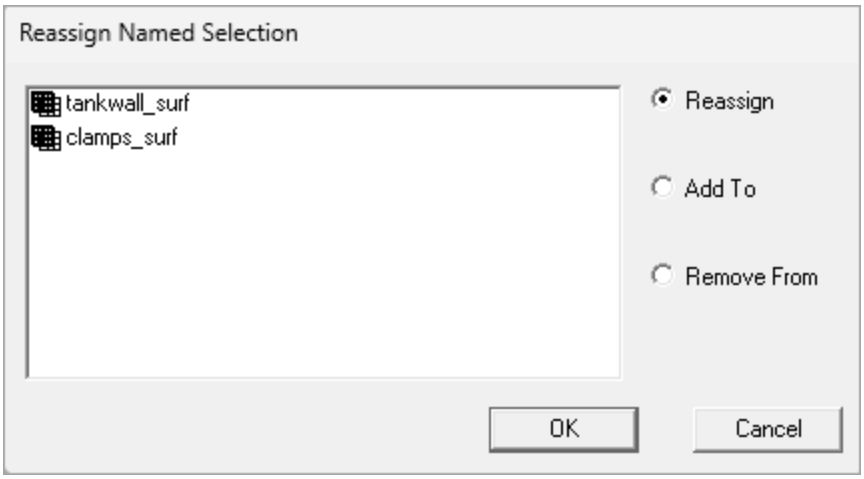
Reassigning Faces to Another Face Selection

You can assign faces after you have created a face selection. Creating a face selection is a convenient way to identify and select a group of faces for a field plot or calculation. Faces in a selection can still be treated as separate faces. The same faces can be included in several different selections.

To reassign faces to an existing face selection:

1. If you are selecting in the Modeler window, make sure that the modeler is in select face mode by pressing the **F** shortcut key. This is not necessary for selecting in the History tree.
2. Select the faces you want to reassign.
3. Click **Modeler** > **Named Selection** > **Reassign**.

   A dialog box with the existing face selections is displayed. (They appear in the History Tree under **Named Selections**.)



4. Click the named selection that you want to modify, and then select **Reassign**, **Add To**, or **Remove From**.
5. Click **OK**.

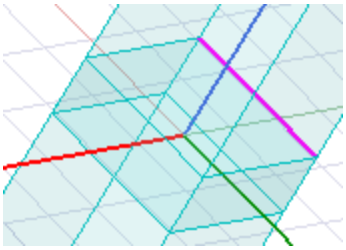
   If **Reassign** was selected, the face is reassigned to the active selection, **replacing** previous list members. The **Add To** and **Remove From** options act only on the selected faces.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The face selection will be treated as one surface when you are plotting and performing fields calculations. It will be listed in the *Geometry* window of the *Fields Calculator*, when you select **Surface**.

Selecting Edges

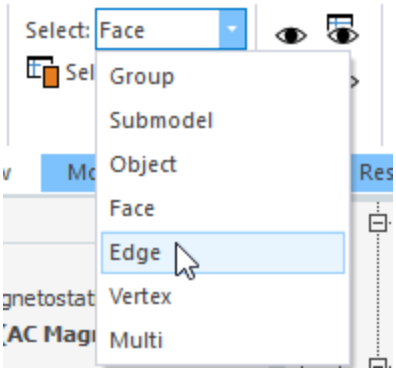
If the modeler is in edge selection mode, simply click an object's edge in the view window and it will be selected. The edge is rendered as nearly opaque using the default *Select* color. All other objects become relatively transparent.



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.

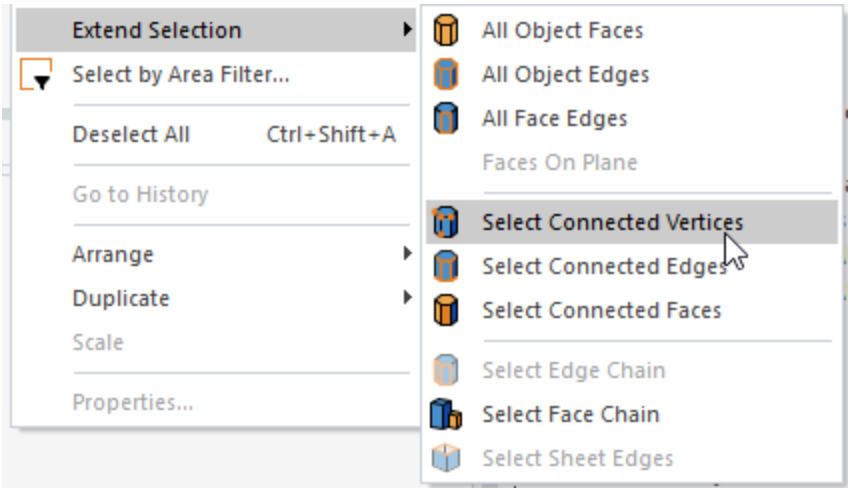
Switch to edge selection mode using one of the following methods:

- 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 2026 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.

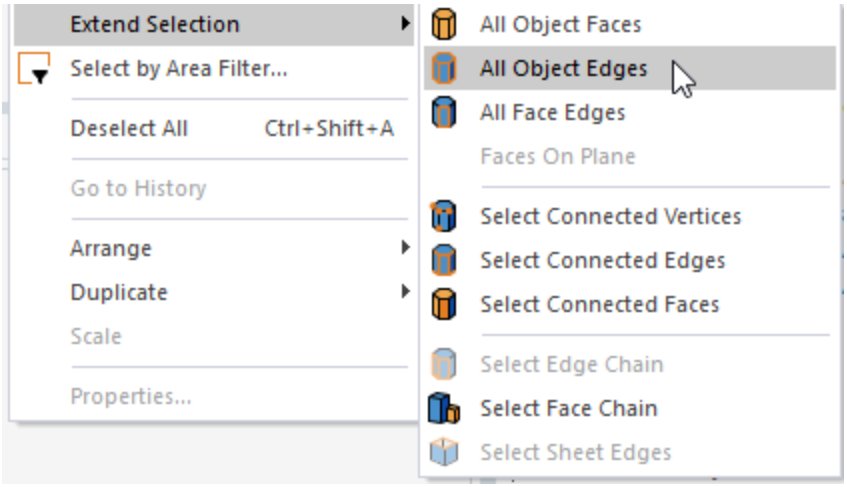
Selecting All Edges of an Object or Face

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 2026 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

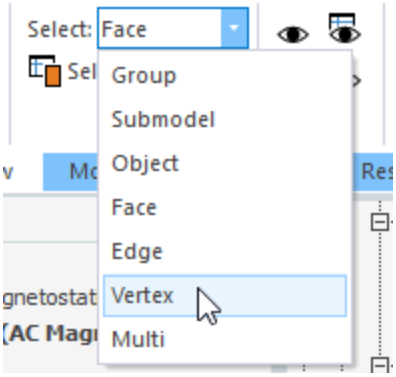
If the modeler is in vertex selection mode, simply click an object's vertex in the view window and it will be selected. To select multiple vertices, hold the **Ctrl** key as you click the vertices.

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.

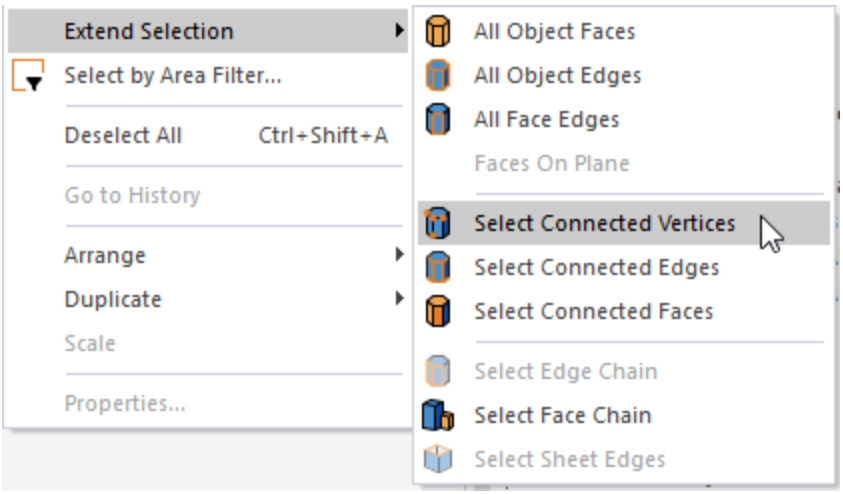
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
- **Select Connected Edges:** Selects all edges that touch the initially selected vertices.
- **Select Connected Faces:** selects all faces touching the initially selected vertices.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

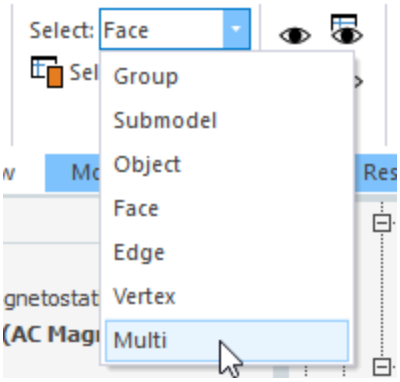
Multiple vertices can exist at the same coordinates, one vertex for each object that intersects at that point. There's no visual way to discern to which object a selected vertex belongs. If you click a vertex and use any of the commands listed above, only the entities that belong to the same object will 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.



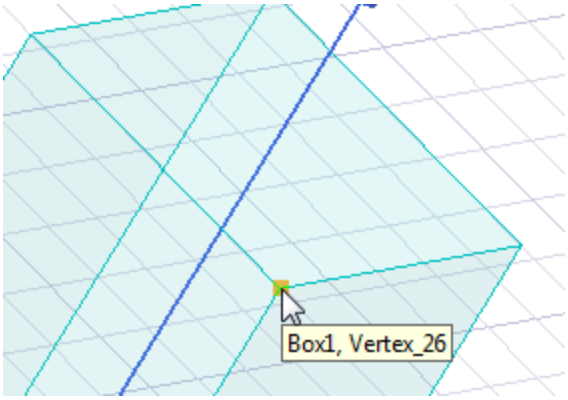
With Multi mode active:

- 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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

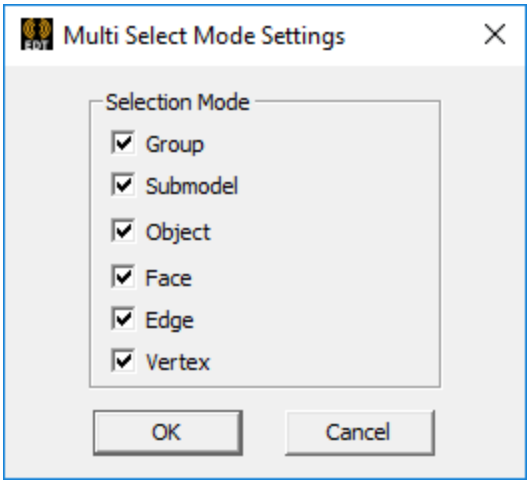
As you hover the cursor over an entity, a Tooltip indicates the type/ID of the entity (object name in the case of objects, Face_id in the case of faces, and so on). For example, this feature helps you to distinguish between the selection of the face of a sheet object versus a sheet object.



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):



Clear a box filters out the associated item, preventing that type of entity from being selected while the *Multi* selection mode is active.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

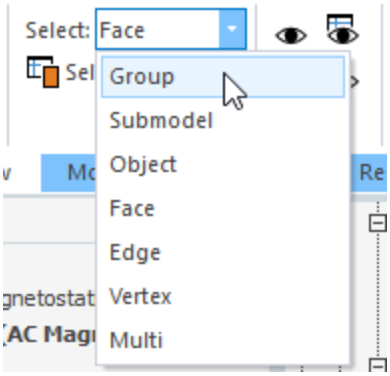
Selecting Groups and Submodels

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:

- 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.

Ansys Electromagnetics Suite 2026 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** > **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.

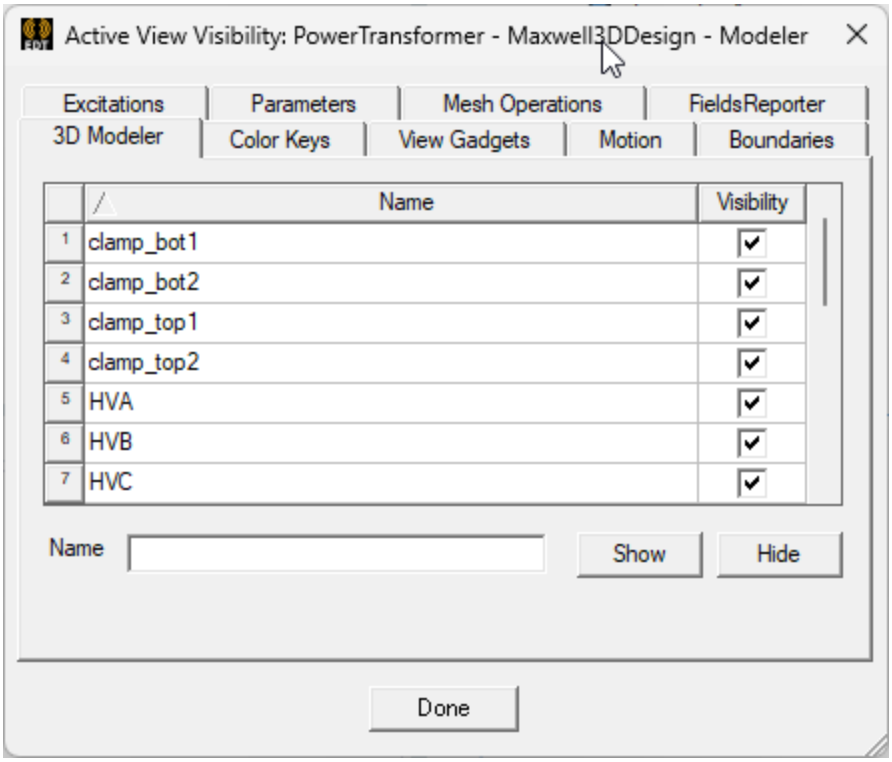
Selecting Objects and Surfaces That Lie Inside Other Objects

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.

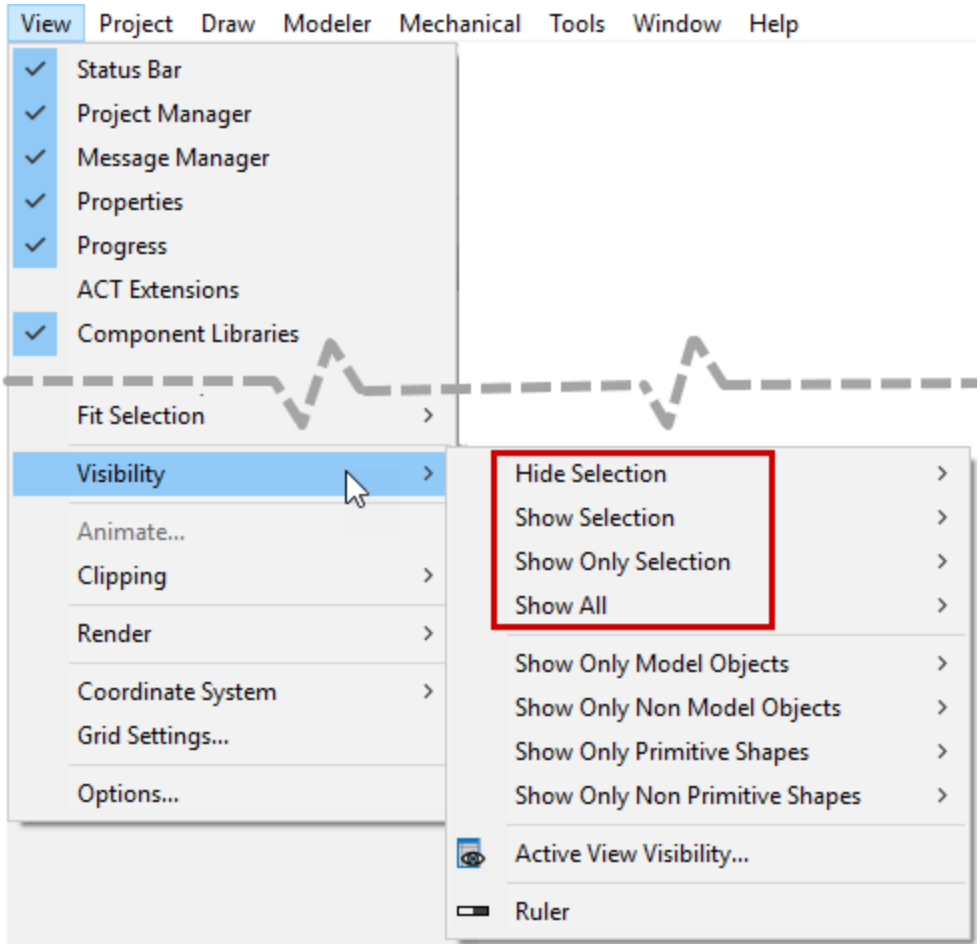
  This method, and the method that follows, is useful when you want to select objects using the mouse, since the mouse cannot select invisible objects.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

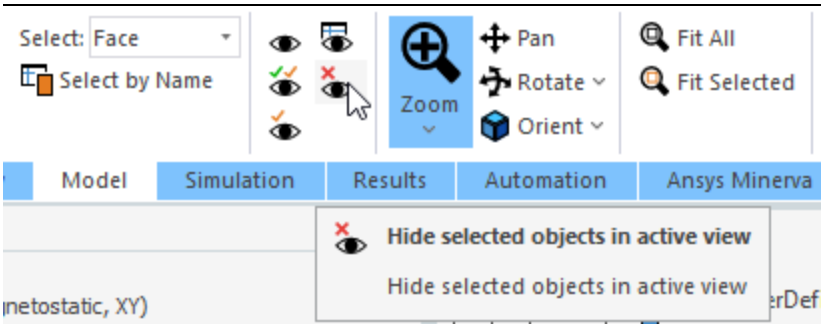


- Make the objects on the outside of the model invisible using the menu bar's other **View** > **Visibility** > commands:

Ansys Electromagnetics Suite 2026 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.

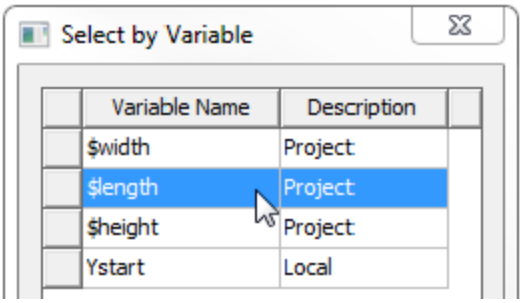
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

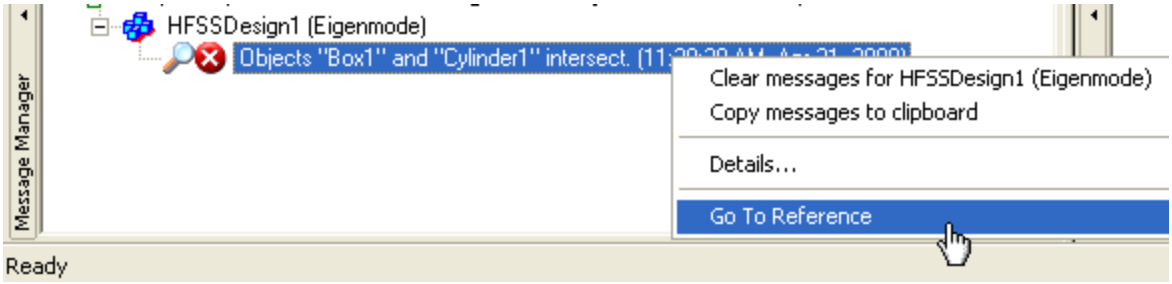
   The *Select by Variable* dialog box appears, which lists the variables in your design. Both columns are sortable by clicking the header. You can resize and move the dialog box. When you next open it, it remembers that size and location.



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 box.

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.

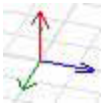
- Setting the Reference Point
- Defining Cartesian Coordinates

Ansys Electromagnetics Suite 2026 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 2026 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.

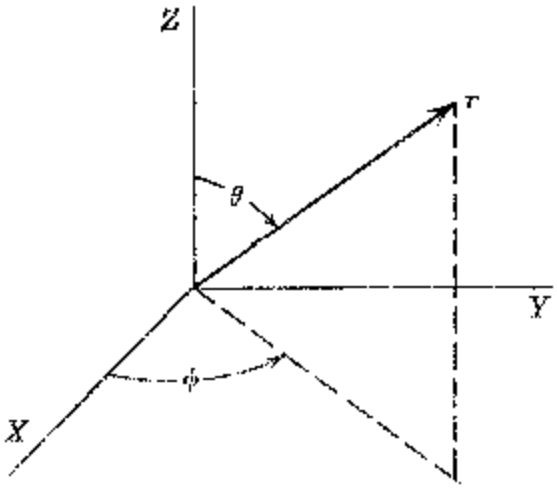
   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - 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 2026 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 2026 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:

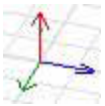


3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

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 2026 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.

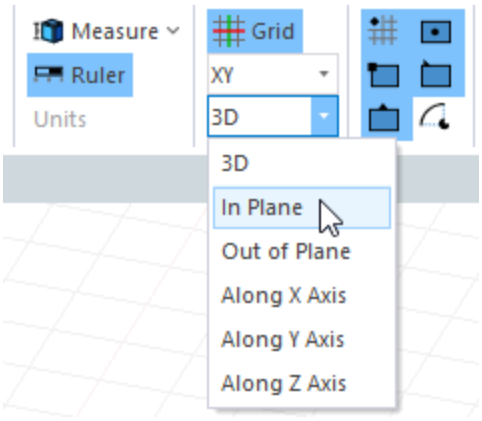
Moving the Cursor In Plane

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

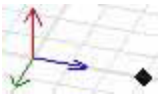
- From the menu bar, click **3D Model** > **Movement Mode** > **In Plane**.
- On the **Draw** ribbon tab, choose **In Plane** from the **Movement mode** drop-down menu:



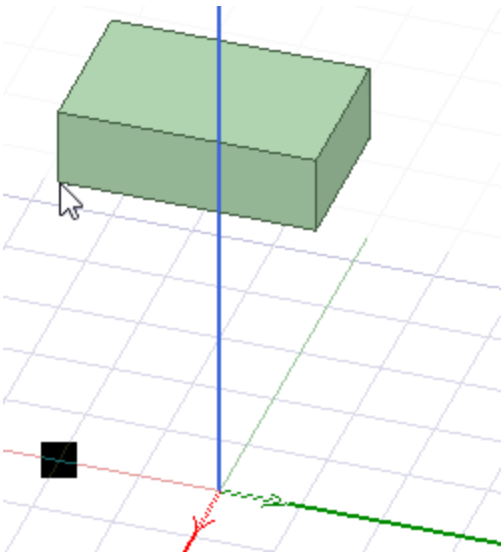
The following additional method is available only after clicking a drawing command:

- 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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The symbol used for projected point matches the type of snapping point on the object (vertex, edge center, face center, and so on).

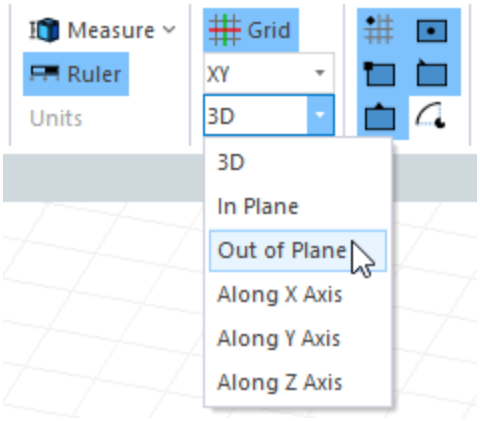
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

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:

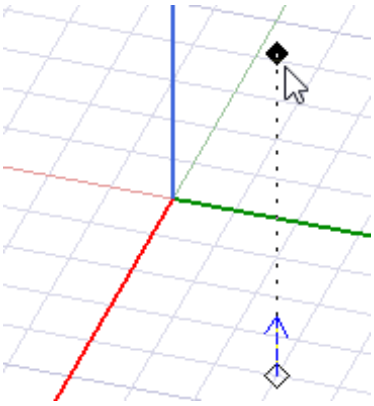


The following additional method is available only after clicking a drawing command:

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



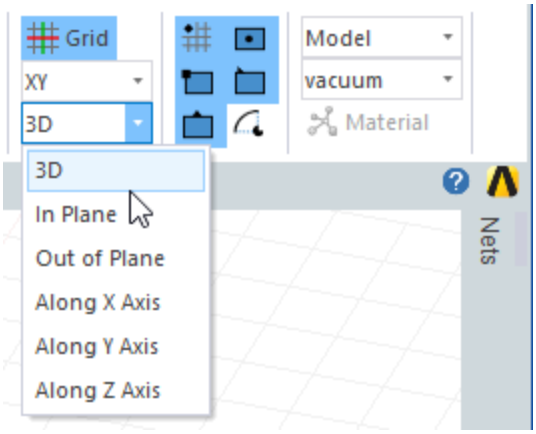
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

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:



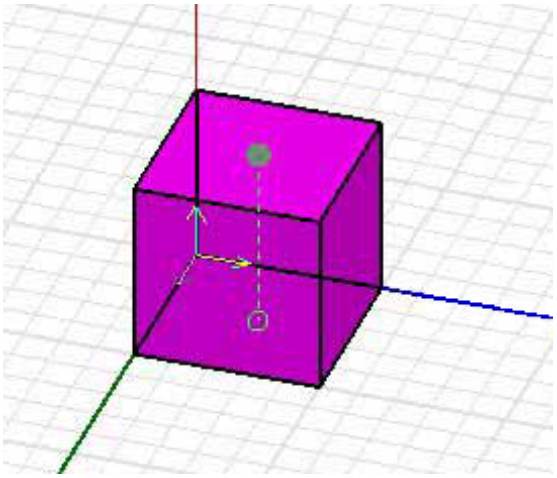
The following additional method is available only after clicking a drawing command:

- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:



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

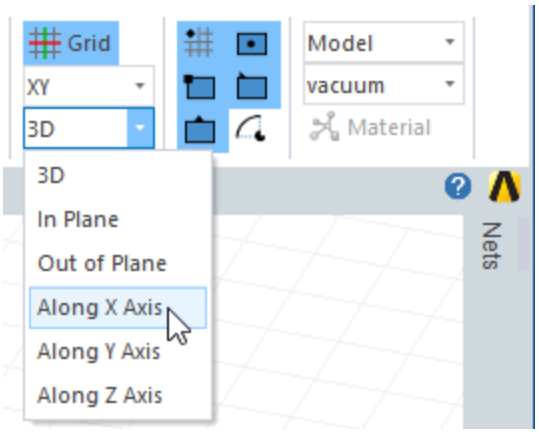
Use this movement mode to constrain the cursor to a moving only in the +/- X direction relative to the reference point. Movement is constrained to the X direction as defined by the working coordinate system, not necessarily the global X direction. The line of motion passes through the reference point, 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu:



The following additional methods are available only after clicking a drawing command:

- 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

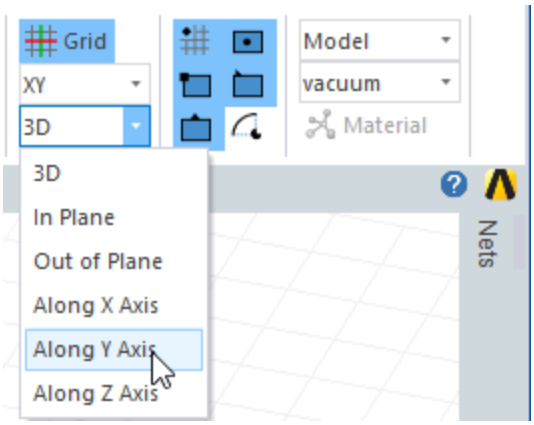
Use this movement mode to constrain the cursor to a moving only in the +/- Y direction relative to the reference point. Movement is constrained to the Y direction as defined by the working coordinate system, not necessarily the global Y direction. The line of motion passes through the reference point, 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Moving the Cursor Along the Z-Axis

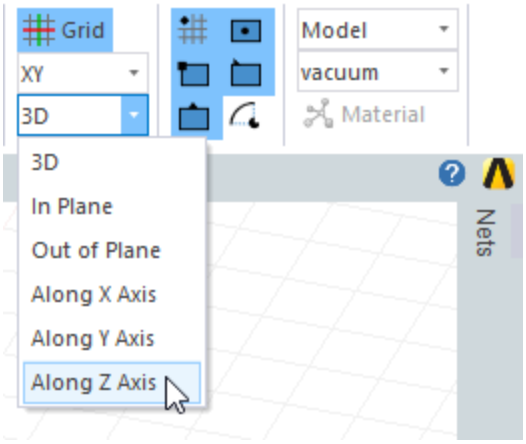
Use this movement mode to constrain the cursor to a moving only in the +/- Z direction relative to the reference point. Movement is constrained to the Z direction as defined by the working coordinate system, not necessarily the global Z direction. The line of motion passes through the reference point, 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

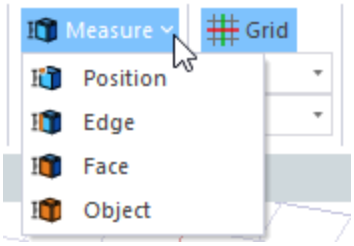
Measure Modes

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 dynamic tracking in order to copy text information from the *Measure Data* window, and then Resume dynamic updating.

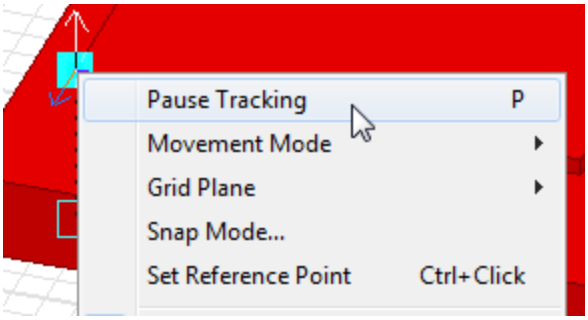
1. To access a Measure mode, do one of the following:
   - From the menu bar, click **Modeler** > **Measure** > [**Position, Edge, Face,** or **Object**].
   - Right-click in the Modeler window and choose **Measure** > [**Position, Edge, Face,** or **Object**] from the short-cut menu.
   - On the **Draw** ribbon tab, choose **Position, Edge, Face,** or **Object** from the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Measure drop-down menu.



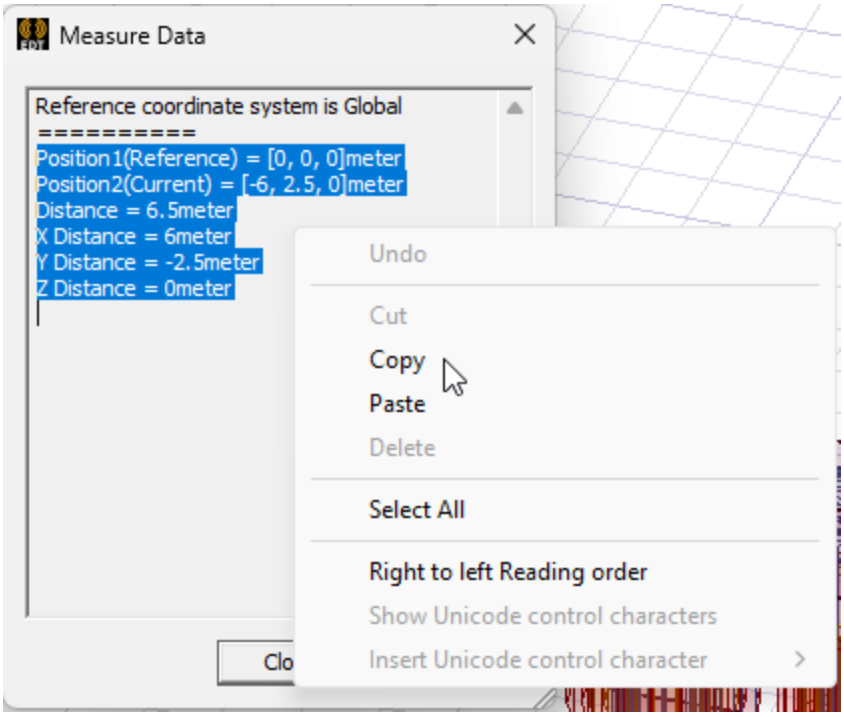
2. Select items to measure as described below under Measurement Modes and Data.
3. Optionally, to *Pause* the dynamic tracking, do one of the following:
   - Right-click in the Modeler window and select **Pause Tracking**:



   - 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 2026 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.

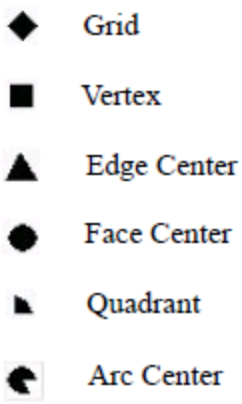
4. To exit the Measure mode, do one of the following:
   - Click **Close** on the *Measure Data* window.
   - Press **Esc**.

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 2026 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 2026 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

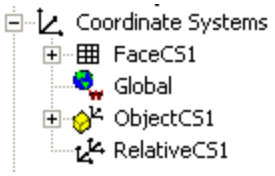
Setting Coordinate Systems

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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Working Coordinate System

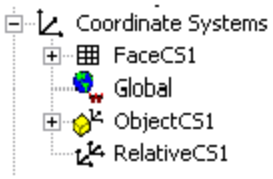
The working coordinate system (CS) is the current CS with which objects being drawn are associated. The working CS can be the Global CS or a user-defined Relative, Object, or Face CS. Select the working CS by clicking its name in the History Tree, or follow this procedure:

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:

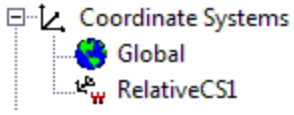
- You can create an offset relative CS, that is, a relative CS whose origin lies a specified distance from another CS's origin. By moving a CS's origin, you can enter coordinates relative to an existing object, without having to add or subtract the existing object's global coordinates.
- You can create 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 2026 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:

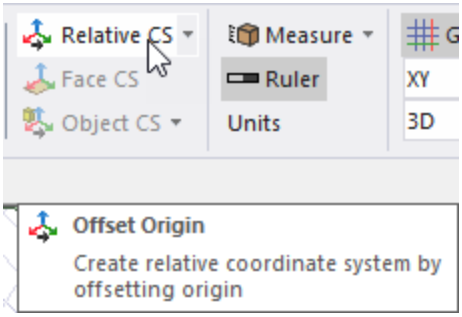


Selecting the CS in the History Tree causes the docked *Properties* window to show the CS properties.

Creating an Offset Relative CS

To create a relative CS with an origin that lies a specified distance from another CS's origin:

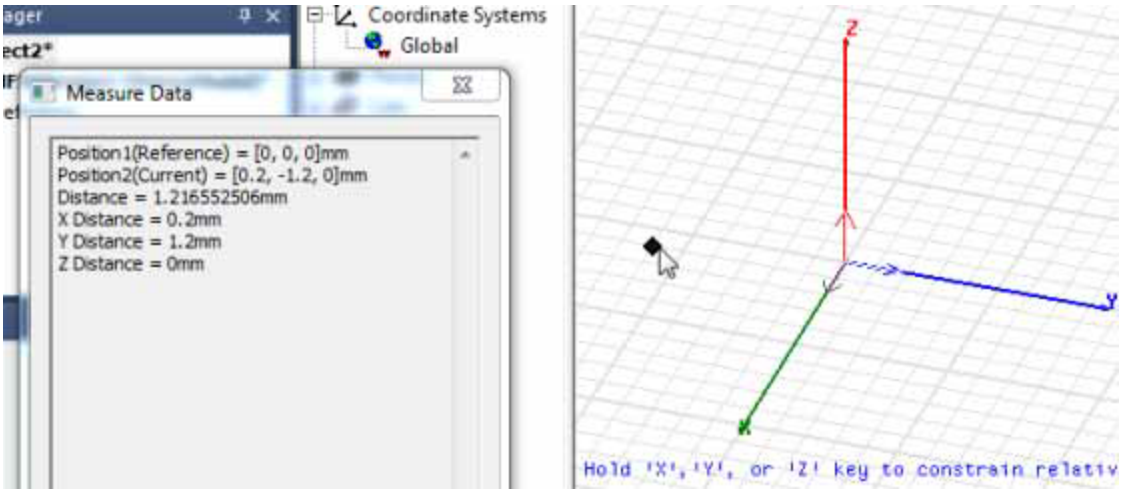
1. In the History Tree, click the CS upon which you want to base the new relative CS, making it the working CS.
2. From the menu bar, click **Modeler** > **Coordinate System** > **Create** > **Relative CS** > **Offset** or, on the **Draw** ribbon tab, click **Relative CS**. (You do not have to access 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):

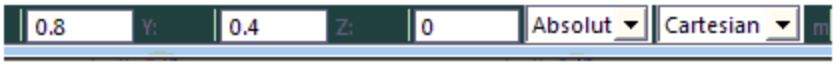
          Use the cursor to click the point. You can see the coordinate information in the *Measure Data* window.

Ansys Electromagnetics Suite 2026 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.

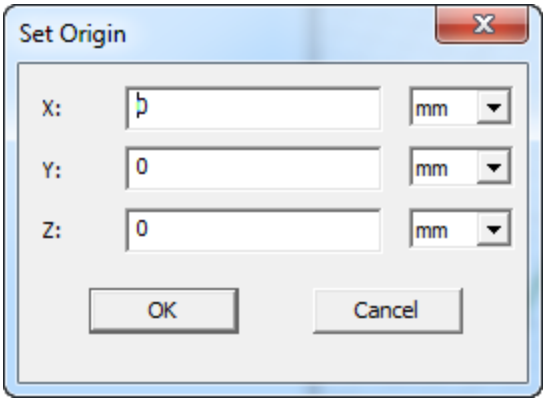
- Specify the coordinates numerically (using the coordinate entry text boxes in the Status Bar):



  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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

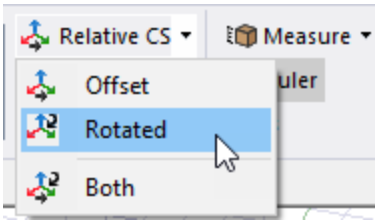
The new relative CS is created. Its origin has moved from the previous working CS, but its axes remain the same. It is listed in the History Tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.



Creating a Rotated Relative CS

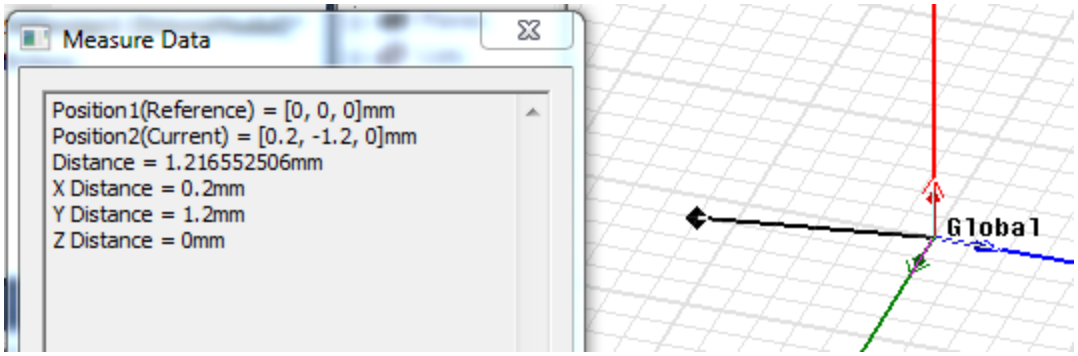
To create a new relative CS with its axes rotated away from another CS's axes:

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**:



.

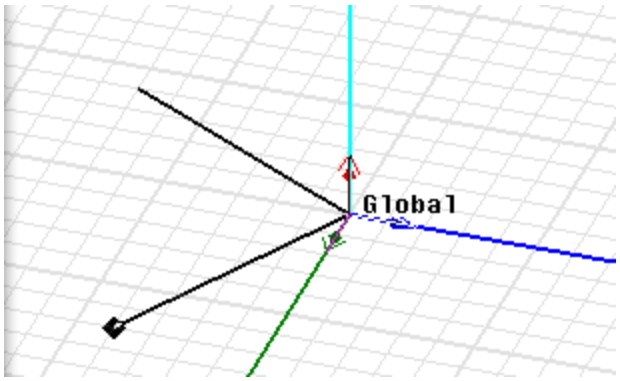
3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools** > **Options** > **General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the axes in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – 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.



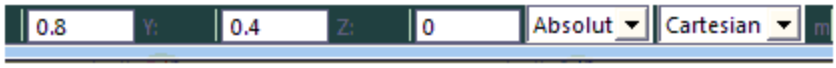
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

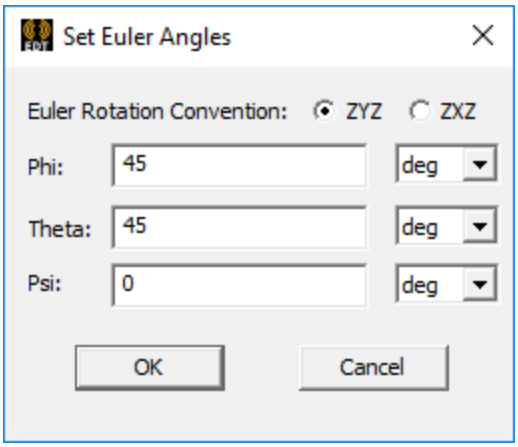
- **Euler angle** option selected, or **F4** pressed:

a. In the *Set Euler Angles* dialog box, select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).

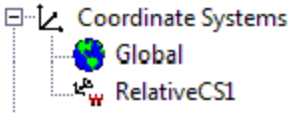
b. Specify the **Phi, Theta,** and **Psi** values and select units for each from the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down menus.



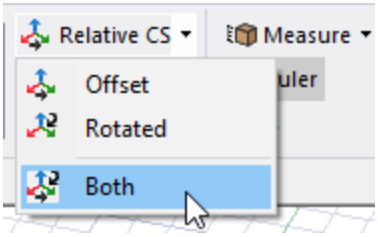
The new relative CS is created. It has the same origin as the previous working CS, but its axes are rotated. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default 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:

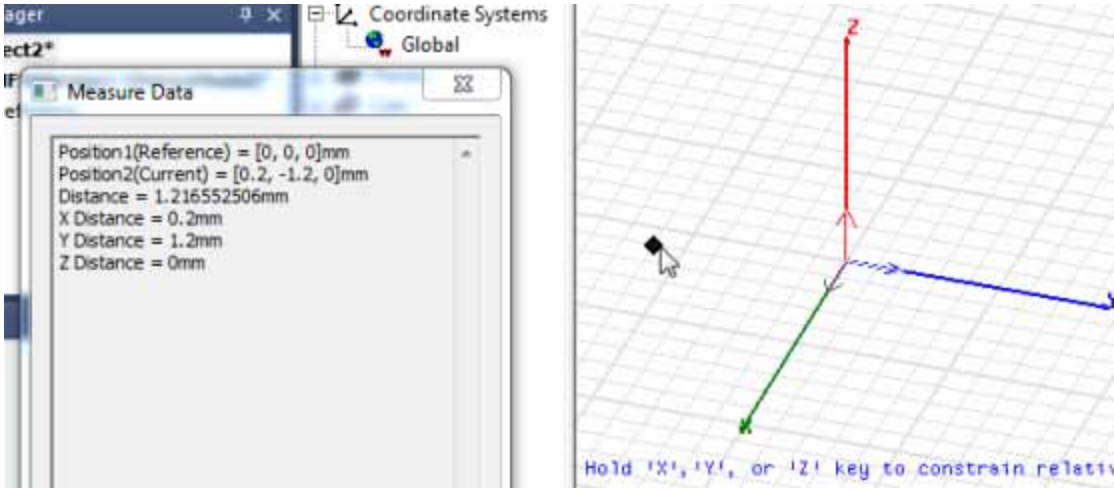
1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler** > **Coordinate System** > **Create** > **Relative CS** > **Both** or, on the **Draw** ribbon tab, click **Relative CS** > **Both**:



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:

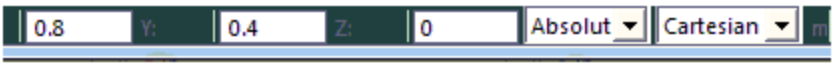
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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:

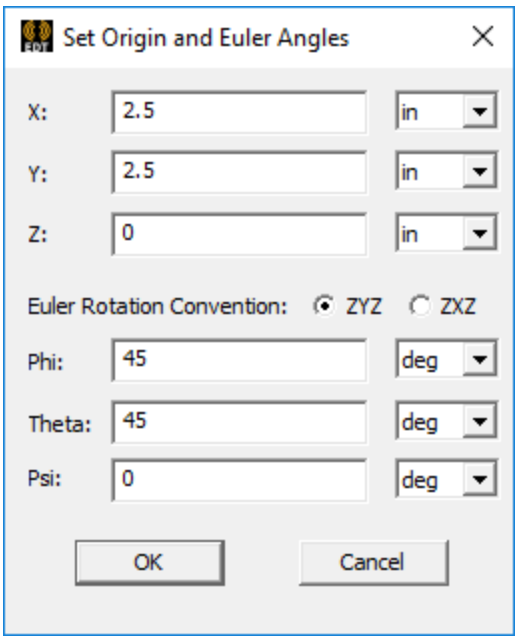


    a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
    b. Select either Relative or Absolute coordinates from the other drop-down menu.
    c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** 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 2026 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.



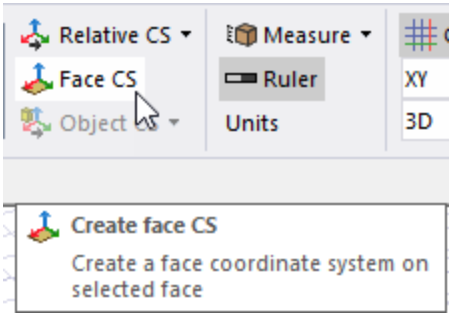
The new relative CS is created. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.



Creating a Face Coordinate System

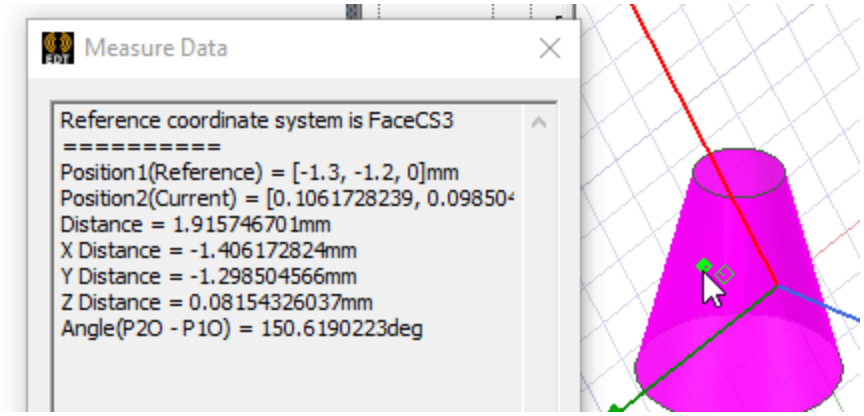
1. Select the object face upon which you want to create the face CS. The CS face can be planar or curved.
2. Click **Modeler** > **Coordinate System** > **Create** > **Face CS** or, on the **Draw** ribbon tab, click the **Create Face CS** icon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



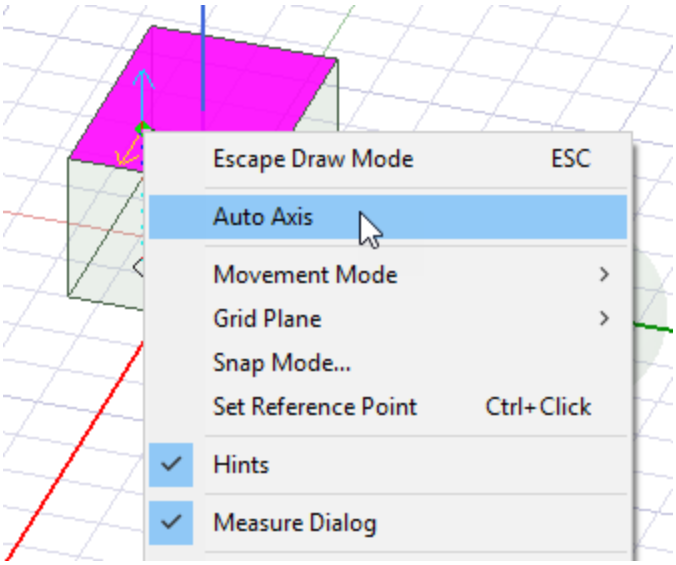
3. Select the Origin in one of the following ways:
   - Click a point on the selected face to define the Origin. If the point is not on the face, you will receive an error message.

     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.

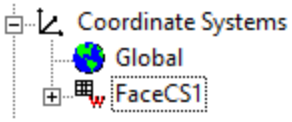
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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 2026 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**.

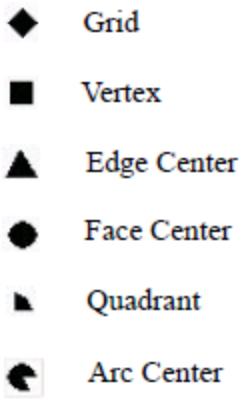
Now, when you select a face, and then click a drawing command, a new face CS will be created on the selected face. The modeler automatically sets the new face CS as the working CS, and the object you draw is oriented according to the new face CS.

> **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

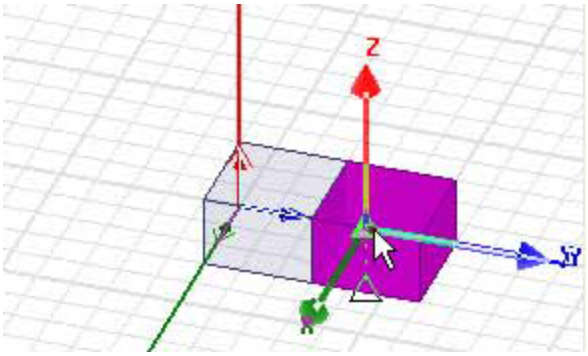
You can create coordinate systems based on any object of solid, sheet, or wire type. The **Modeler** > **Coordinate System** > **Create** > **Object CS** command, and the equivalent ribbon command, is enabled when you select an object. An Object CS can be Offset, Rotated or Both. Executing one of the 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 2026 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.



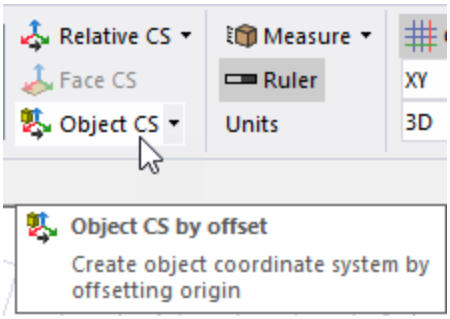
Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS, and in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. Also see the Move CS to End command.

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

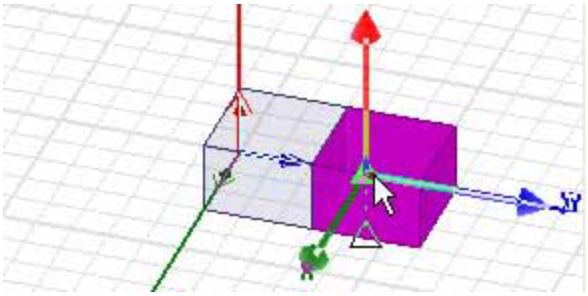
To create an Offset Object coordinate system (CS):

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 2026 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.

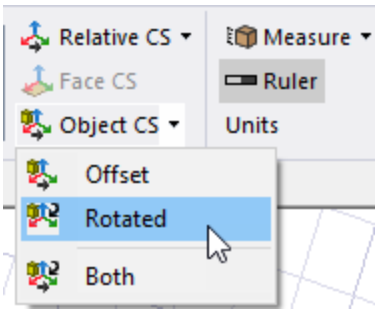


    - The point must be on the selected object.
    - When you select the Origin point, the location is validated, and an appropriate message indicates when a point is invalid. Points where a CS preview appears are always valid.
    - The X-axis {1,0,0}, Y-axis {0,1,0} and Z-axis {0,0,1} are parallel to 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.

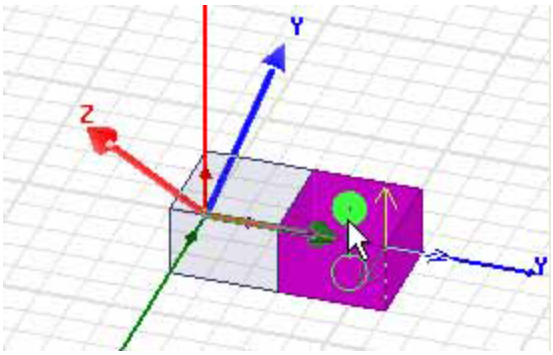
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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).

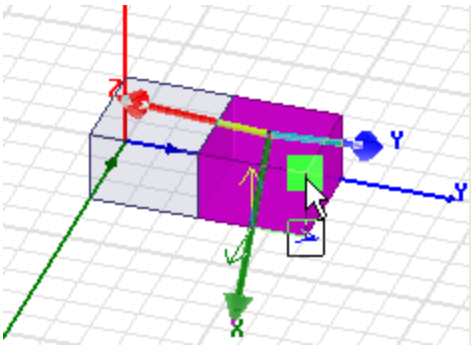
3. Select a point on the object to define the Origin.

   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 2026 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.

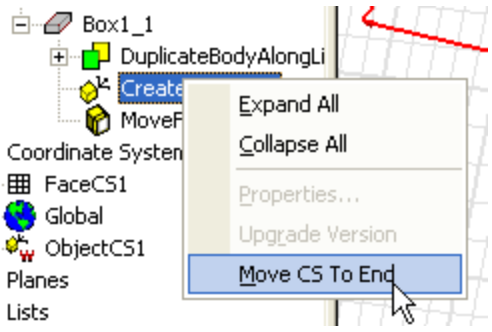
Move CS to End Command for History Tree

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 2026 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).

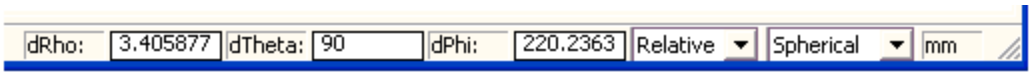
If you want to also modify the whether the coordinate system is Absolute or Relative, and to change how your express the coordinates, do the following:

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

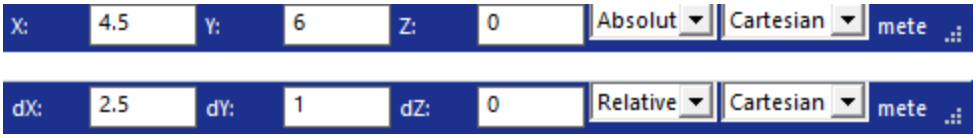
An Object CS can be edited in the same way as it was created. For example, if the CS was created in Offset mode, it will be edited in Offset mode only.

- 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*.

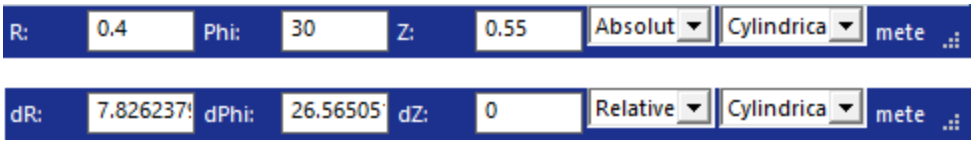
Expressing Cartesian, Cylindrical, or Spherical Coordinates

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.

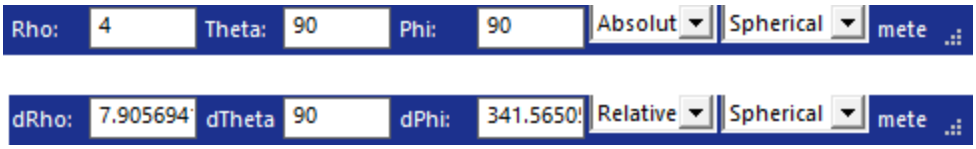


- **Cylindrical:** Specify the point's radius, measured from the origin, is the **R** text box, the angle from the x-axis in the **Theta** text box, and the z-distance from the origin in the **Z** text box.



Ansys Electromagnetics Suite 2026 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**.

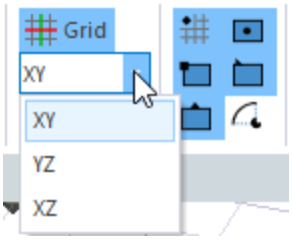
   The CS will be deleted and, all objects drawn on it will be deleted. Furthermore, any CS that was dependent upon the deleted CS will be deleted, and any objects that were drawn on the dependent CS will also be deleted.

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:

- On the **Draw** ribbon tab, select **XY, YZ,** or **XZ** from the **Drawing plane** drop-down menu:

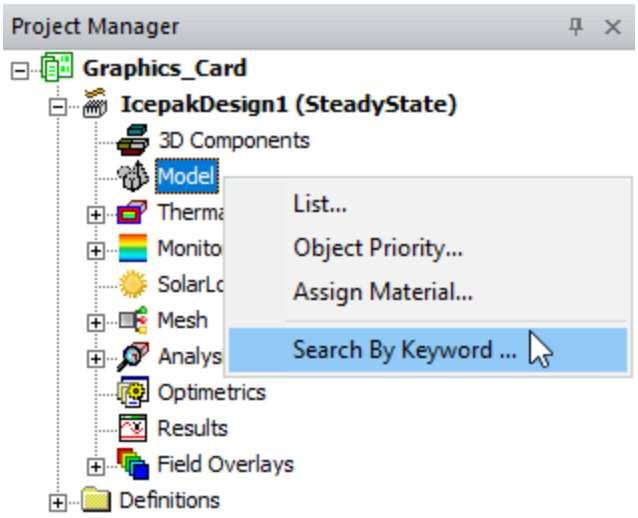


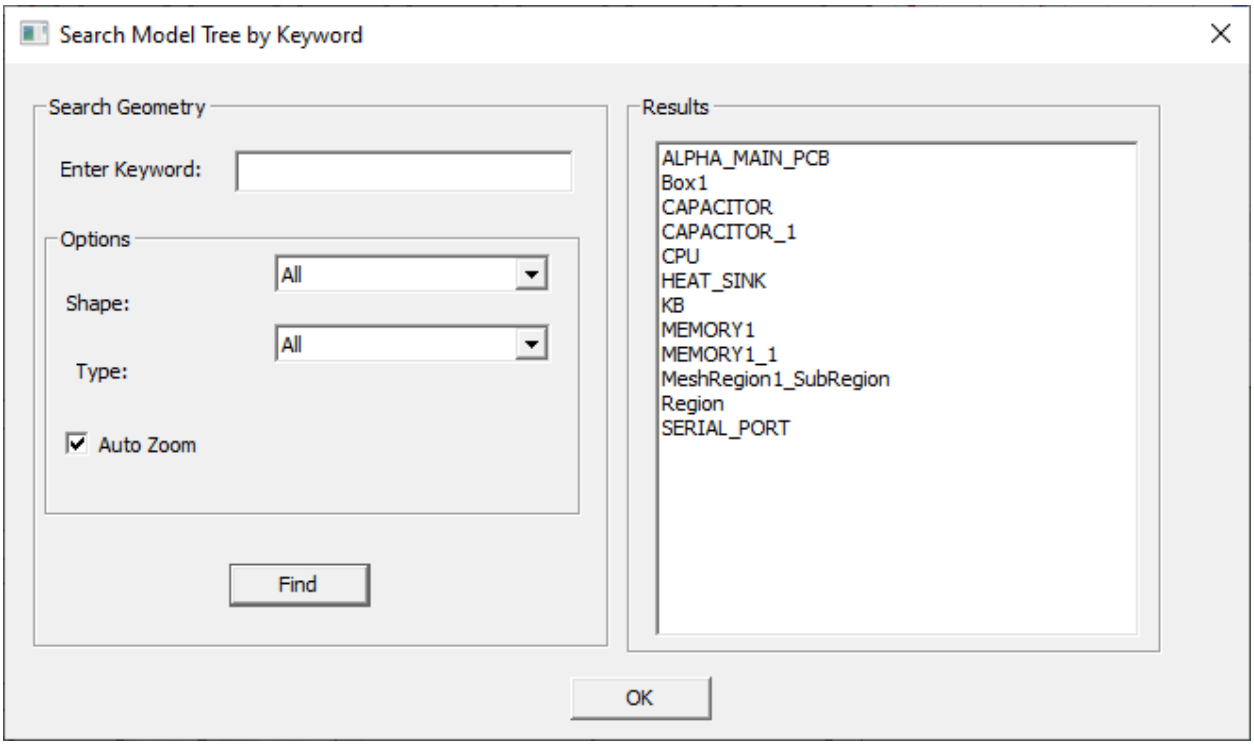
- From the menu bar, click **Modeler** > **Grid Plane** > and select one of the planes **XY, YZ,** or **XZ**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Searching for Model Geometry

In Icepak designs with many objects, you can use the **Search Model Tree by Keyword** dialog box to quickly filter geometry and select the desired geometry in the 3D Modeler window. In the **Project Manager** select **Search By Keyword** from the Model module's right-click menu.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Search Geometry	
Enter Keyword	Type letters or words in the **Enter Keyword** to filter by the specified phrase.
Options	
Shape	Select an option from the **Shape** drop-down list to narrow the results by shape.
Type	Select a **Model** or **Non-Model** from the **Type** drop-down list to narrow the results by type.
Auto Zoom	Enable **Auto Zoom** to zoom in on geometry when selected from the **Results** list.
Find	Click **Find** to filter the **Results** list based on the specified keyword and selected options.
Results	
Results	The **Results** list displays the geometry matching the specified keyword and selected options.
OK	Click **OK** to close the **Search Model Tree by Keyword** dialog box.

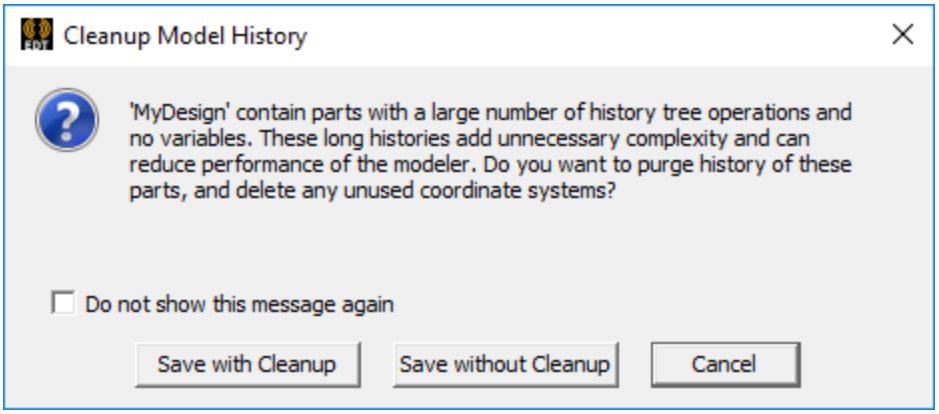
Cleaning Up Model History

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



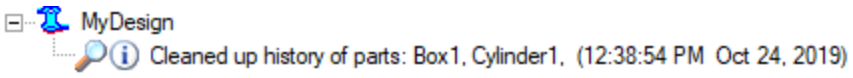
The number of actions that triggers this prompt can be set in 3D Modeler Operations Options.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

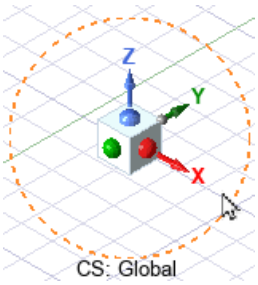
6 - Modifying the 3D Model View

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.

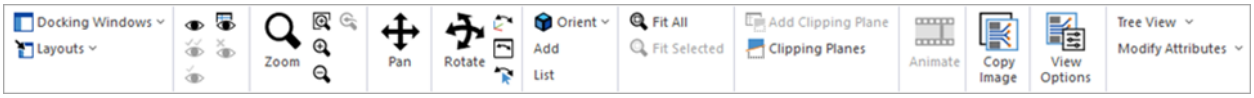
What do you want to do?

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
- Set the View Options

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rotate the View
- Pan the View
- Zoom In or Out
- Zoom to Selected Excitation
- Zoom In or Out Using a Mouse Wheel
- Fit Contents in the View Window
- Use Clip Planes

Show or Hide Design Objects

- Show or Hide Objects
- Showing Only Selected Objects in All or Active Views

You can also access the Show or Hide features by using the following icons on the **View, Draw,** or **Model** ribbon tabs:



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

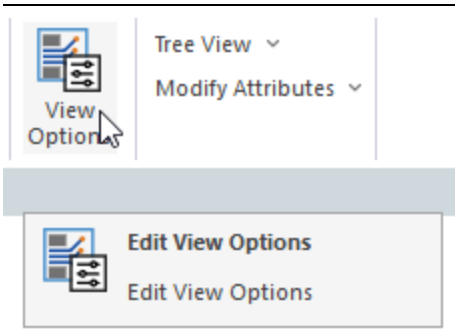
Change the Background

- Setting the Background Color
- Modify the appearance of the Coordinate System Axes
- Modify the appearance of the Grid

View Options: 3D UI Options

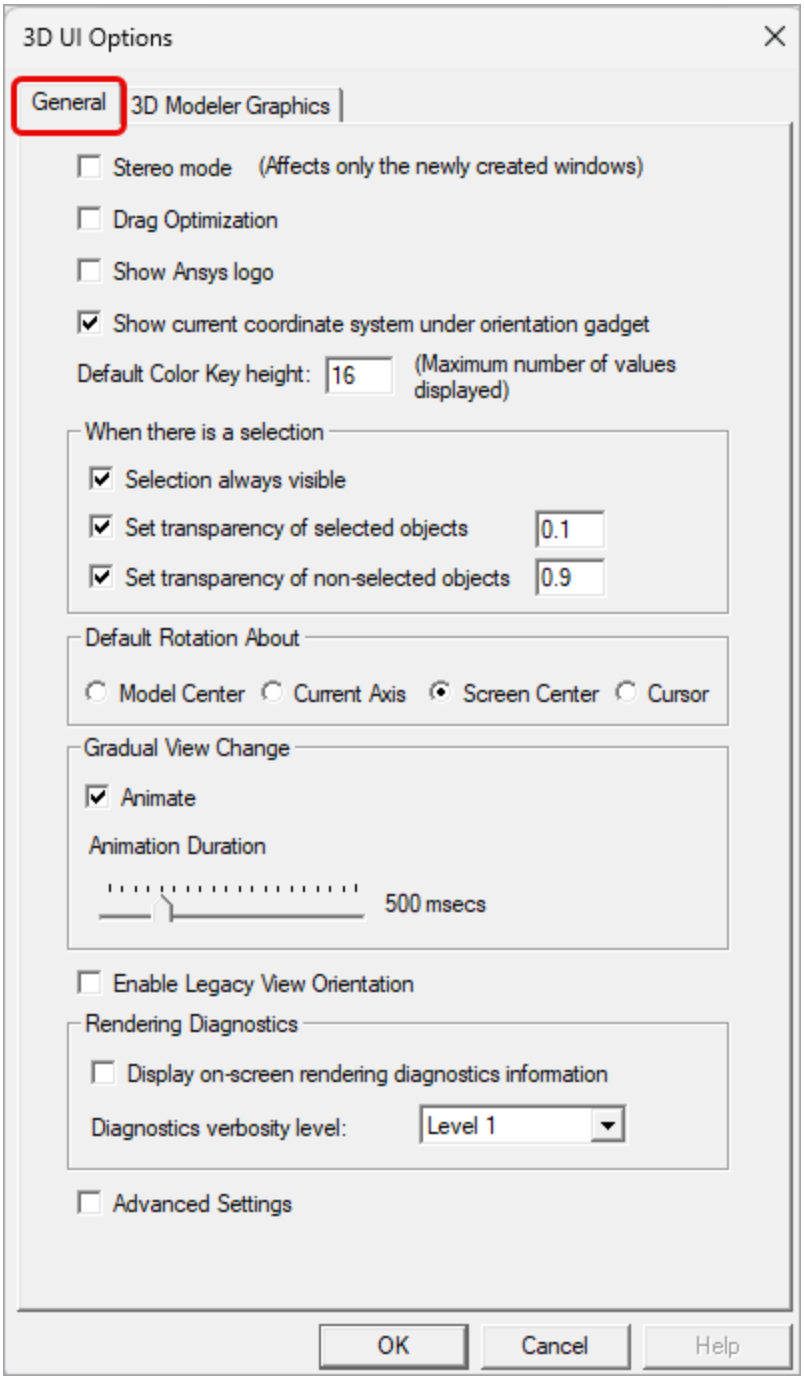
To access View Options, use the **View** > **Options** command, or click **View Options** on the **View** ribbon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **3D UI Options** window appears.

Ansys Electromagnetics Suite 2026 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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

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

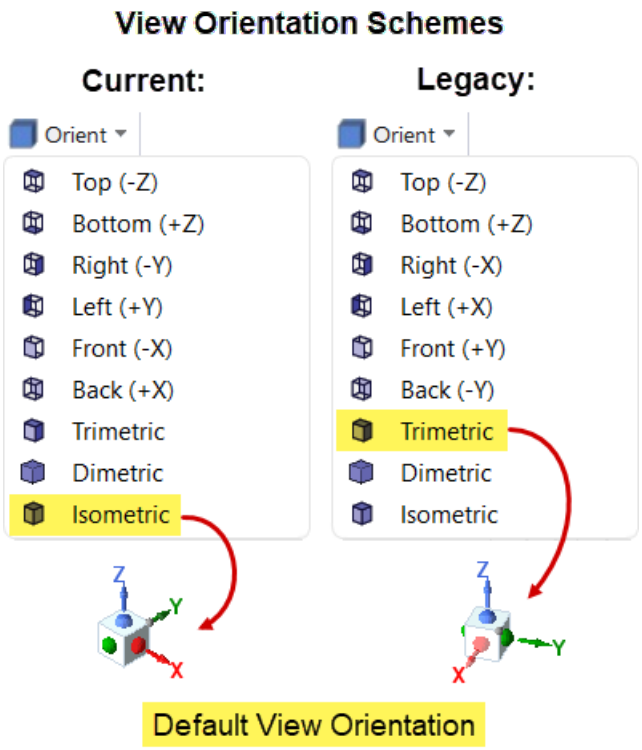
  For operations such as Fit Selection, Fit All, and Zoom, these settings determine whether Electronics Desktop animates the transition between images and how smoothly it does so.

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 2026 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:



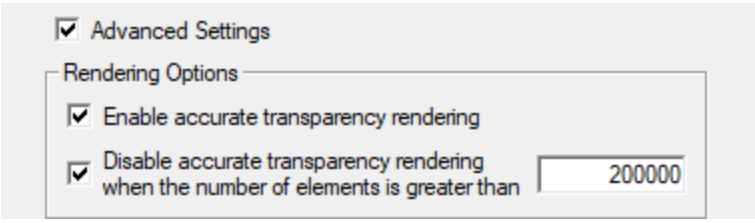
Rendering Diagnostics Options: .

You can select to display on-screen rendering information, and the diagnostics verbosity level from level 1 to level 4.

Advanced Settings: Rendering Options:

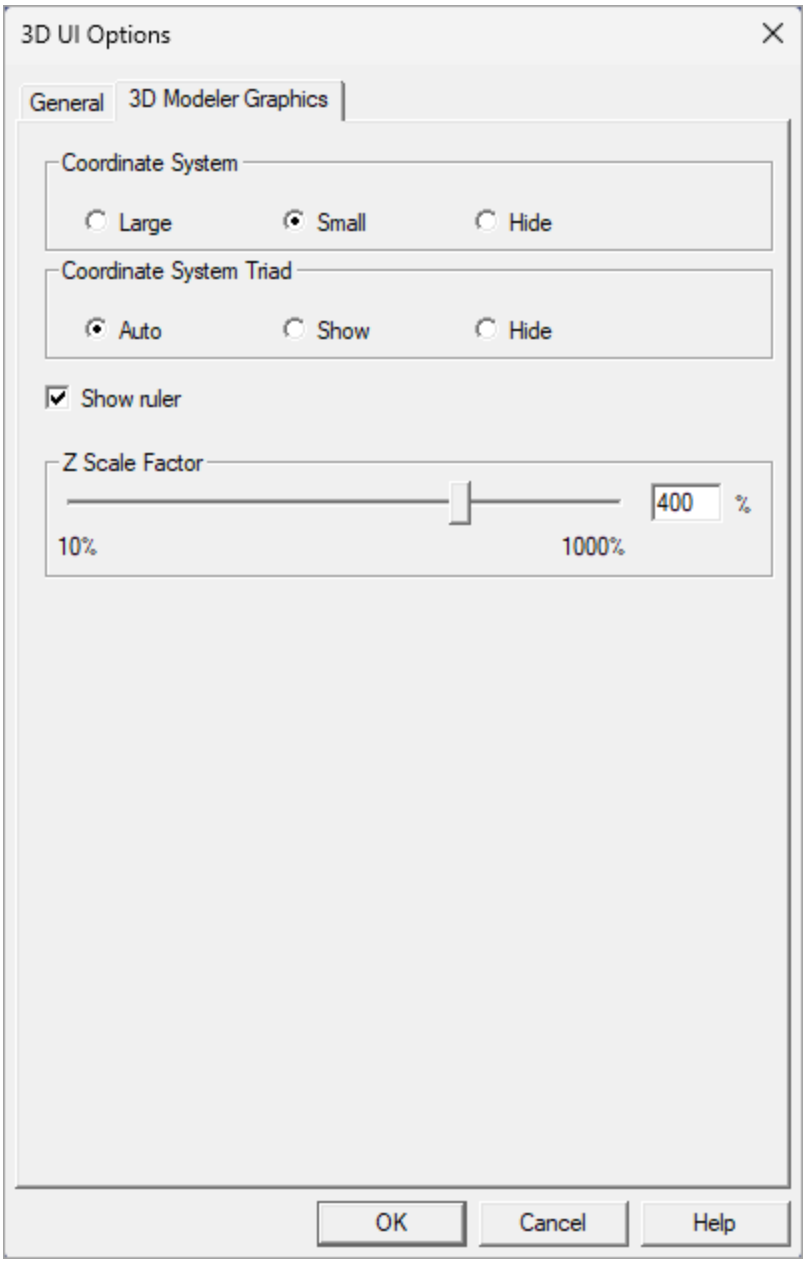
Because rendering transparency correctly can be compute intensive, especially for complex models with thousands of objects, it can bring down the frame rate. In such cases, you can select whether to **Enable accurate transparency rendering**.

You can also choose to **Disable accurate transparency rendering** when the number of elements is greater than a specified number. The default is 200000.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Graphics Tab



The **3D Modeler Graphics** tab lets you set the following options which are invoked for the next Project and design created.

Ansys Electromagnetics Suite 2026 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:

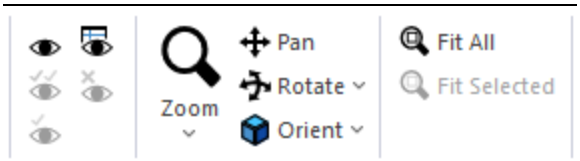
1. Click **View** > **Interaction** >  **Pan** or click the **Pan** icon on the **View, Draw,** or **Model** ribbon tab.

   View ribbon tab:

   Draw or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2026 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 2026 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.

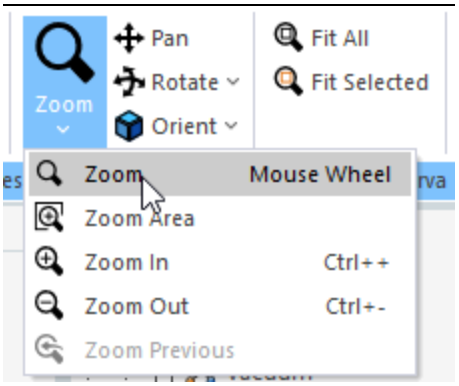
To zoom by dragging the mouse.

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 2026 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

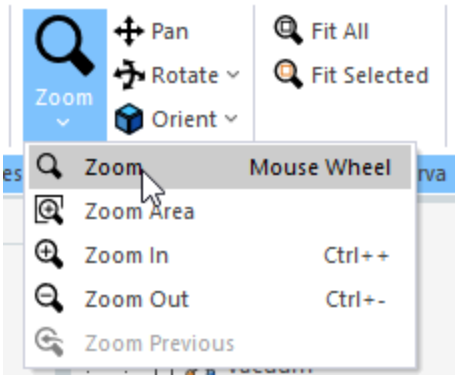
To use **Zoom In**, **Zoom Out** to zoom in or out in 5% increments:

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.

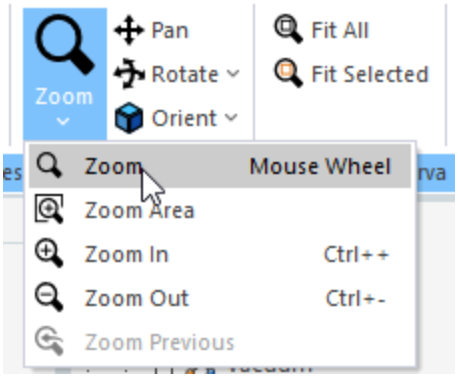
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

Ansys Electromagnetics Suite 2026 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**.

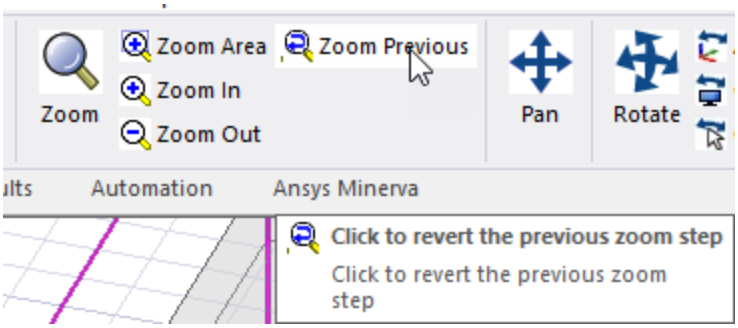
Zoom Previous Command

The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.



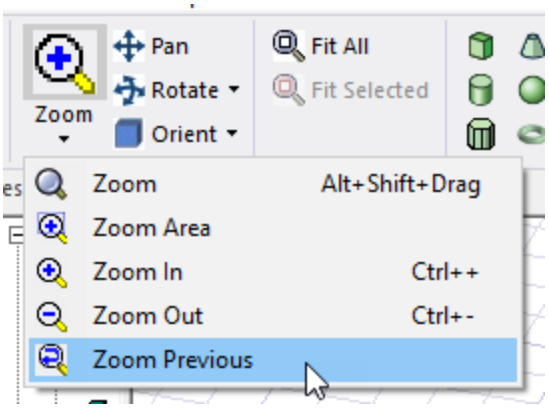
It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

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.



Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom Previous**:

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

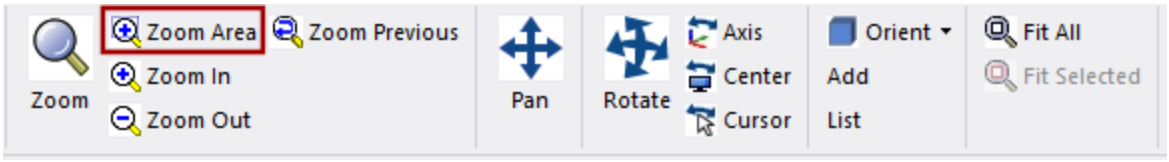
If zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

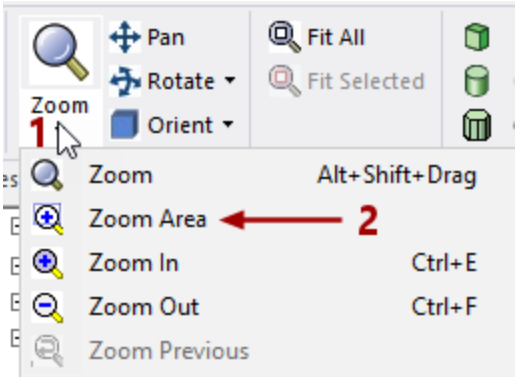
Zooming into a Rectangular Area

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 2026 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**.

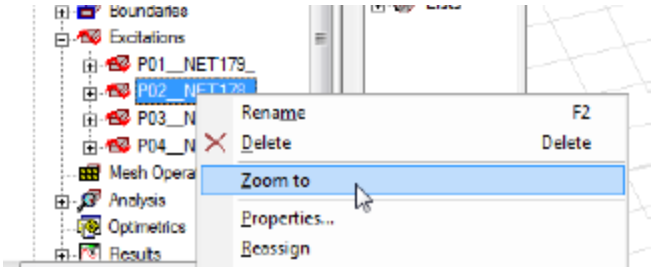
Zoom to a Selected Boundary or Excitation

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.

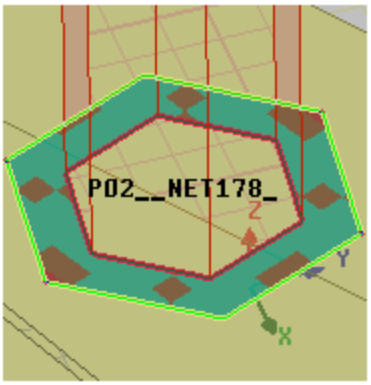
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

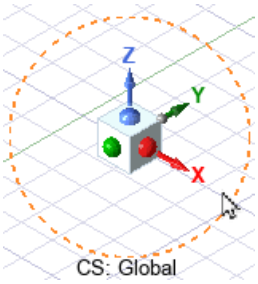


Modifying the View Orientation

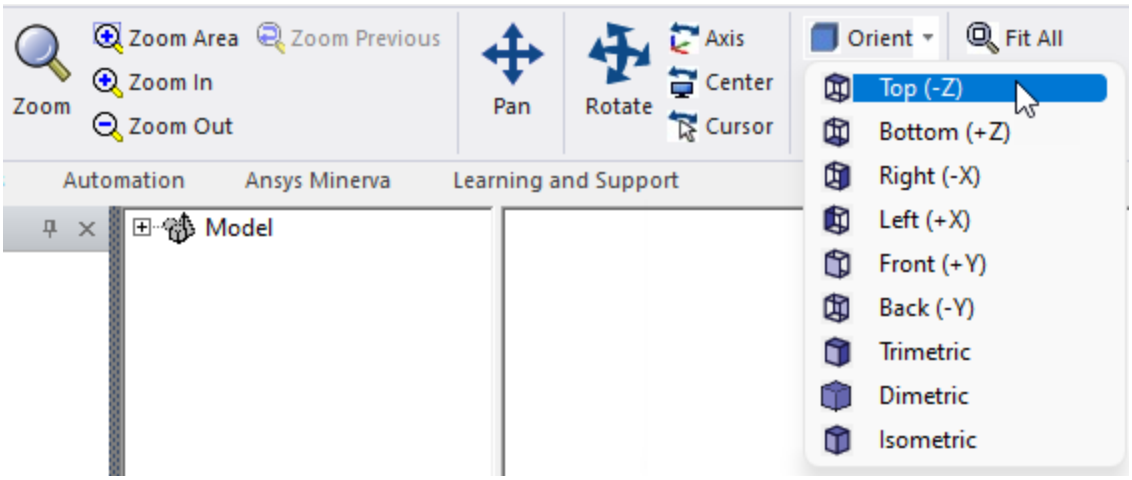
There are several places you can access commands to change the view orientation in a *Modeler* or *3D Plot* window:

Ansys Electromagnetics Suite 2026 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 2026 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.

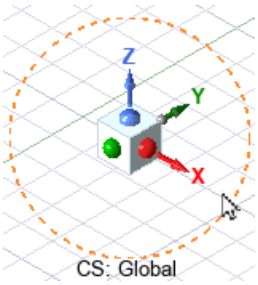
1. Select a default orientation or create a new orientation.
2. Click **Apply to View** for the selected view to appear in the window.
3. Optionally, click the **Reset View Orientation** button to restore the selected view to the original angles.
4. Click **Make Default** if you want the selected viewing 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

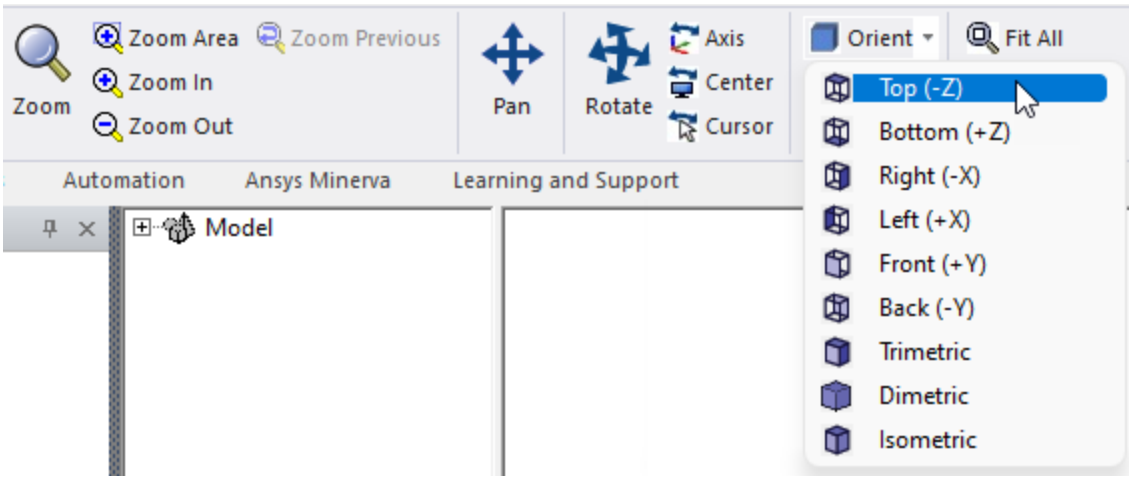
Apply view orientation settings using any one of several available methods:

- Use the Orientation Gadget:

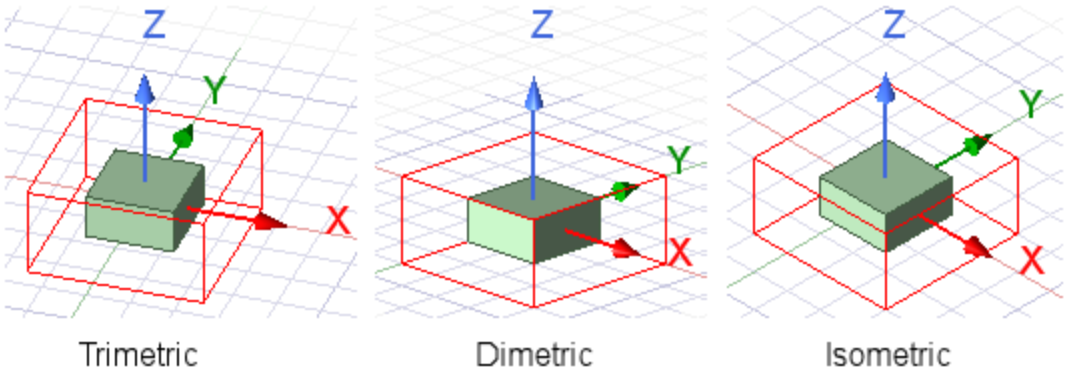


- Use the commands in the **Orient** drop-down menu located on the **View**, **Draw**, or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

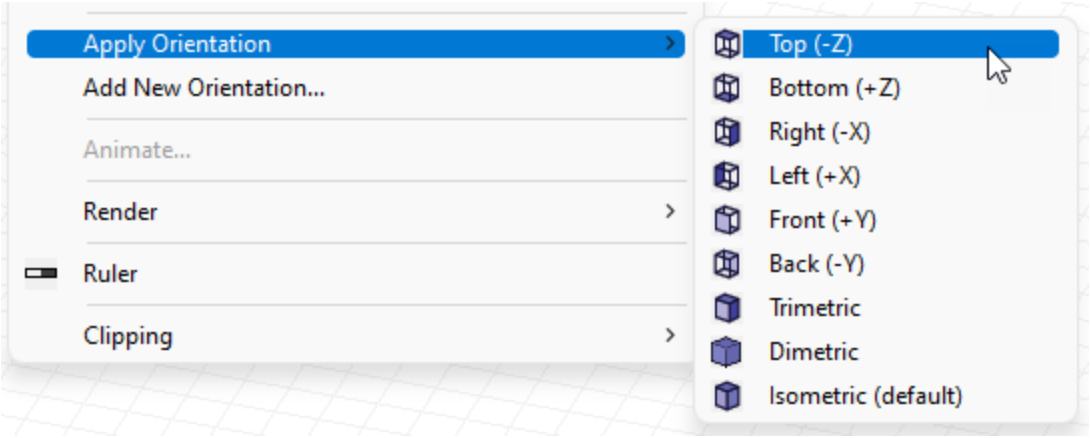


Possible orientations are Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric. These last three differ as shown below:



- Right-click in the Modeler window. In the shortcut menu that appears, go to the **View** > **Apply Orientation** submenu:

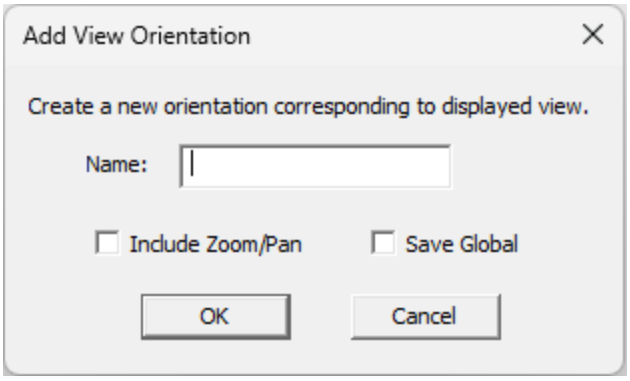
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



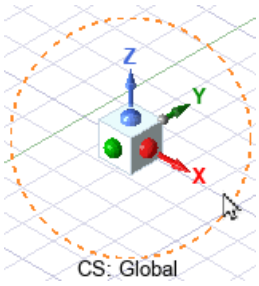
The **Name** that you assign will appear in the **View** > **Apply Orientation** submenu within the Modeler window's shortcut menu. You can specify whether to **Include Zoom/Pan** (in addition to the viewpoint orientation) and whether to **Save Global**, which makes the custom orientation available for 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The active coordinate system is displayed underneath the orientation gadget.



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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Click the small gray dot at the cube's +X, +Y, +Z corner:**

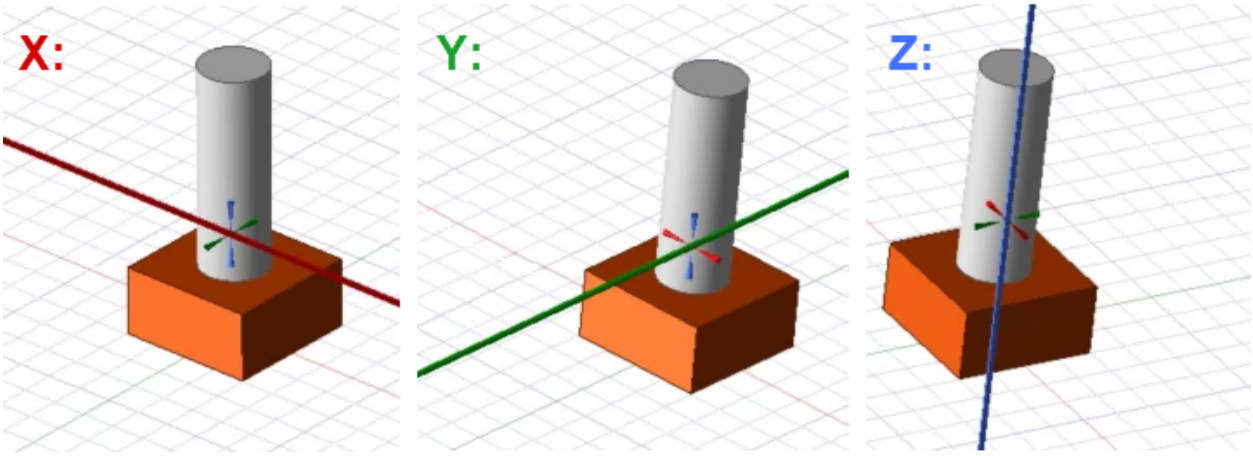
  This action produces the standard *Isometric* view orientation.

- **Click on or very close to the cube (but not on an axis or dot) and drag the mouse:**

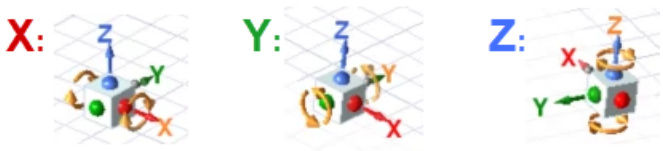
  This action freely rotates the view orientation. The rotation center depends on the **Default Rotation About** option selected in the *3D UI Options* dialog box. The behavior is equivalent to clicking and dragging in the display area using the middle button / wheel but is performed with the left mouse 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 center point 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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

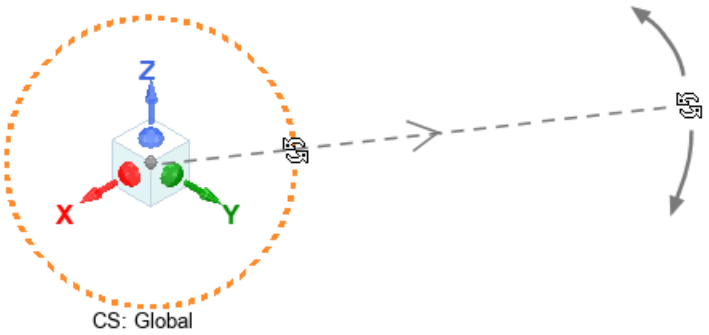
For more information and tips about the usage of this feature, see Rotating or Spinning the View.

- **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:

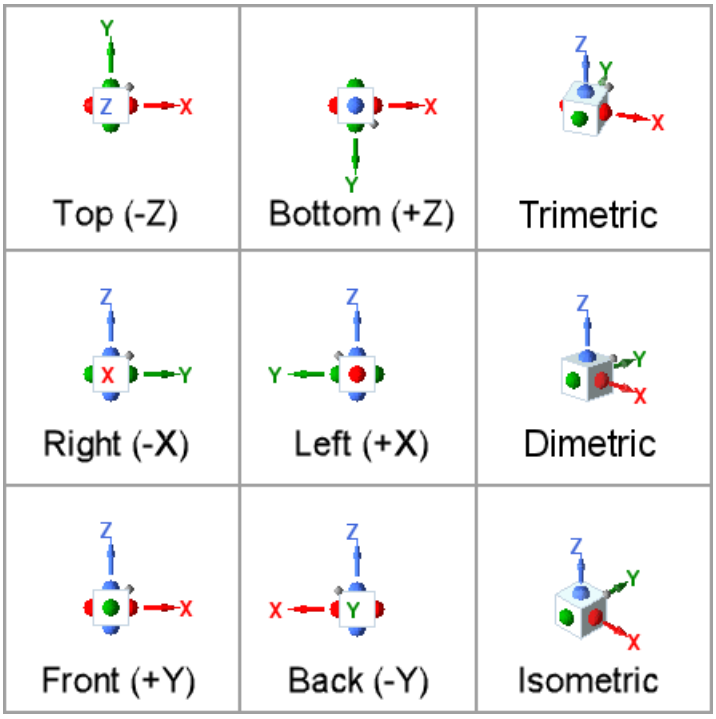
  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl+click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

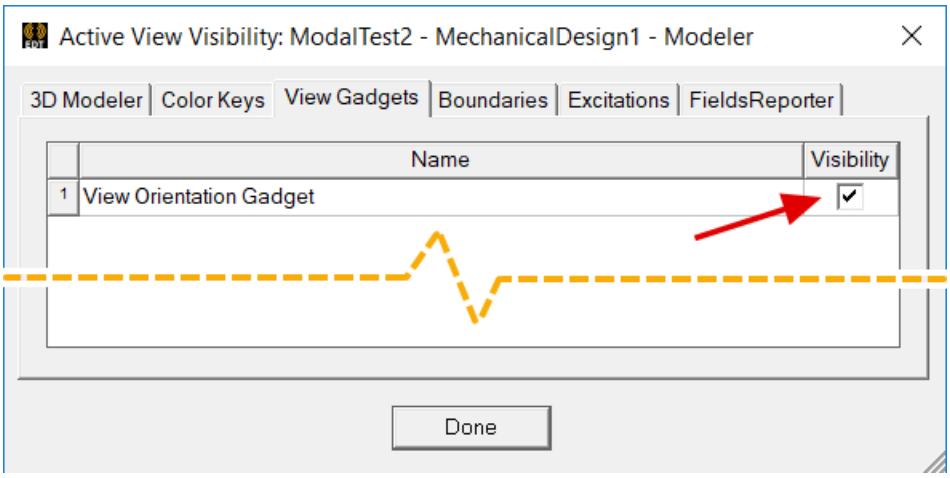
The orientation, colored dots, and axes of the Orientation Gadget correspond to the active coordinate system, whether **Global** or **user-defined**. All standard views are defined in the same manner as they are for the global CS, but Ux, Uy, and Uz directions are used instead of global X, Y, and Z.

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 2026 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:**

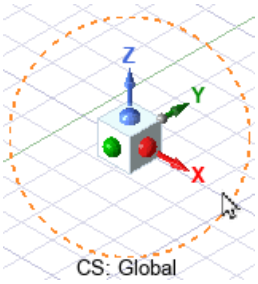
  The view is constrained to rotate about an axis that is perpendicular to the screen. The location of the axis, which is the rotation center line, depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag the cursor within the interior of the display area (not close to a border):**

Ansys Electromagnetics Suite 2026 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:**



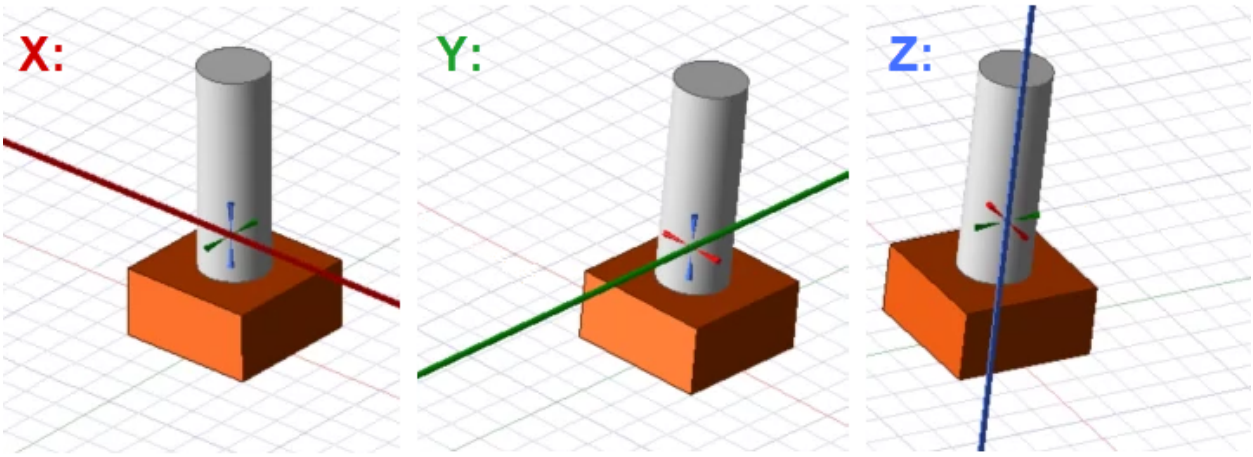
- **Constrained rotation about the X, Y, or Z axis**:

  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** > **...**).

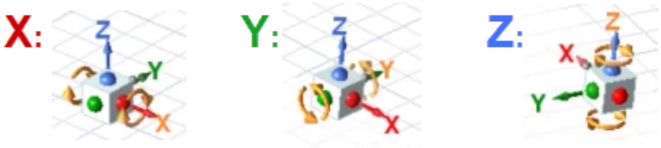
  - **X + click and drag** to rotate the view orientation about the active **X** axis.
  - **Y + click and drag** to rotate the view orientation about the active **Y** axis.
  - **Z + click and drag** to rotate the view orientation about the active **Z** axis.

  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 center point of the drawing canvas. For menu and ribbon commands, you can choose alternative rotation center points (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 2026 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 2026 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 center point. The model orientation will follow the cursor movement, with the angle of rotation matching the angle of the cursor's arc about the same center point.
- 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:**

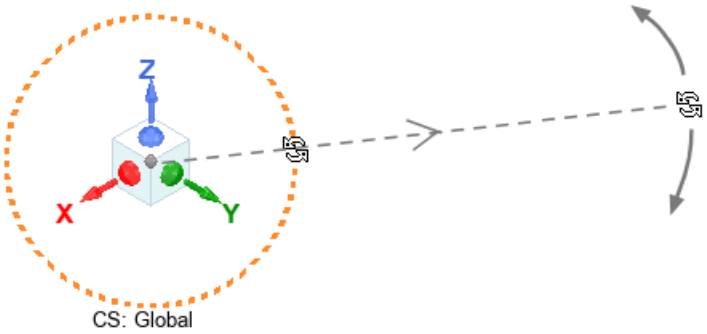
  This orange dashed circle that appears around the view orientation gadget when the cursor is nearby is used to rotate the current view about the screen-normal axis. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the 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 2026 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.



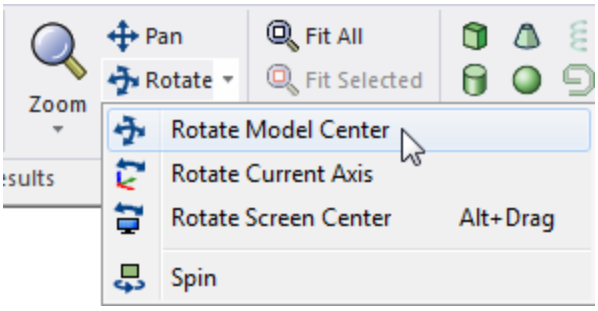
To rotate the view about the **Model Center**:

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 2026 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:

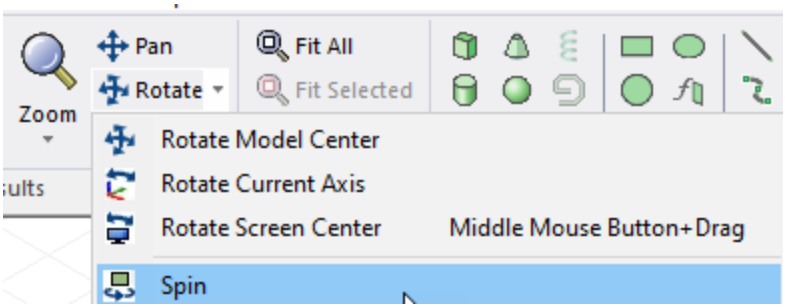
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

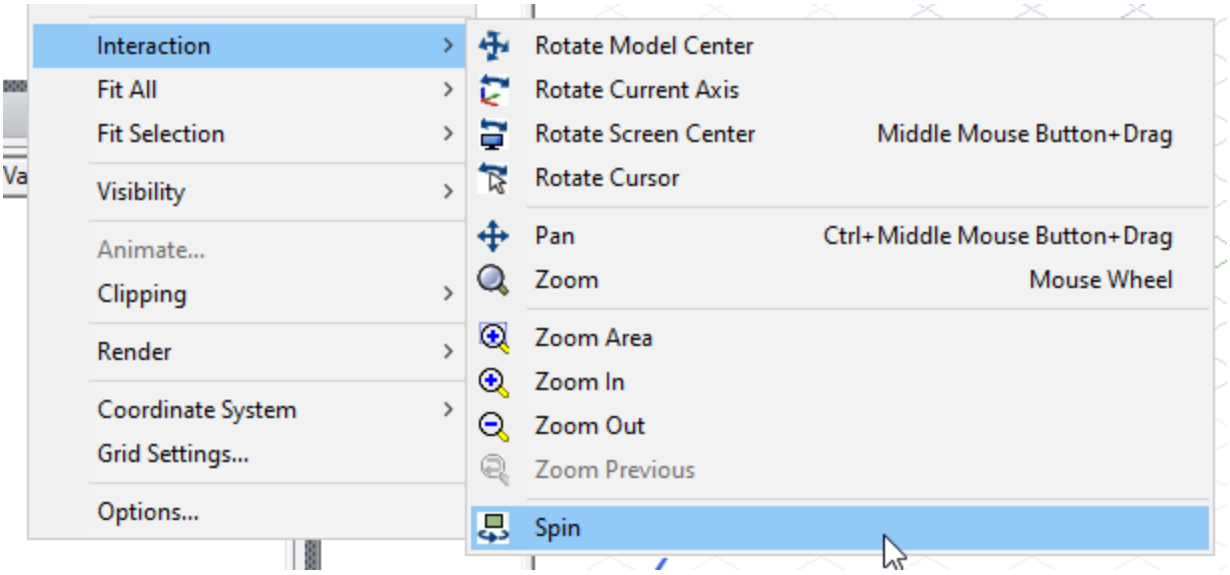
Spin Mode: This tool is useful for capturing animations (for presentation purposes) that show a 360 degree view of the model geometry, mesh, or simulation results.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

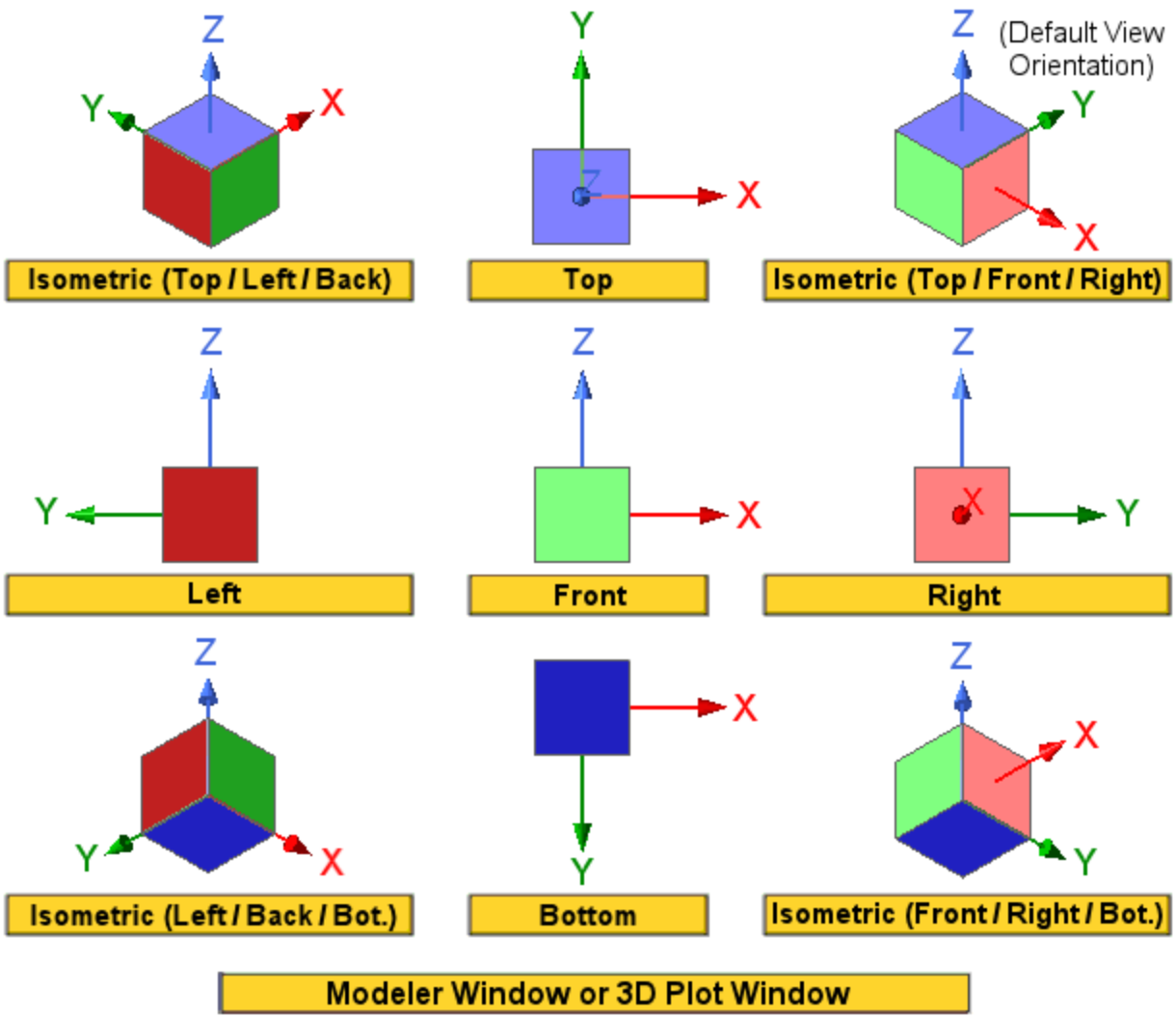
2. Click and drag the mouse in the direction you want to spin the view. Release the mouse button while still dragging.
   - The model will continue to spin in the specified direction. The spinning speed will be proportional to how fast you were dragging the cursor when you released the mouse button.
   - 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 2026 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*).

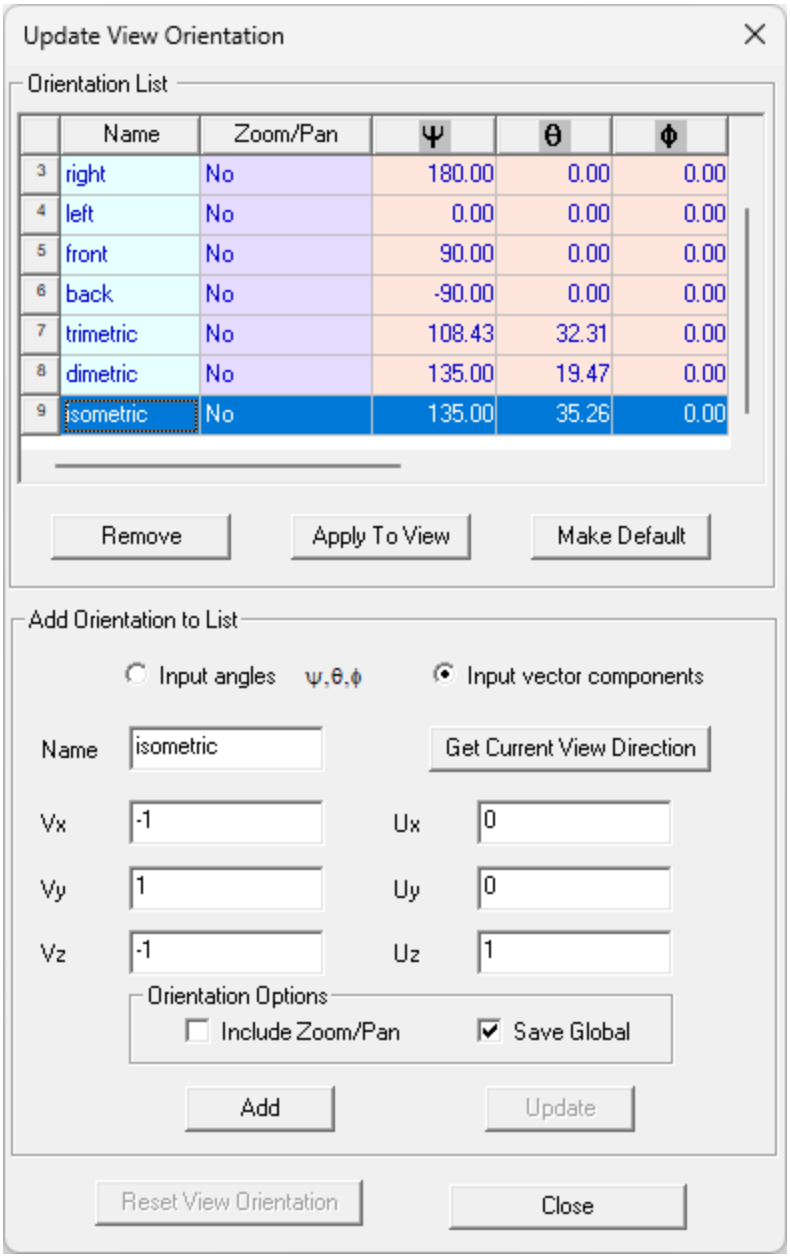
Applying a Default View Orientation

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 2026 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**.

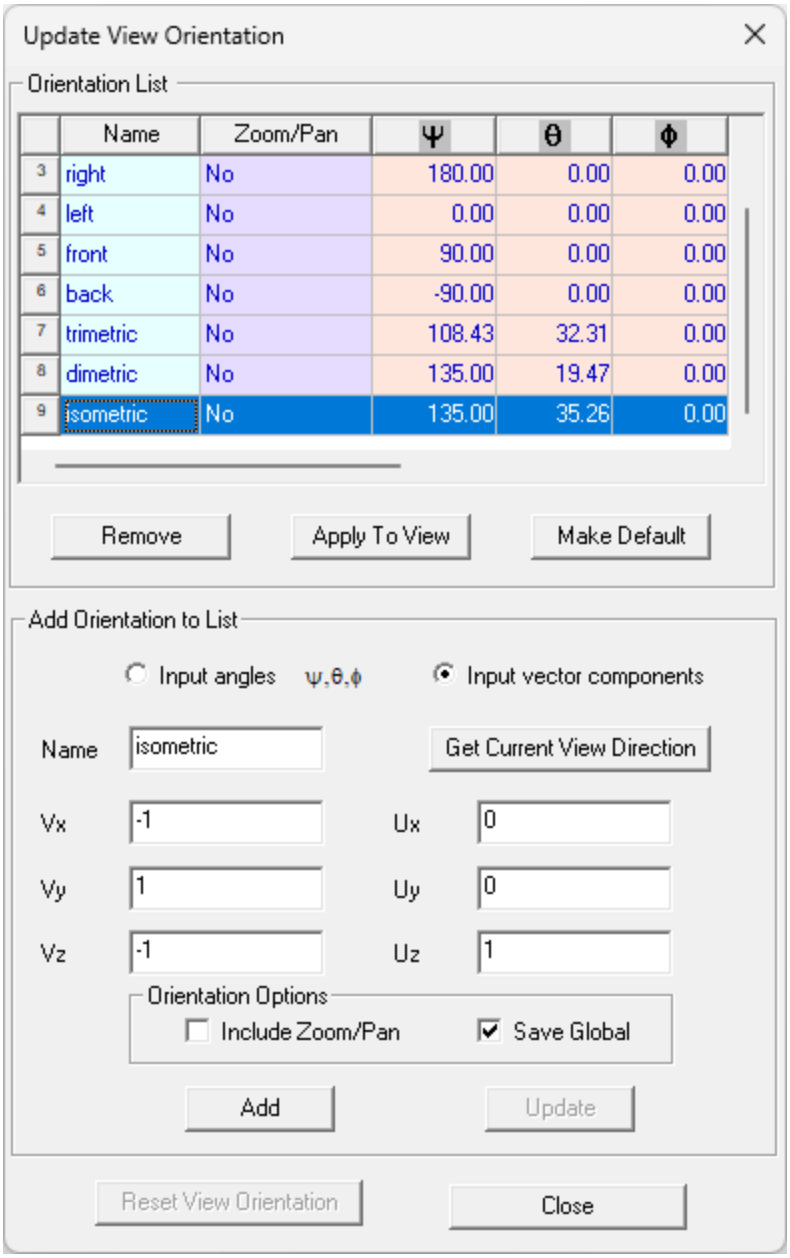
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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**.

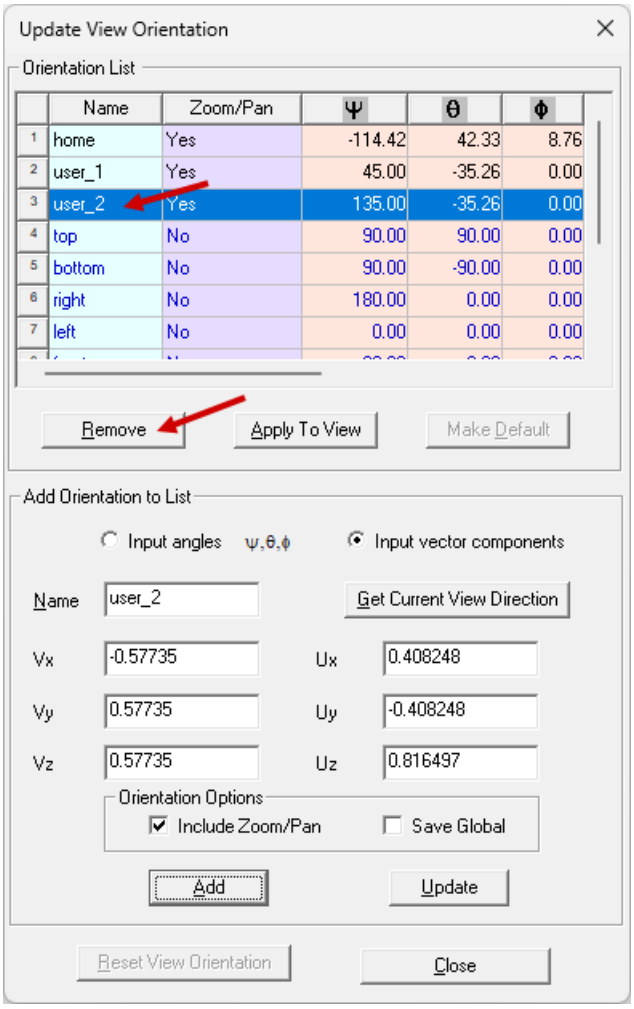
Removing an Orientation

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 2026 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

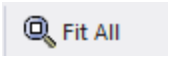
What do you want to do?

- Fit all objects or traces in a view window
- Fit selected objects in a view window

Ansys Electromagnetics Suite 2026 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

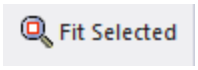
To fit one or more selected objects, 3D Components, or User Defined Models in the Modeler window or traces selected in the Reporter:

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.

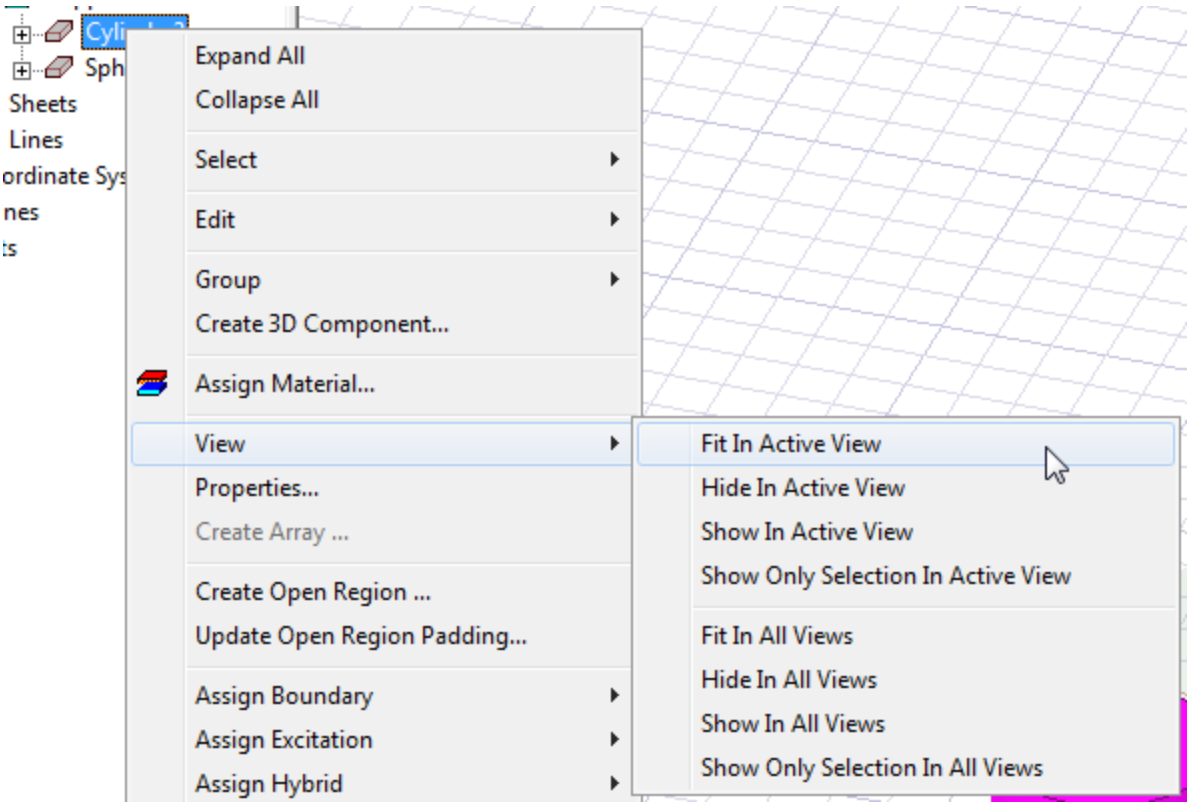
   You can also use the Fit Selected icon in the **Draw** and **Model** ribbon tabs:

Ansys Electromagnetics Suite 2026 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.

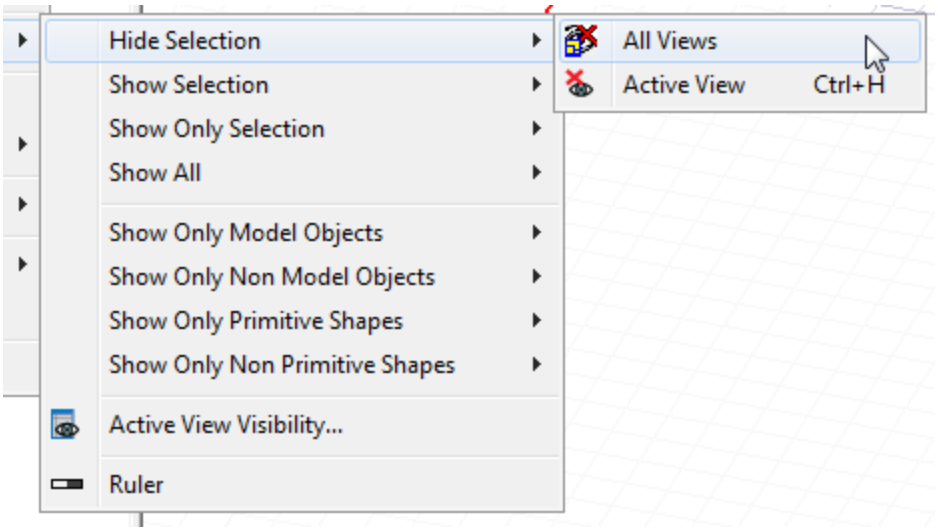
Hiding Objects from View

To hide selected objects.

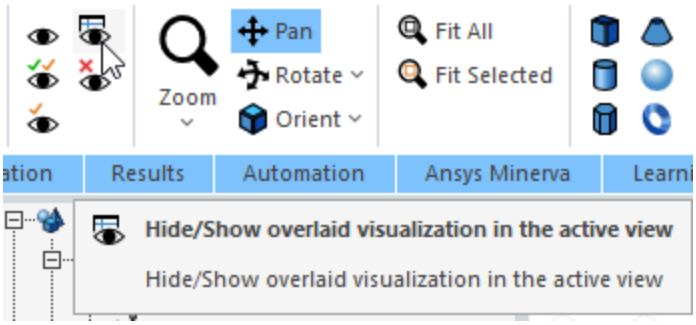
1. Select the object you want to hide from view. This enables the menu commands and **View, Draw,** and **Model** ribbon tabs icons for hiding objects.
2. Click **View** > **Visibility** > **Hide Selection** and one of the following commands:
   - **All Views** to hide the selected object in every open 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

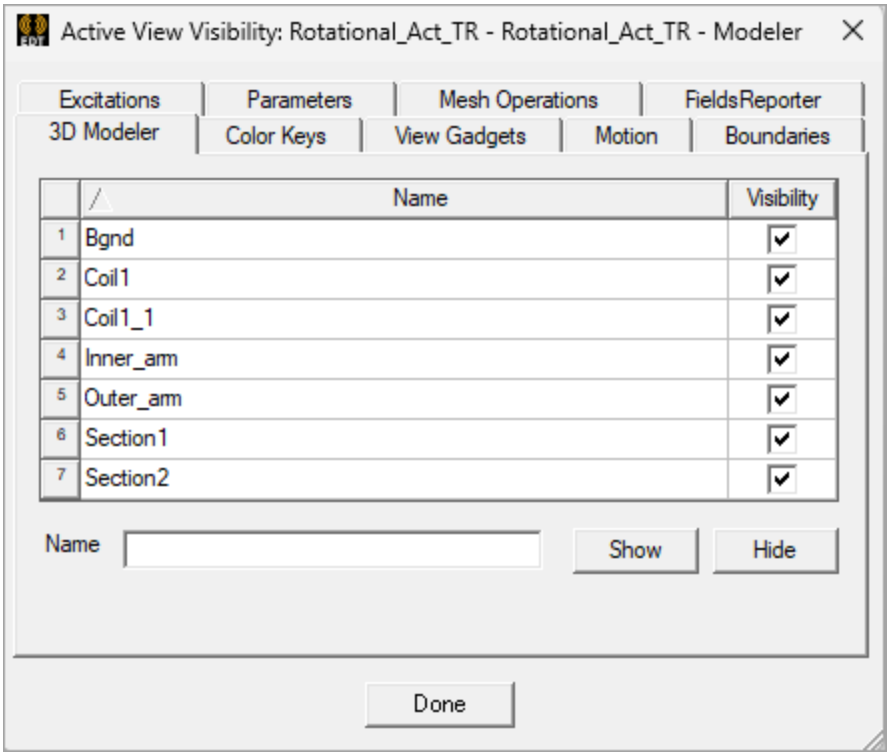
> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Showing Objects

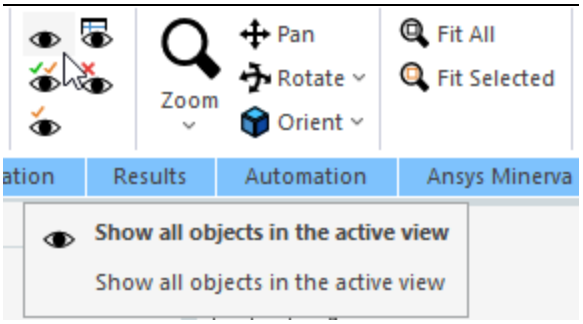
You can show or hide objects in several ways, using the **Active View Visibility** dialog box for individual objects, or **Visibility** menu selections for All Objects, Only Selection, Model Objects, Non-model Objects, Primitive Shapes, or Non-primitive shapes.

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**.

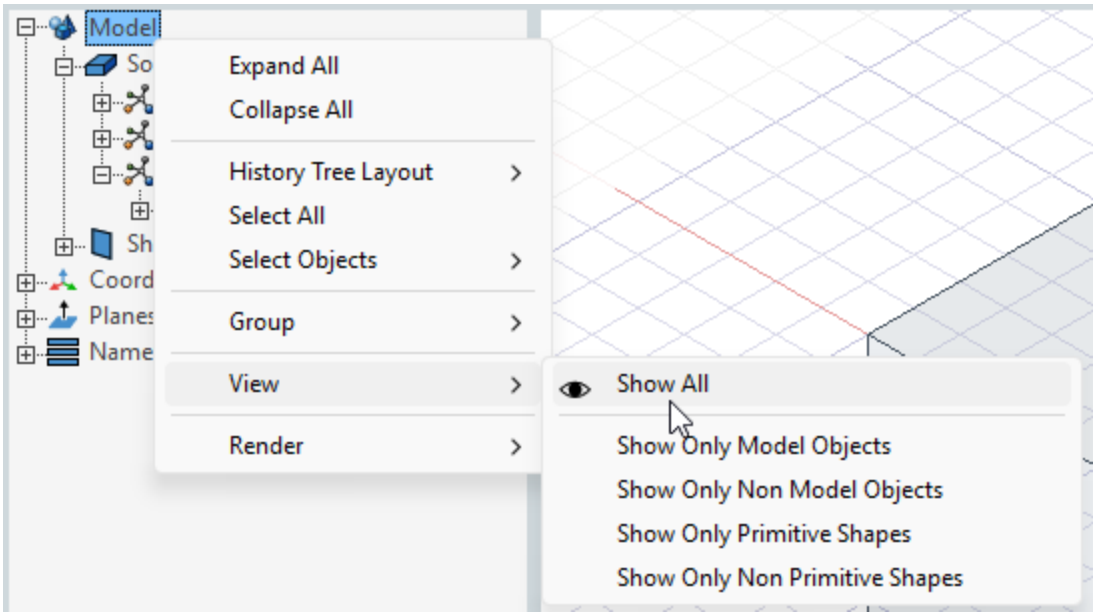


You can access the dialog box from the menu bar (**View** > **Visibility** > **Active View Visibility...**) or by clicking the Hide/Show visualization icons in the **View, Draw,** or **Model** Ribbon tabs.

Ansys Electromagnetics Suite 2026 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:



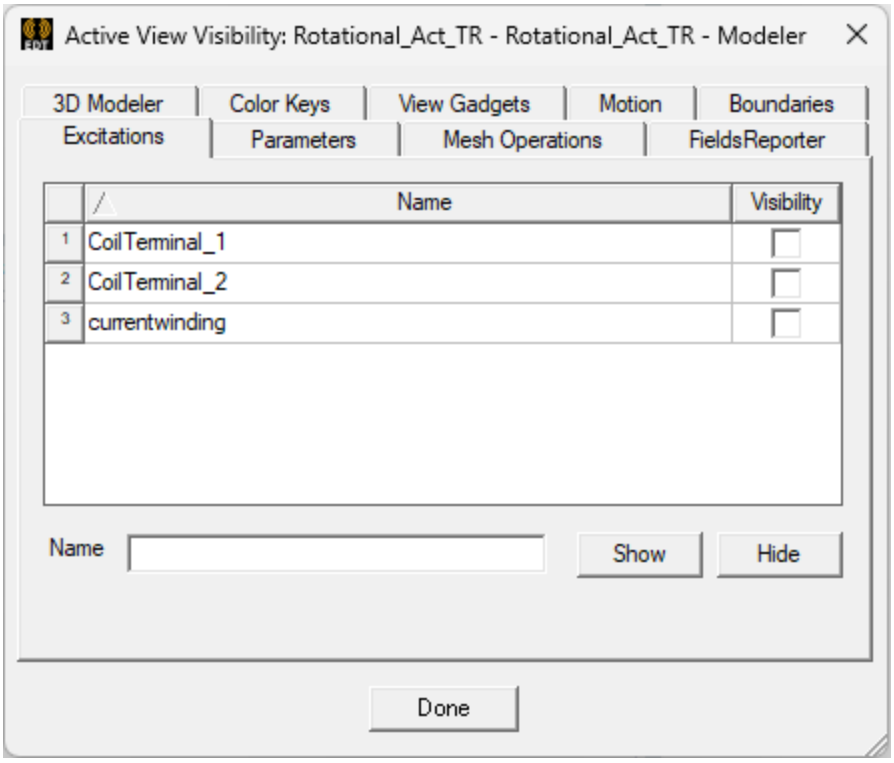
To show *one or more objects* that are currently hidden by using the *Active View Visibility* dialog box:

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 2026 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **View** > **Visibility** > **Show All** or use the shortcut menu in the Modeler window or History Tree and one of the following commands:
   - **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

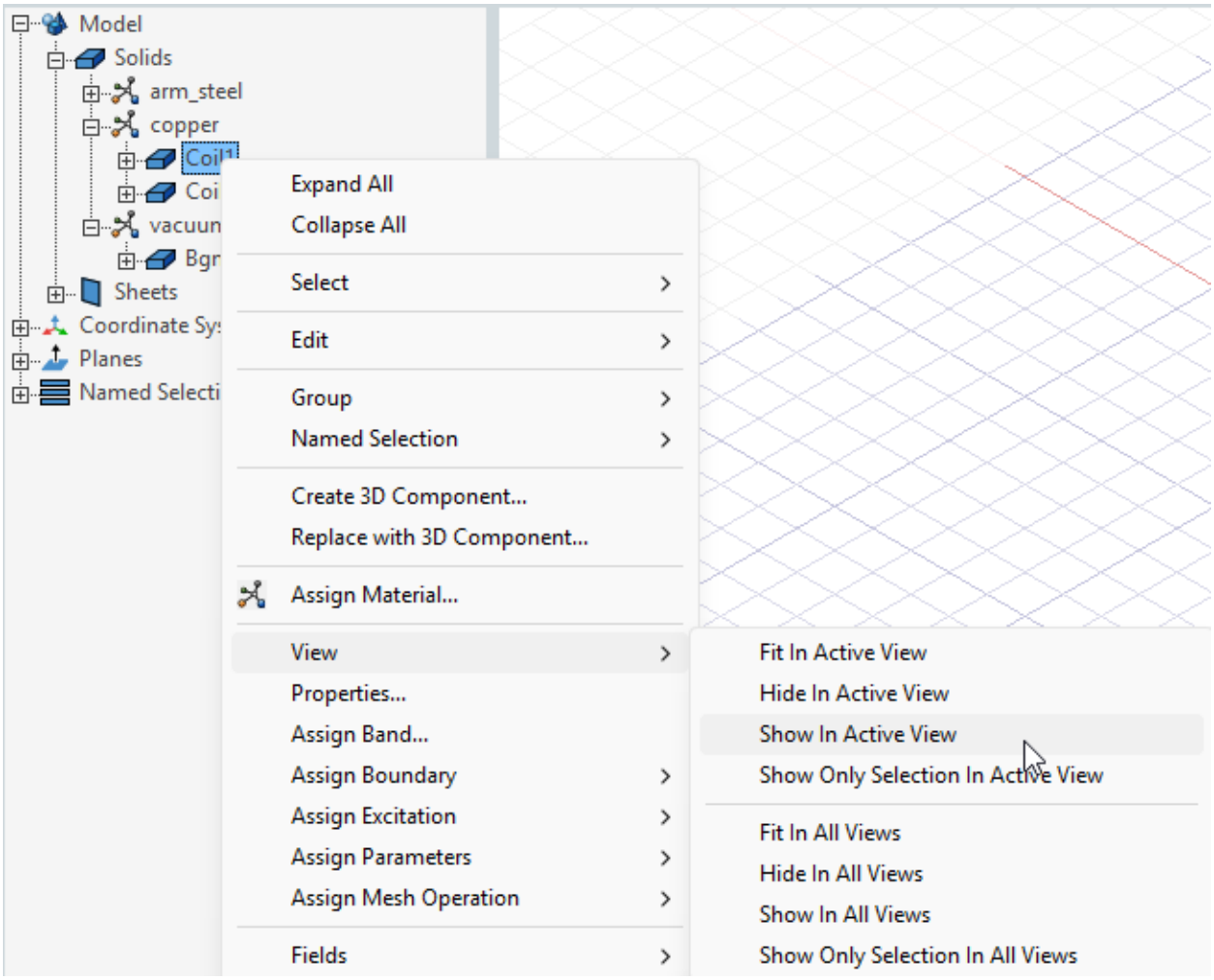
To show only *selected objects*:

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

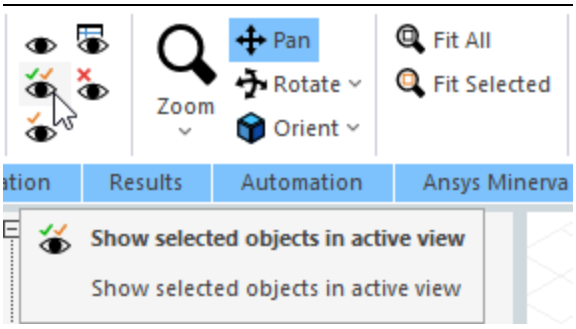
   You can also right-click an object in the History Tree and use the shortcut menu:

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also use the **View/Draw/Model** ribbon icons to **Show selected objects in all views** and **Show selected objects in active views**.



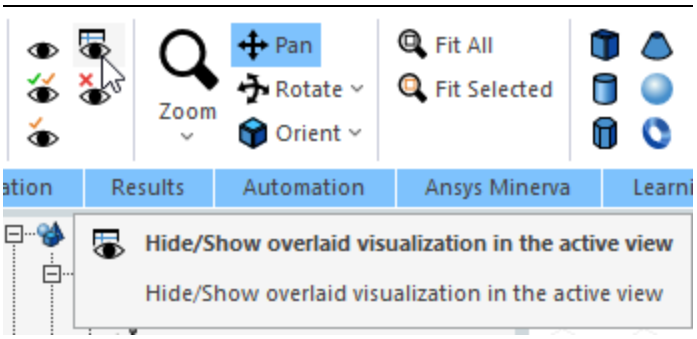
The selected objects reappear.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

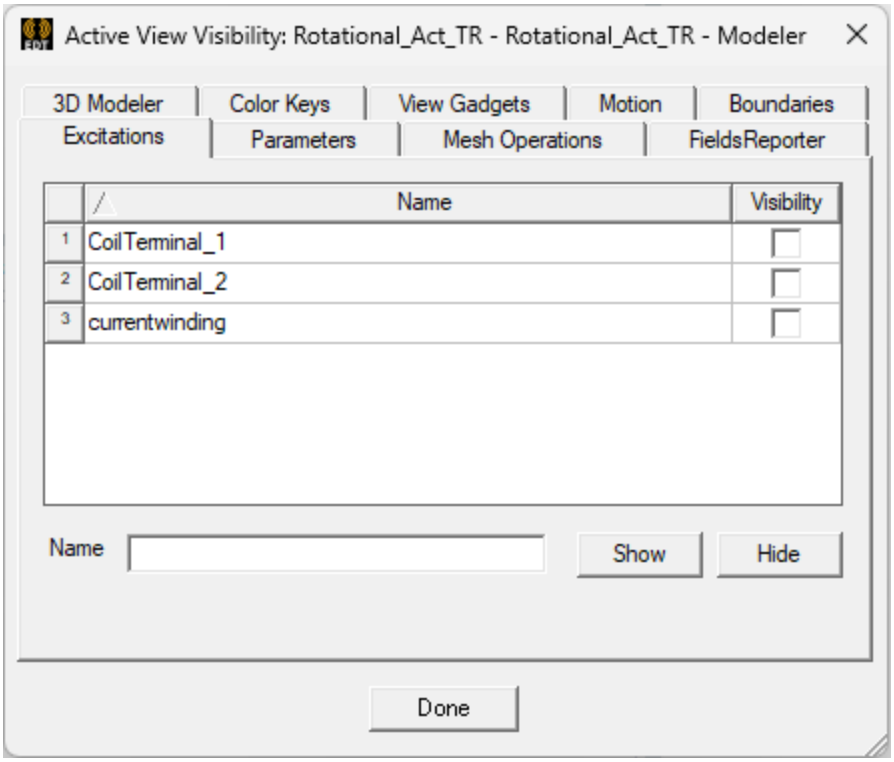
Active View Visibility Dialog Box

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.



The **Active View Visibility** dialog box appears.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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. Select **View**> **Render** and click one of the following:
    - **Wire Frame**:

      The objects in the view window appear as skeletal structures, enabling you to see all sides of the objects at one time. Objects in wireframe mode retain the same coloring as in smooth shaded mode.

      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. Note that when you interact with an object (for example, dragging or rotating) in smooth shaded mode, that object will temporarily shift to wireframe mode. Once you place the object, it will revert back to the smooth shaded display.

      You can also use the **F7** key to display a smooth shaded view.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:
   - **Wire Frame**.

     The objects in the view window appear as skeletal structures, enabling you to see all sides of the objects at one time. Objects in wireframe mode retain the same coloring as in smooth shaded mode.

   - **Smooth Shaded**.

     The objects in the view window will be displayed as shaded objects with smooth edges. Note that when you interact with an object (for example, dragging or rotating) in smooth shaded mode, that object will temporarily shift to wireframe mode. Once you place the object, it will revert back to the smooth shaded display.
4. Click **OK**.

   The rendering mode will be applied to all new objects you create.

Enabling Optimized Rendering (Beta)

Complex models can slow down the creation of new objects, visibility changes, and object interactions such as rotation, zoom, and selection.

To improve interactivity and frame rate of complex models, enable optimized rendering:

1. From Electronics Desktop, select **Tools** > **Options** > **General Options**.
2. From the **Options** window, click **Beta Options**.
3. In the **Beta Options** window, under **General**, select **Enable Optimized Rendering**.
4. Click **OK** to accept, then restart Electronics Desktop to apply the change.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

When this feature is enabled, the following functionalities may not work as expected:

- Vertices selection
- Object and face selection in wireframe mode
- Non-selected transparent objects have transparent outlines

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:

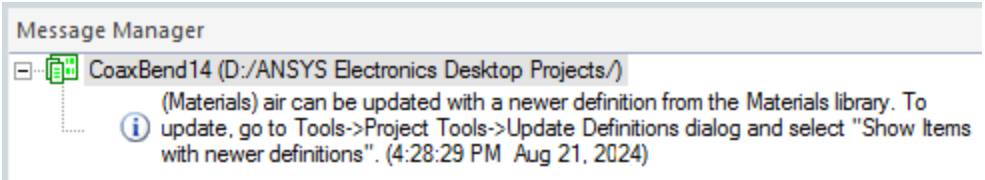
1. Click **View** > **Render** > **Enhanced Display** or press **F8**.

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.

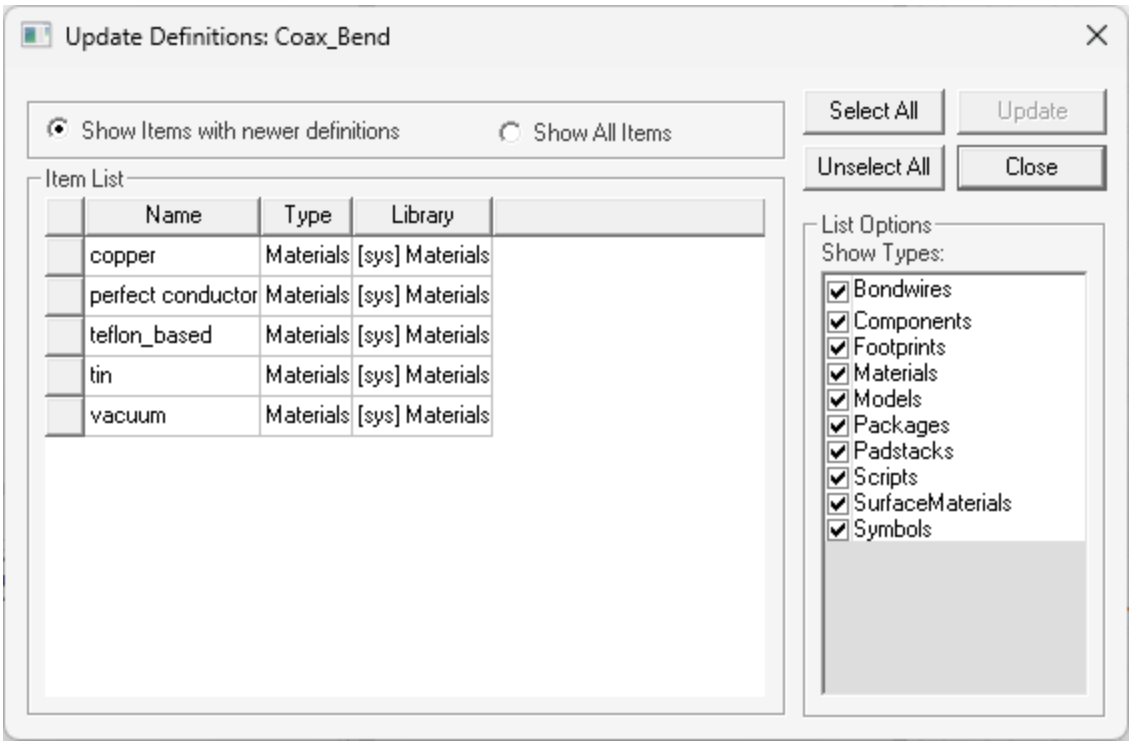
Sheet objects in the 3D modeler do not have a material assignment, but they could have a boundary assignment with material assigned to the boundary. In this case sheet object visualization resembles solid object visualization, where the material appearance from the assignment is used.

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.



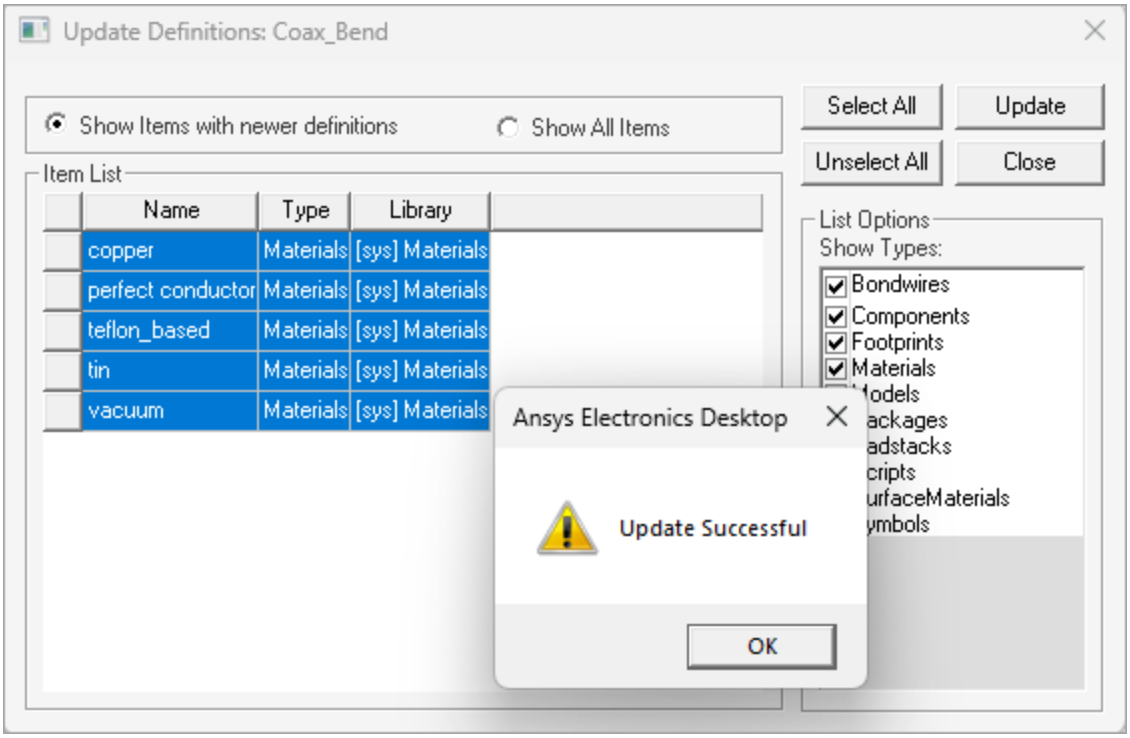
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

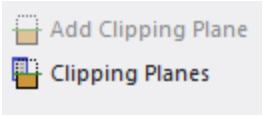
Using Clip Planes

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.

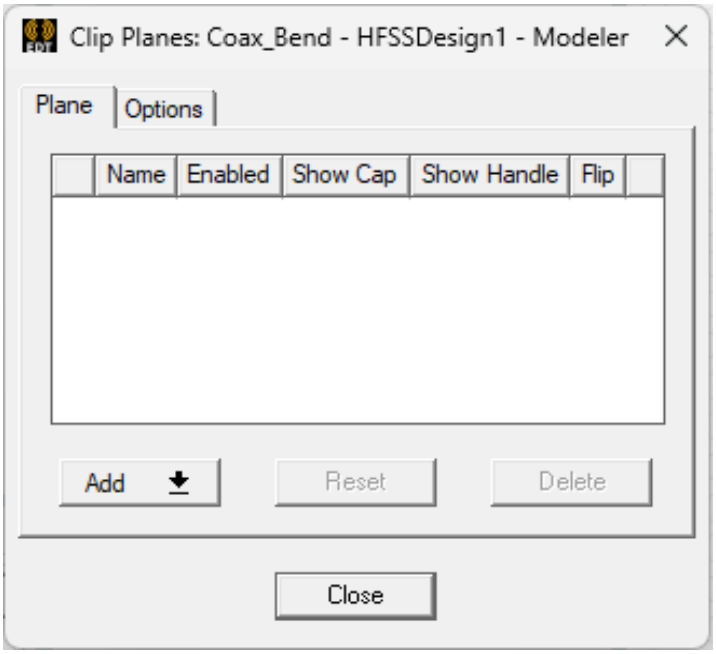
To add a clip plane:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

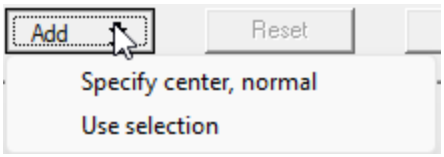
1. Click **View** > **Clipping** > **Clip Plane** or click the **Clipping Planes** icon on the **View** ribbon tab.



This displays the **Clip Plane** dialog box with the **Plane** tab selected.

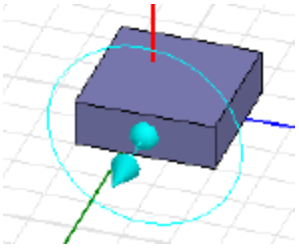


2. The **Add** button contains a drop-down menu with choices for **Specify center, normal**, and **Use selection**. If you want to use a selection, you must first select a face or an existing cut plane.

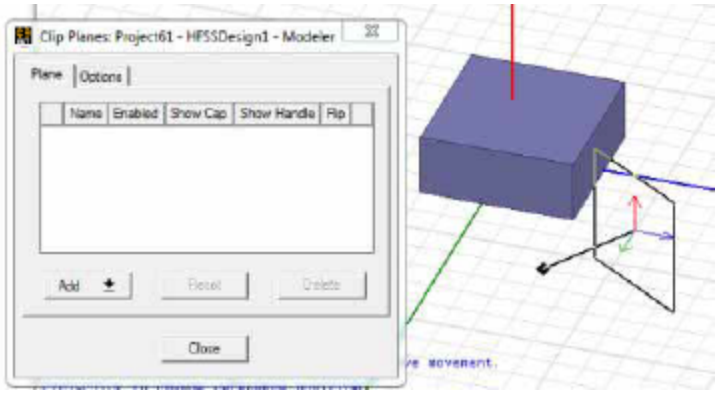


- If you first select a face or cut plane, and then click **Add** > **Use Selection**, the clip plane is added on that face.

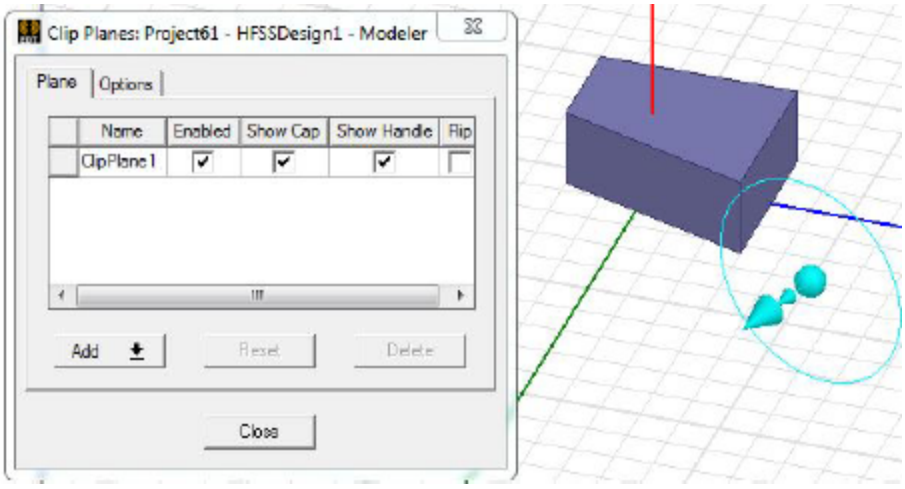
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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.



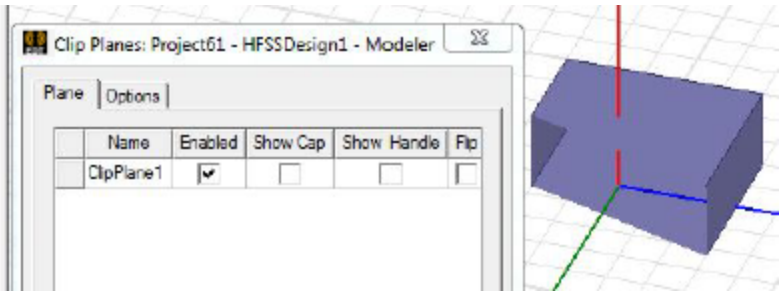
  b. Click again to set the reference position.



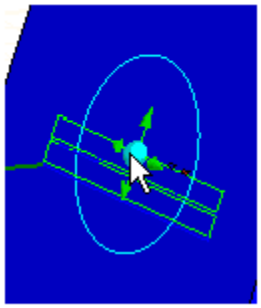
  After the second click, the clip plane is active. The handle is visible as a circle with a sphere at the center, and an arrow pointing the normal for the plane.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

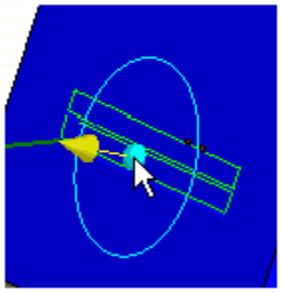
The **Clip Planes** dialog box shows the clip plane name, that it is enabled, the cap (which is the plane surface), and the handle. **Flip selection** lets you reverse the direction of the clip plane. If you deselect **Show cap** and/or **show handle**, they disappear from the display.



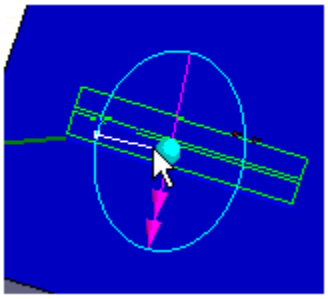
3. With **Show Handle** enabled, you can use the handle to manipulate the location and orientation of the clip plane. The handle changes appearance and function relative to the position of the cursor. Dragging the cursor makes use of the current function.



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 2026 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

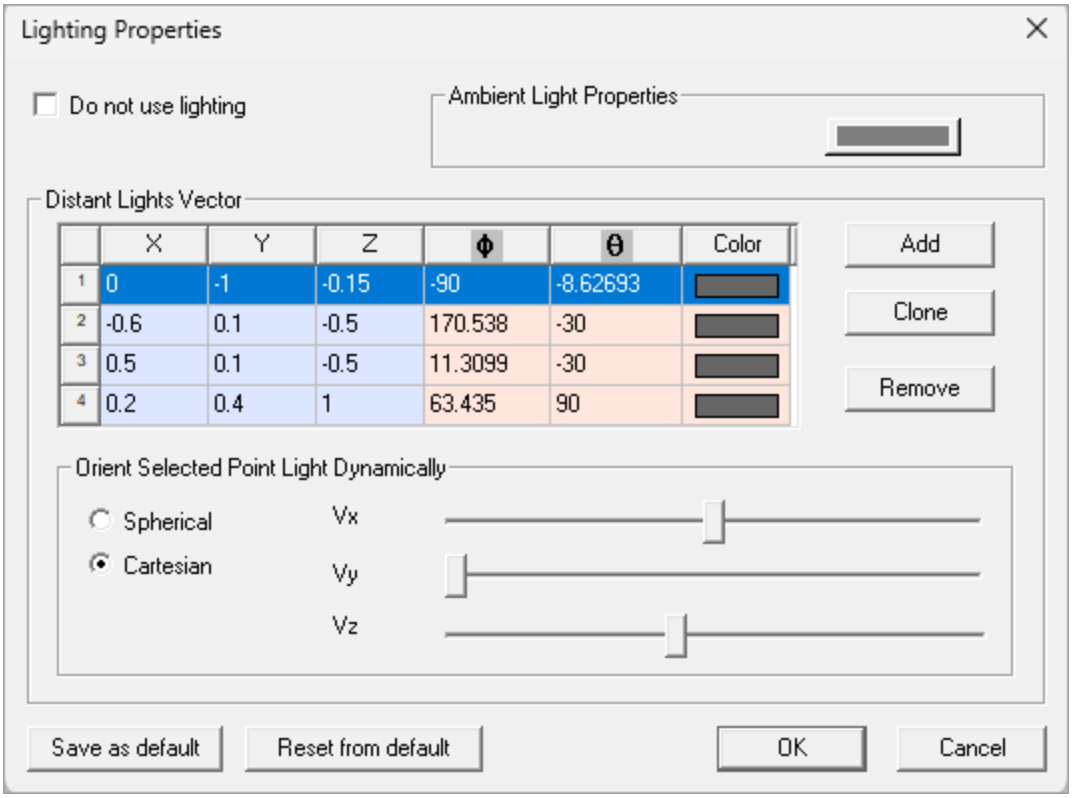
You have the option to emit the following types of light on a design:

- *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 2026 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:
      - 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.

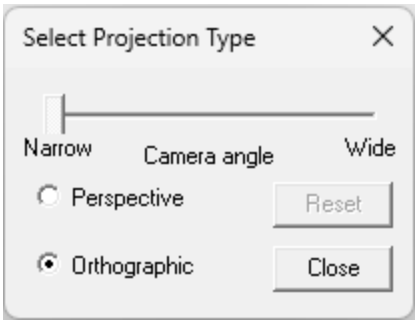
Setting the Projection View

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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.

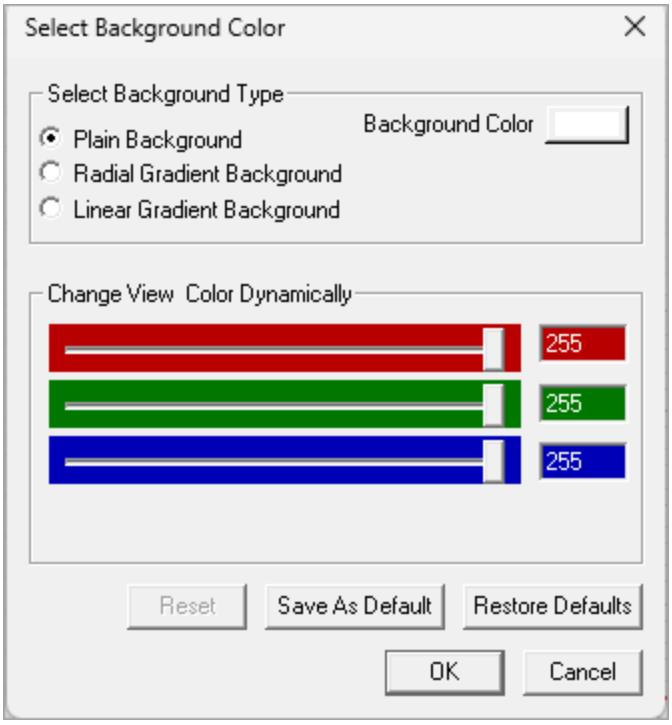
Setting the Background Color

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.

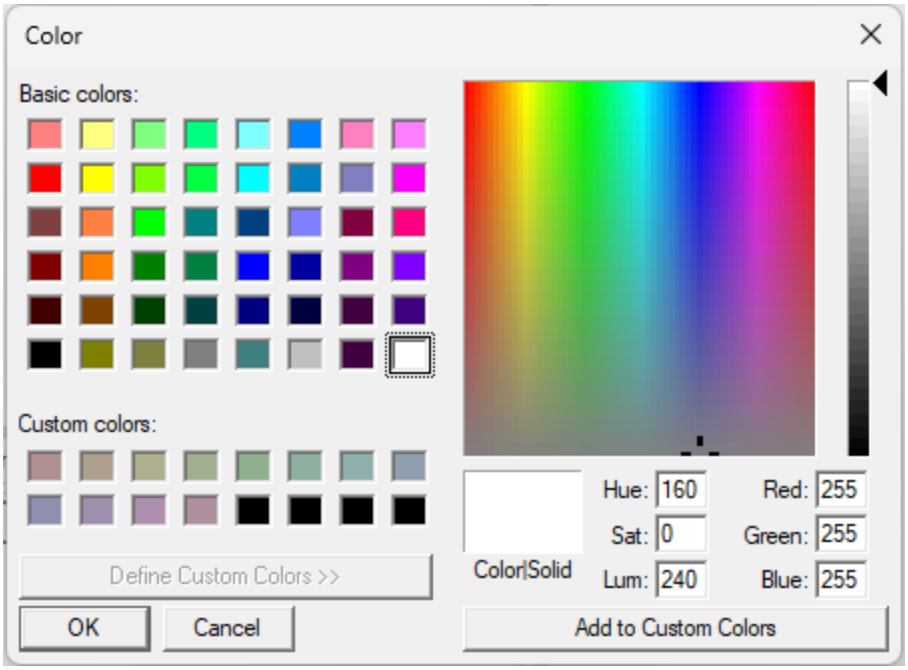
   The **Select Background Color** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

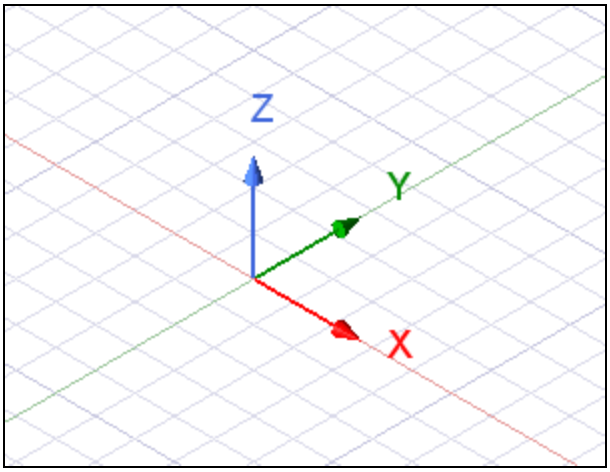
3. To assign a background color that gradually changes from one color to another, do the following:
   a. Select **Gradient Background**.
   b. Specify the background color at the top and bottom of the view window in one of the following ways:
      - Under **Select Background Type**, click the **Top 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **General** subgroup.
5. Select **Show orientation of selected objects**.
6. Click **OK** to close the dialog box.

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:

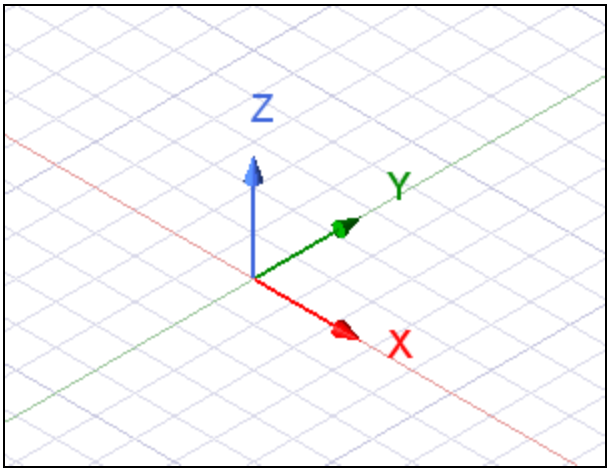
1. Click **View** > **Coordinate System**> **Triad**, then click one of the following:
    - **Hide** to hide the triad x-, y-, and z-axes at the lower right of the active view window.
    - **Show** to display the triad x-, y-, and z-axes in the lower right active view window.
    - **Auto** to generally hide the triad axes.

Choosing Grid Settings

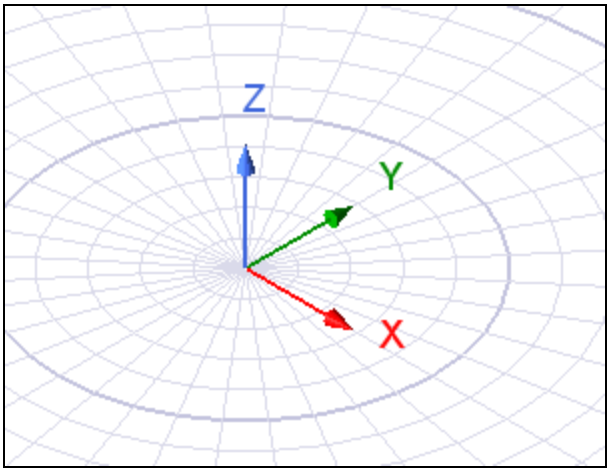
The grid displayed in the **Modeler** window is a drawing aid that helps to visualize the location of objects.

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 2026 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

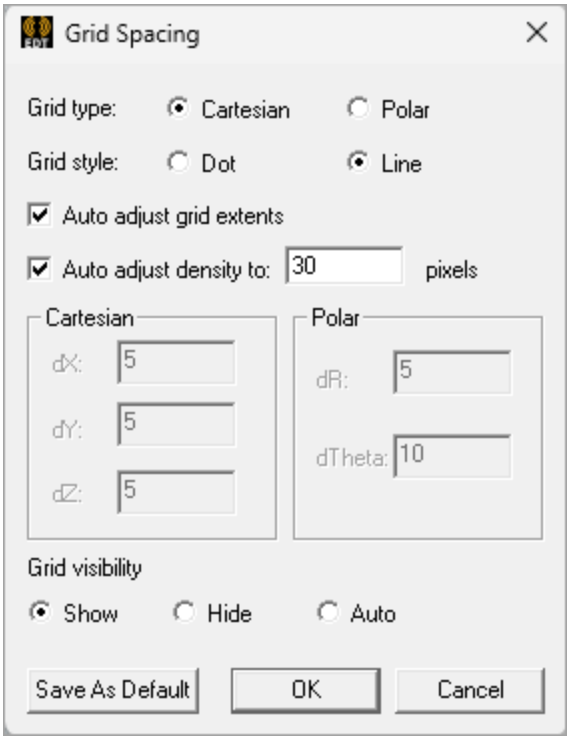
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Snap settings
- Grid plane

Setting the Grid Type

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



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.

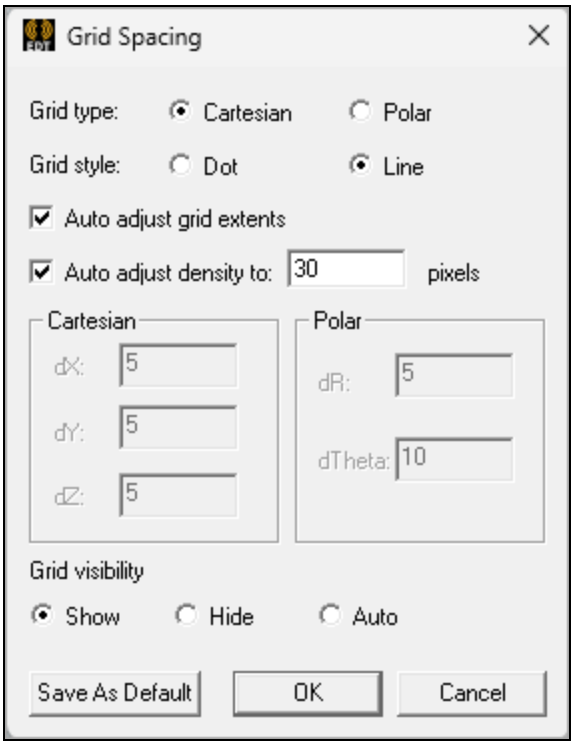
For polar grids, you define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Grid Style

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



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.

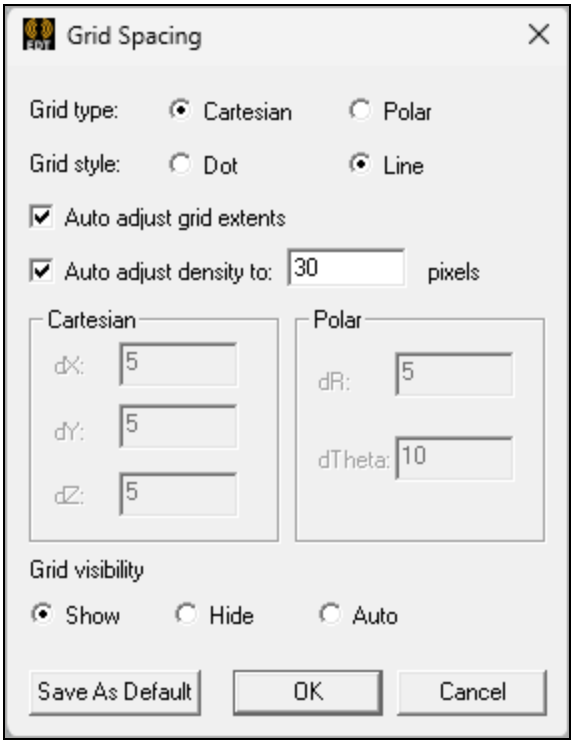
3. To show a Minimal grid around existing objects, check **Auto adjust grid extends**. To display a grid as a plane, deselect **Auto adjust grid extends**.

Setting the Grid Density and Spacing

1. Click **View** > **Grid Settings**.

   The **Grid Settings** dialog box appears.

Ansys Electromagnetics Suite 2026 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.

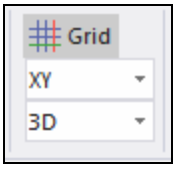
    The distance between grid points will increase and decrease proportionately as you zoom in and out in the active Modeler window.

4. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Grid's Visibility

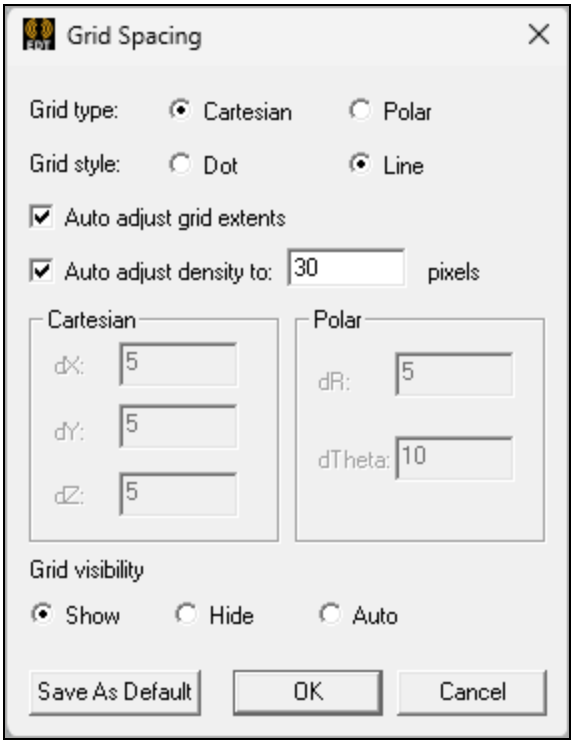
To hide the grid in the Modeler window, on the **Draw** ribbon tab, click the Grid toolbar icon. Click it again to show the grid.



Alternatively:

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



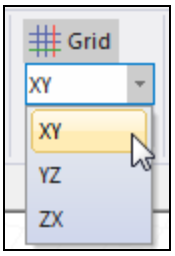
2. Select **Show** to make the grid always visible in the active **Modeler** window. Select **Hide** to make the grid always invisible in the active **Modeler** window. Select **Auto** so the grid appears only while you are drawing an object in the **Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7 - 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 Icepak. 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 2026 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 Icepak 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

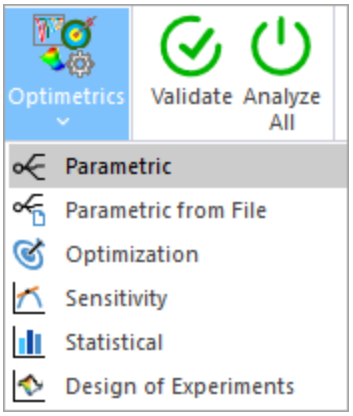
of variable values within a range that you want the software to solve when you run the parametric setup.

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 **Icepak** > **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 2026 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 **Icepak** > **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 box.
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 2026 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 2026 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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:

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:

Ansys Electromagnetics Suite 2026 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.
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, Icepak extracts the solution quantities and lists them in the results table.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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**.

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 2026 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.

Using Distributed Analysis

If you have purchased the appropriate license, the Electronics Desktop supports a distributed solve, which involves distributing rows of a parametric table during the 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 Manager 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:

   - Right-click the progress window, and select **Show Subtask Progress Bars**.

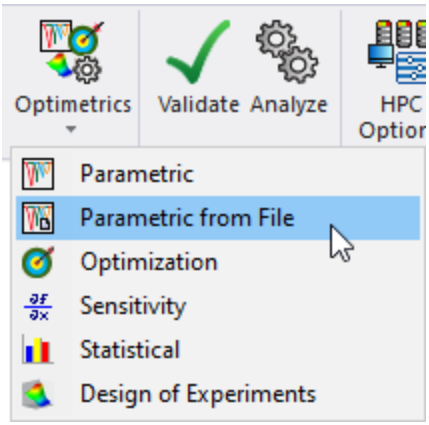
     To hide the child progress bars:

Ansys Electromagnetics Suite 2026 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 **Hide Subtask Progress Bars**.

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 **Icepak** > **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.

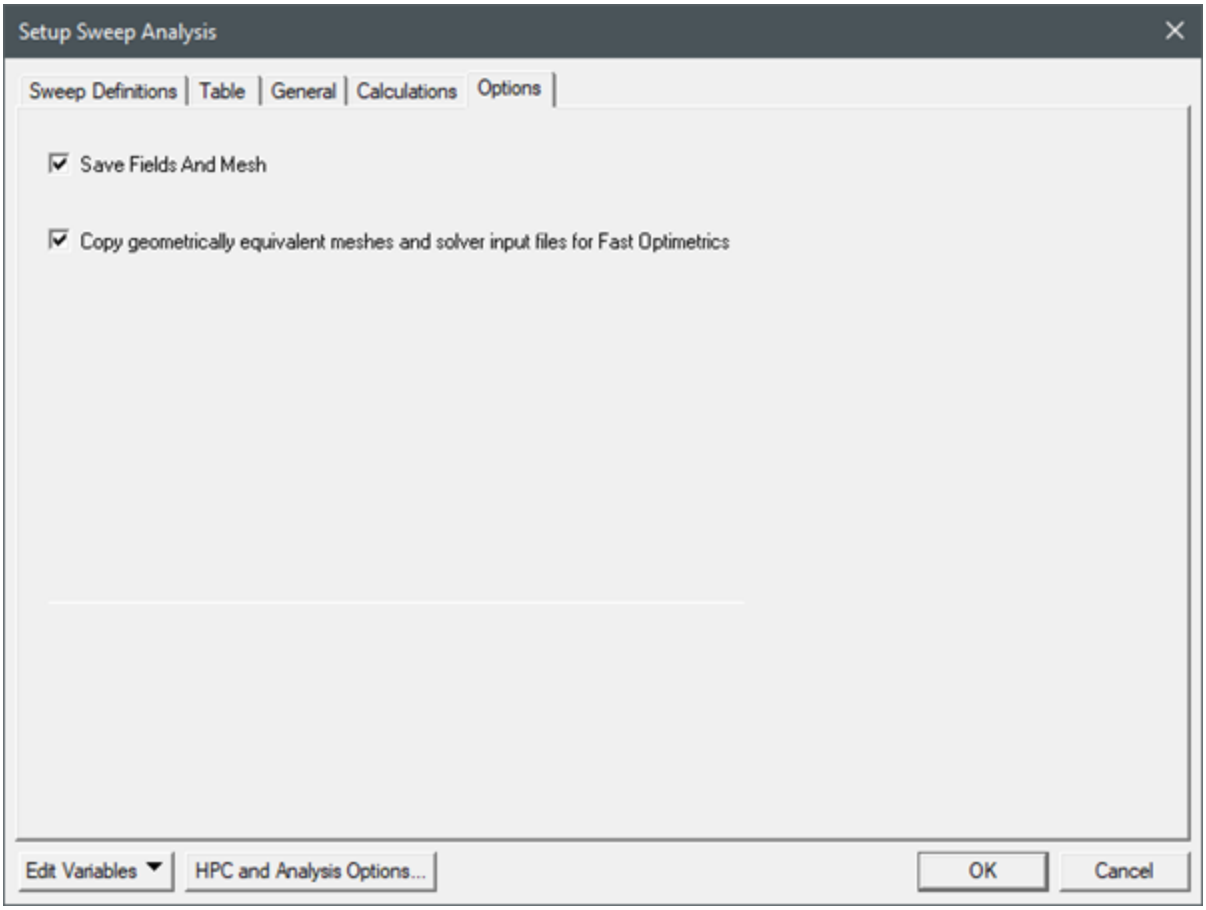
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Fast Optimetrics

Fast Optimetrics allows you to copy solution data that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation, thereby decreasing the overall analysis time. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

To copy and reuse solution data 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 and solver input files for Fast Optimetrics**.

   Ansys Electronics Desktop copies the equivalent meshes and solution data for a particular parametric sweep for reuse on each geometrically equivalent sweep variation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

In Icepak sweeps, the **Copy geometrically equivalent meshes and solver input files for Fast Optimetrics** option does not copy solution data when the design contains any of the following:

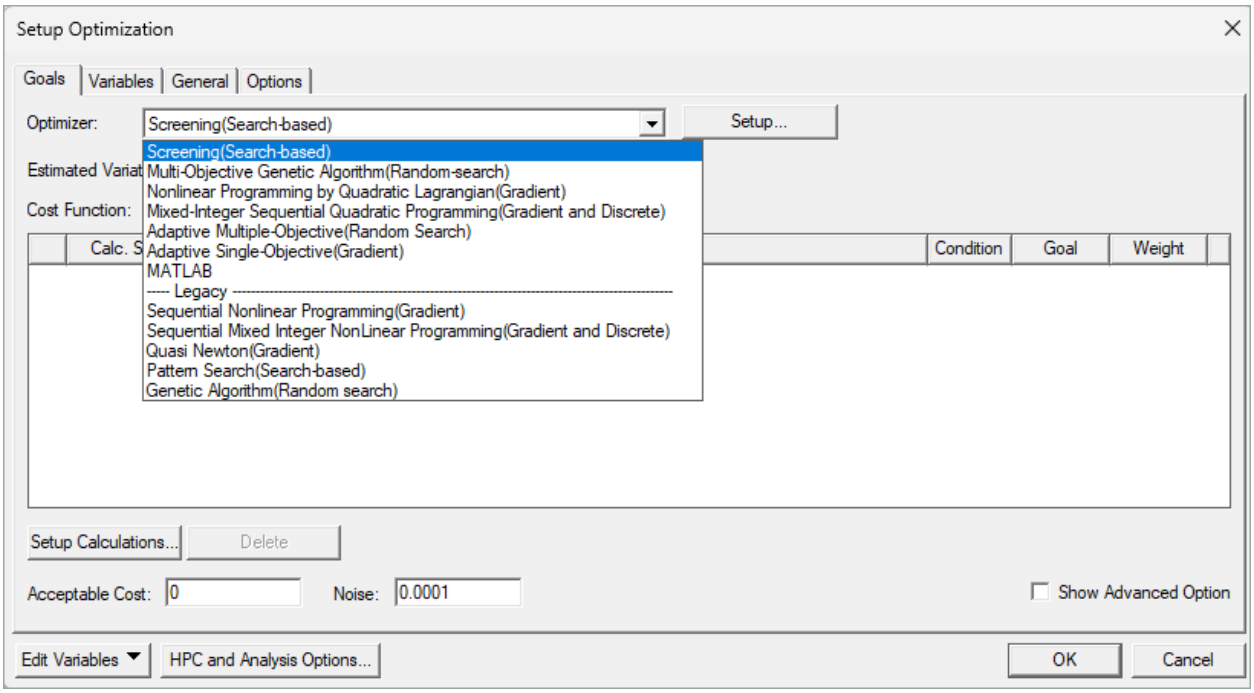
- a PCB native component
- an EM loss boundary condition that has a parameter defined using a variable
- a network boundary condition that has a parameter defined using a variable

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.



Goal-Driven Optimizers

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi-Newton (Gradient)
- Pattern Search (Search-Based)
- Genetic Algorithm (Random Search)

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

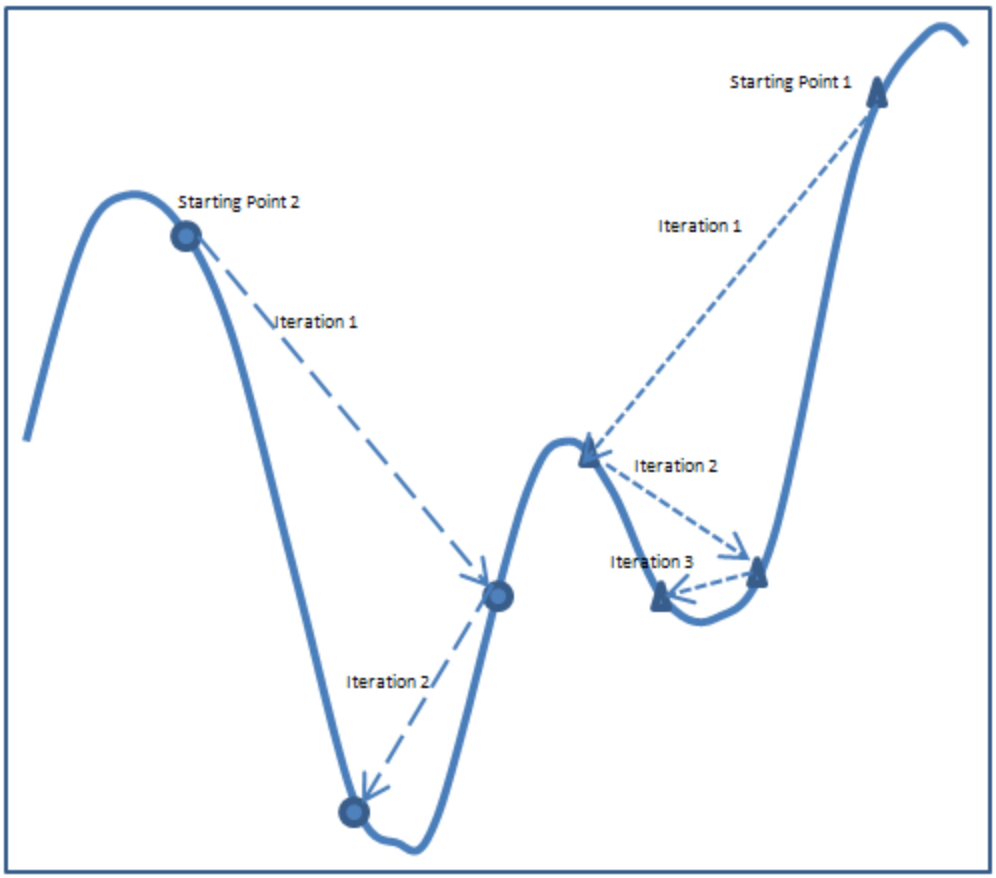
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

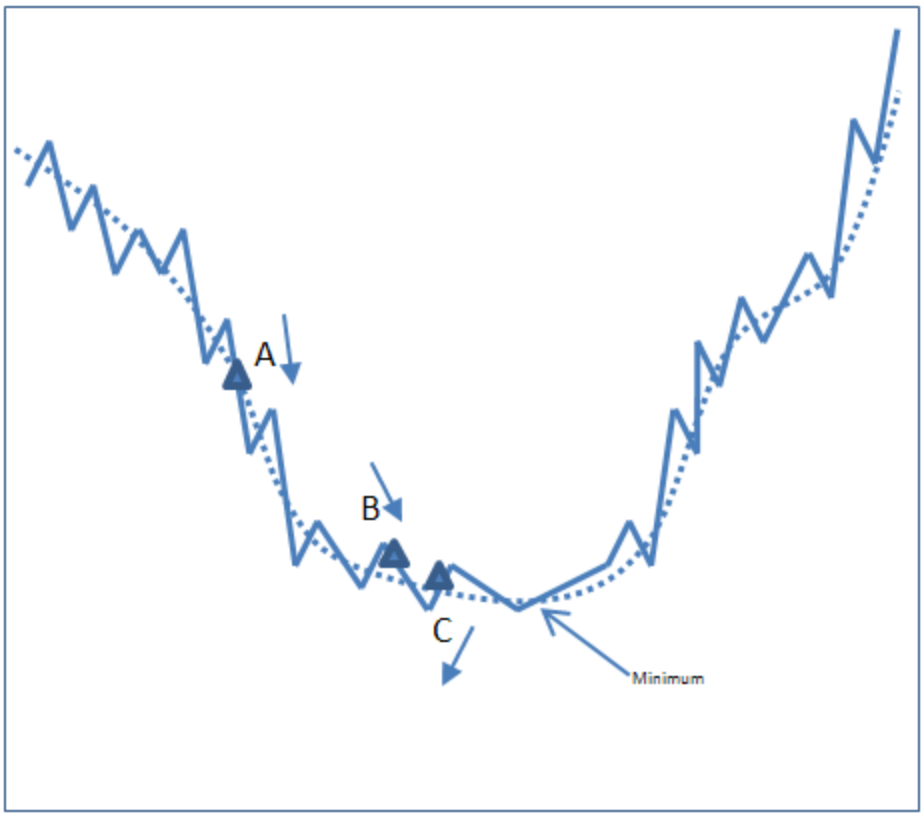
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Schoenberg, Ronald. *Optimization with the Quasi-Newton Method*. Aptech Systems, Inc. 2001.

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.

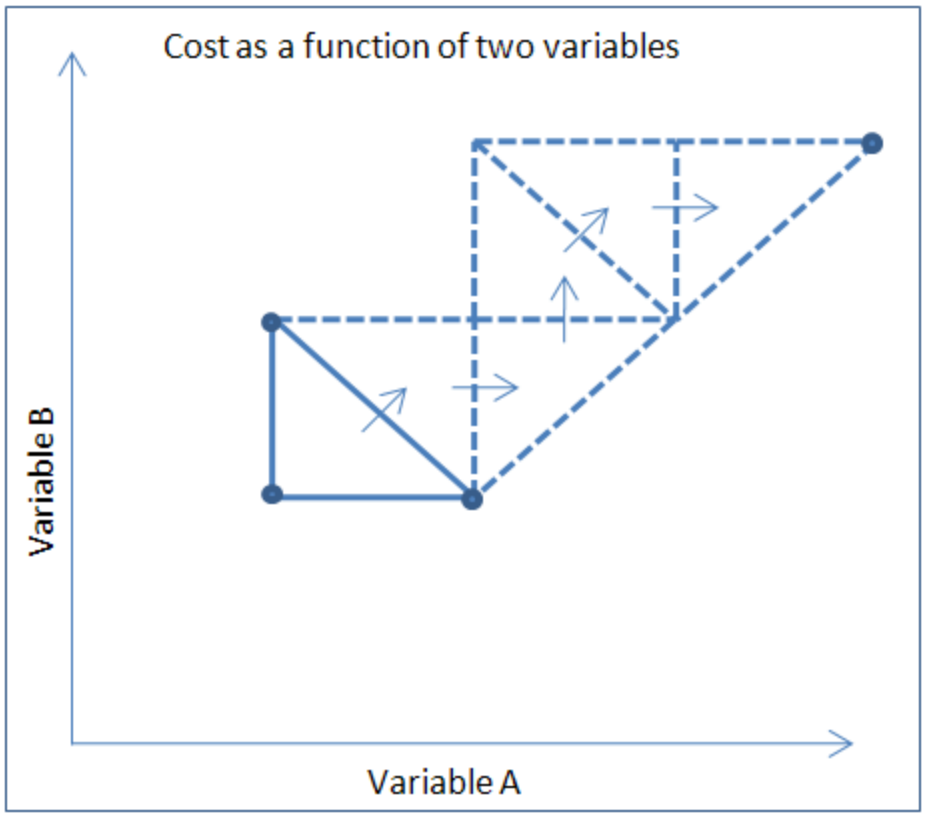
The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern 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.

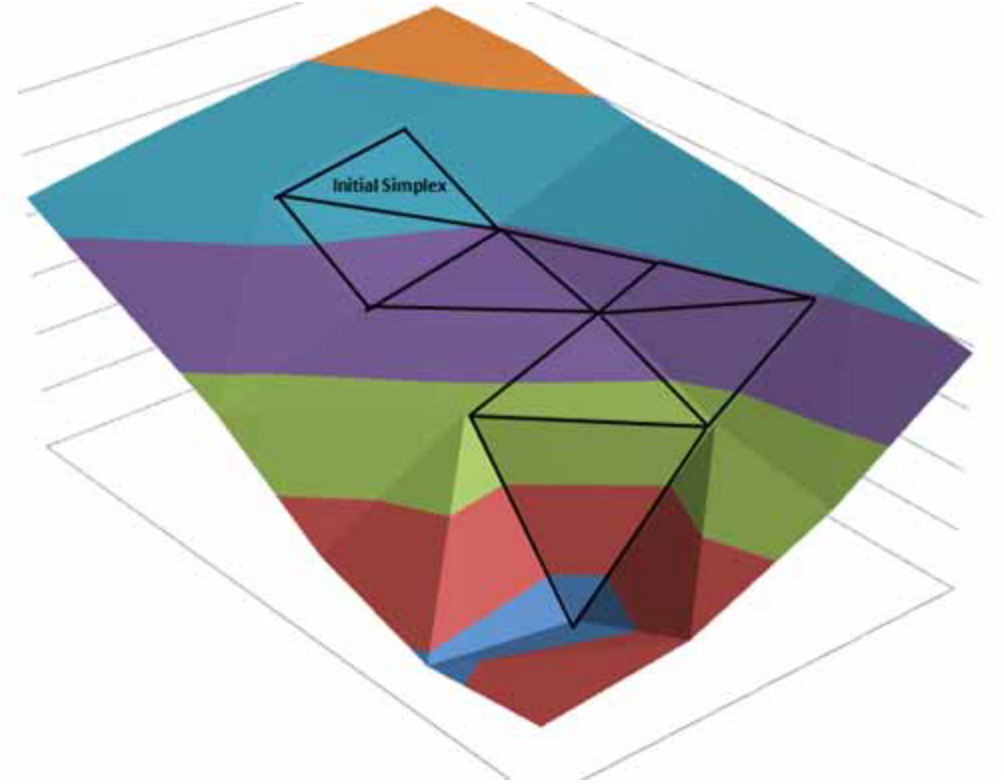
> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

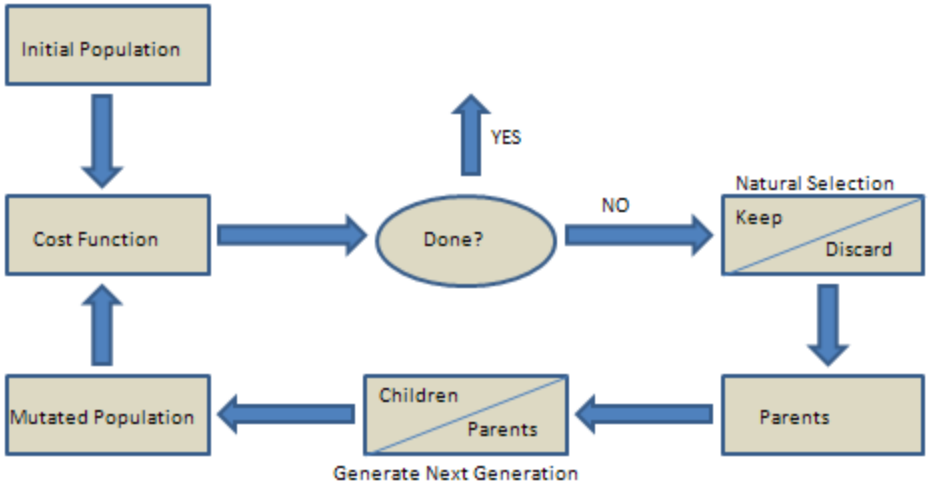
resembles the setup for SNLP, except that you must flag the supporting integer variables. You can set up internal variables based on the integer optimization variable.

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.

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).



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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
-----------------------------------------------------------------------
---------------------------------
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
MATLAB Version: 8.1.0.604 (R2023a)
MATLAB License Number: 162684
Operating System: Microsoft Windows 10
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
```

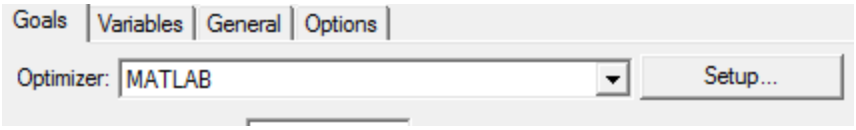
The answer will be 1 if the MATLAB Optimization Toolbox is licensed or 0 otherwise.

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.

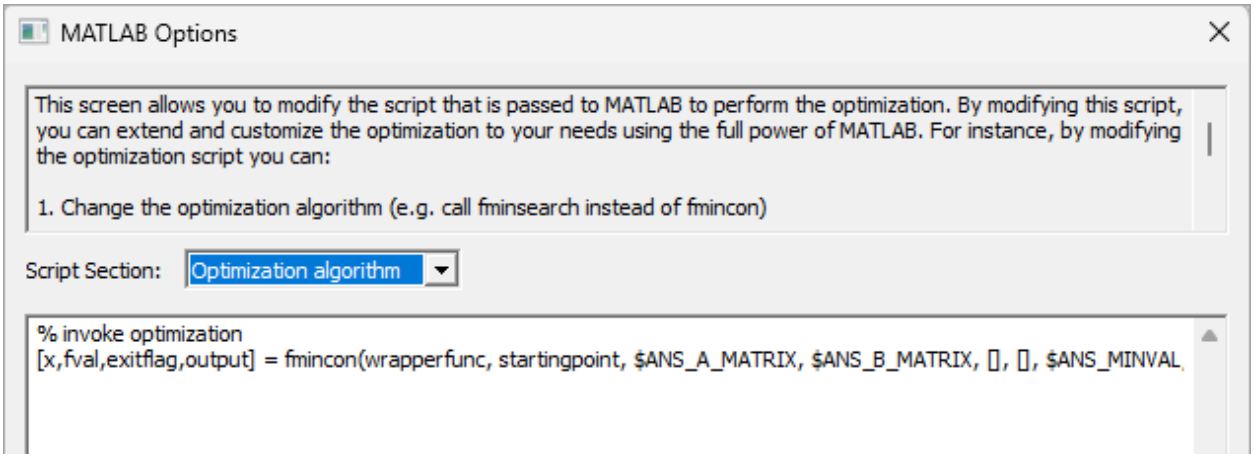
MATLAB Optimization Setup

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 2026 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 drop-down menu 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
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
options = optimset(options, 'display', 'iter')
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 2026 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 2026 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
```

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.

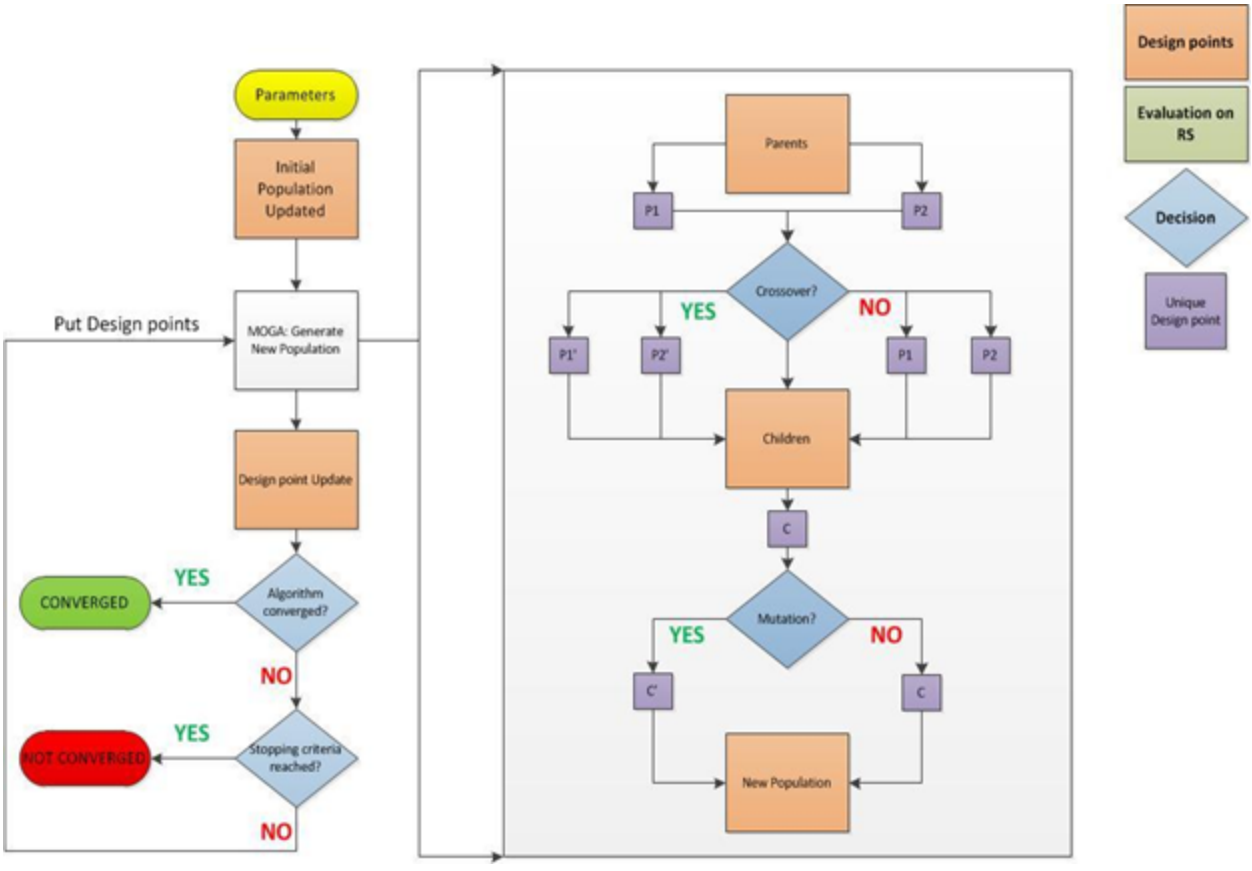
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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.

   - **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

     $$\text{Offspring1}=\mathbf{a}*\text{Parent1}+\left(1-\mathbf{a}\right)*\text{Parent2}$$
     $$\text{Offspring2}=\left(1-\mathbf{a}\right)*\text{Parent1}+\mathbf{a}*\text{Parent2}$$

     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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

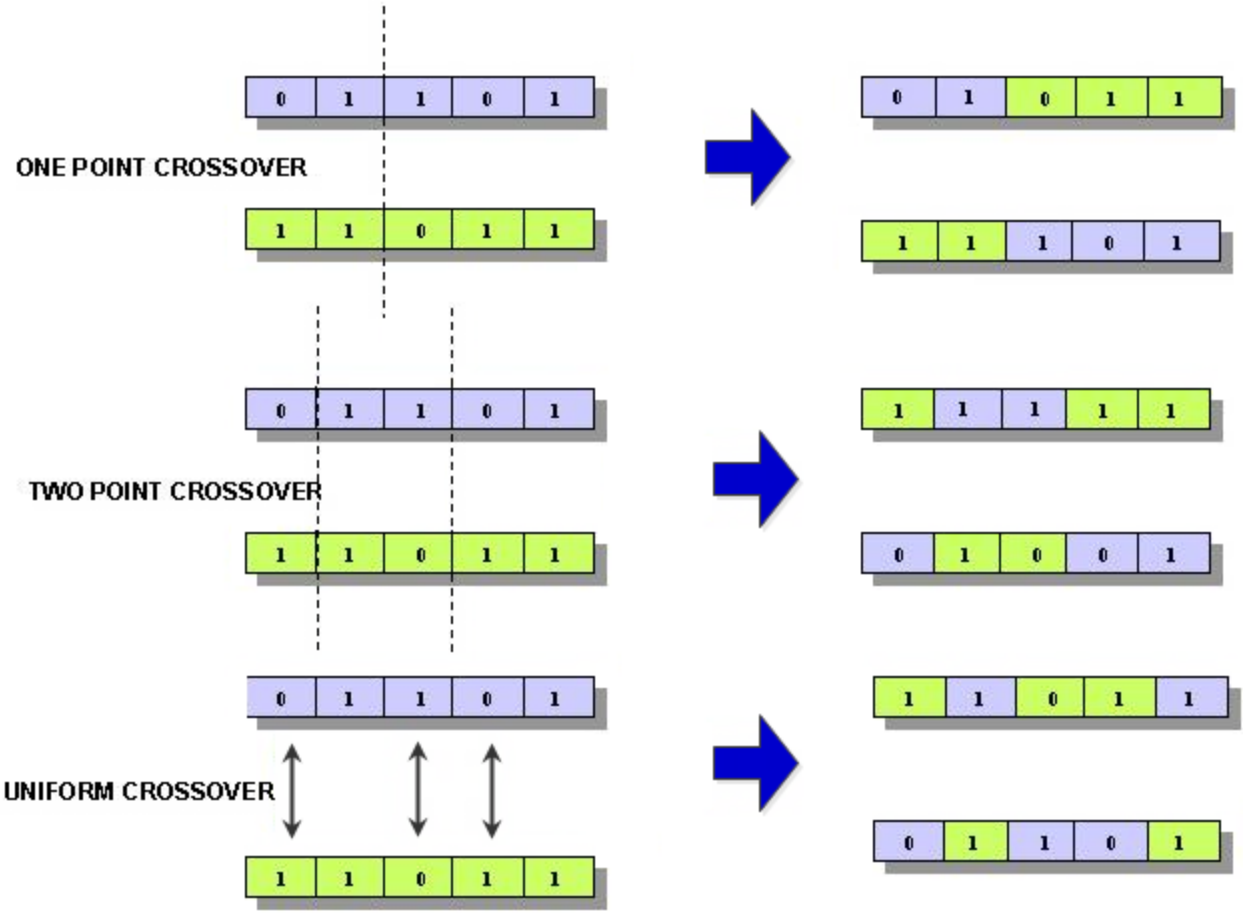
- Two-Point

  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.

- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

   - **Mutation for Continuous Parameters**

     For continuous parameters, a polynomial mutation operator is applied to implement mutation.

$$C = P + (\text{UpperBound} - \text{LowerBound})\delta$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

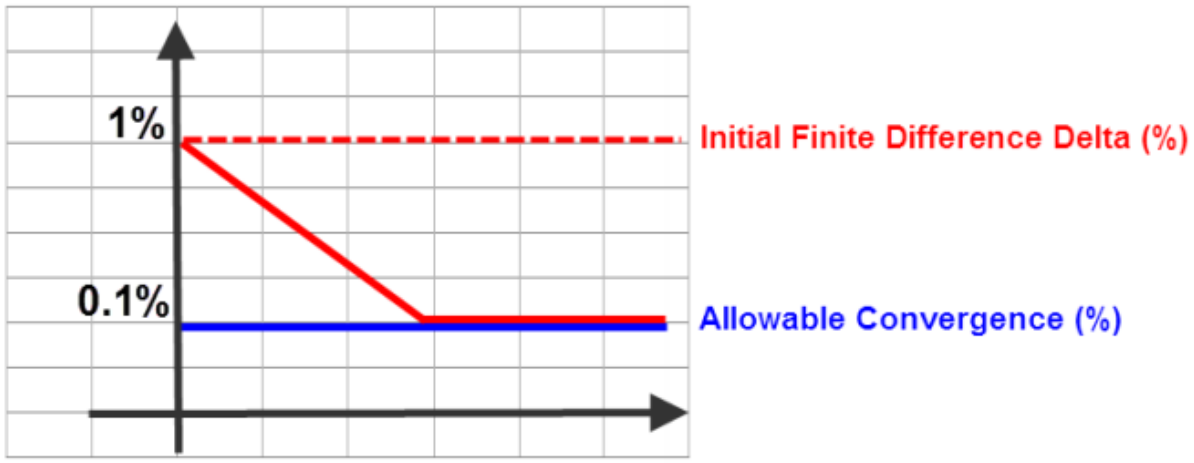
For NLPQL and MISQP, the default convergence rate, which is specified by the Allowable Convergence (%) property, is set to 0.1% for a Direct Optimization system. The maximum value for this property is 100%. This is computed based on the (normalized) Karush-Kuhn-Tucker (KKT) condition. This implies that 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. 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.

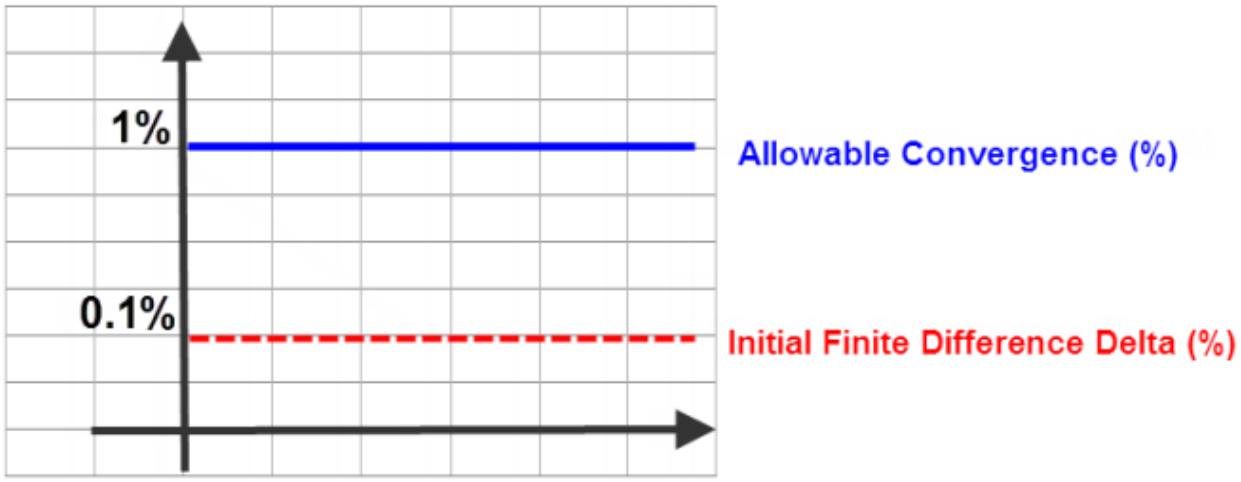
The advantage of this approach is that for large problems, it is possible to get a near-optimal feasible solution quickly without being trapped into a series of iterations involving small solution steps near the optima. To work most effectively with NLPQL and MISQP, keep the following guidelines in mind:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.



- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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:

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Adaptive Multiple Objective Optimization

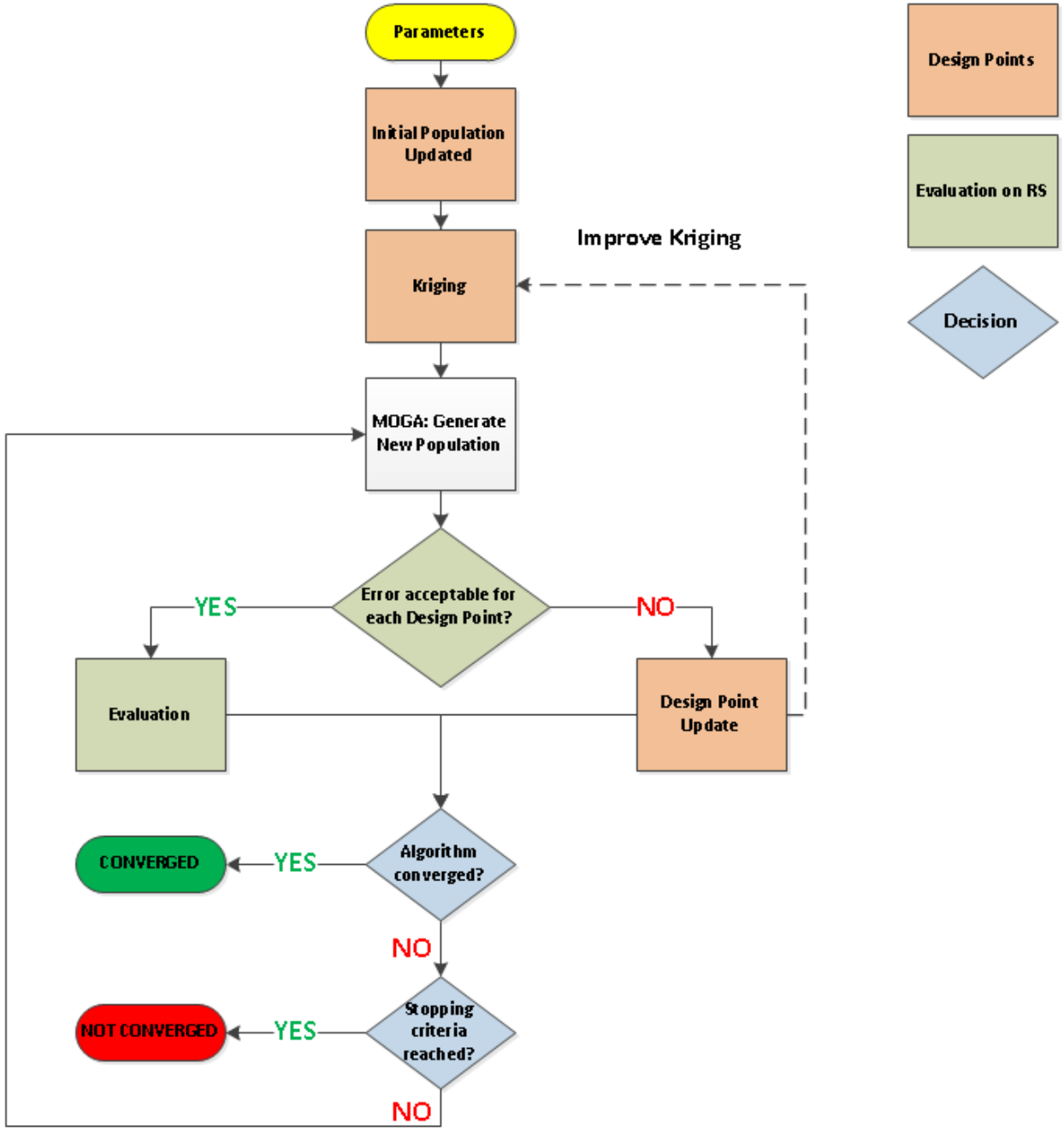
Adaptive Multiple-Objective (AMO) is a mathematical optimization that combines a Kriging response surface and MOGA. It allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. Except when necessary, the optimizer does not evaluate 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 2026 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 2026 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 2026 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.

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:

$$g_k(\{x\}) \leq 0, \forall k = 1,2,\ldots,K$$

$$h_l(\{x\}) = 0, \forall I = 1,2\ldots,L$$

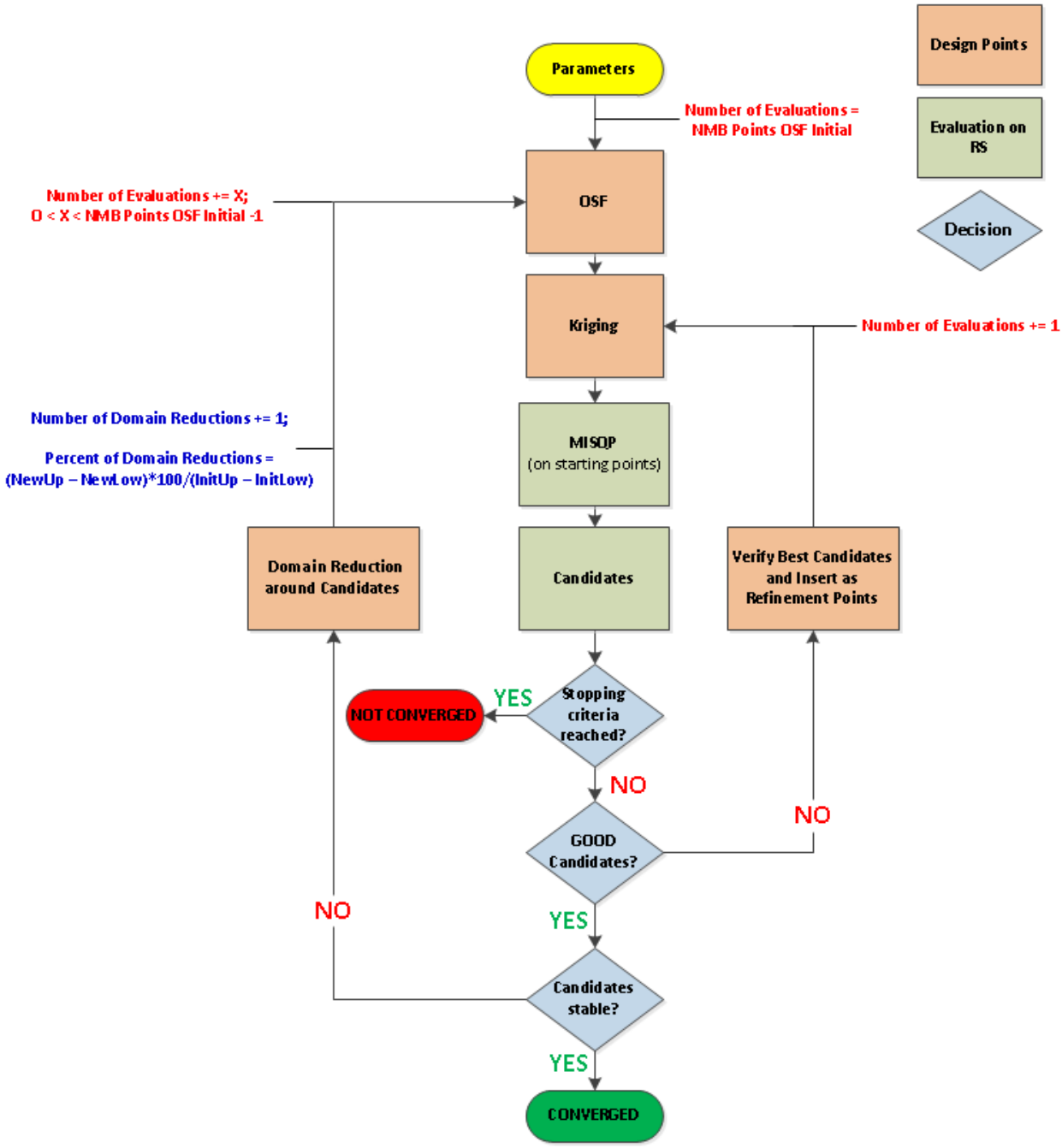
Where:

$$\{x_L\} \leq \{x\} \leq \{x_U\}$$

ASO Workflow

The workflow of ASO follows:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

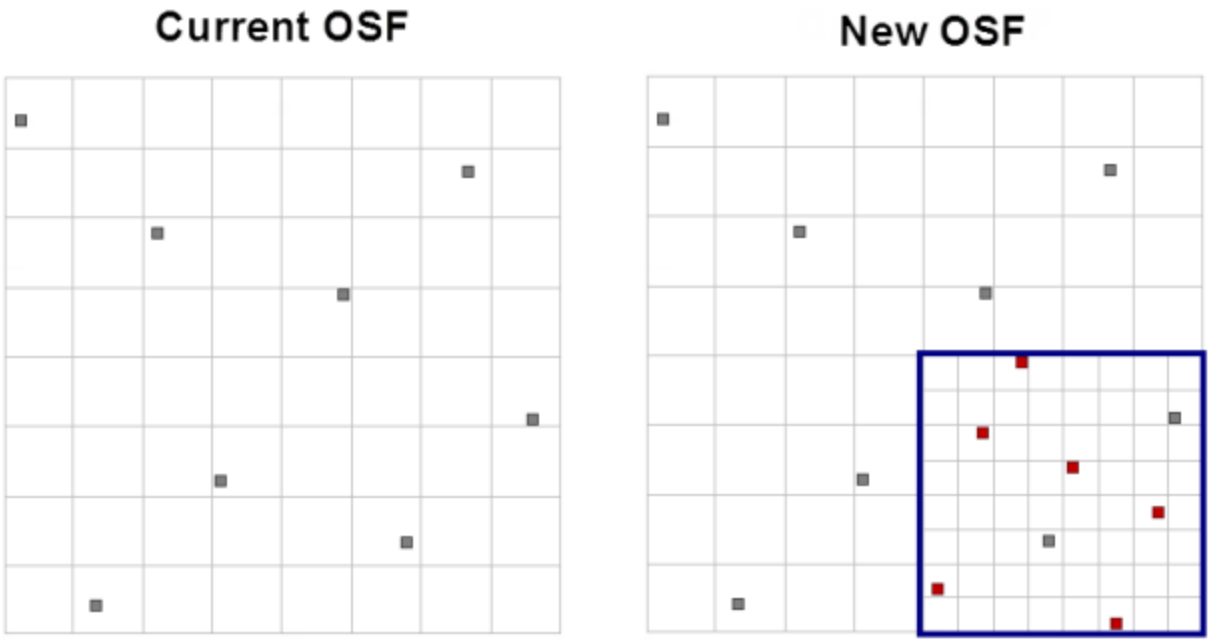
AMO Steps

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 2026 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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:

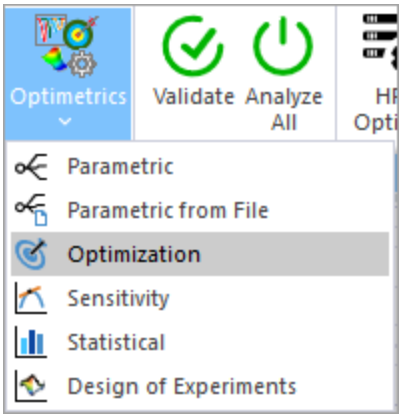
- Screening (Search based)
- Multi-Objective Genetic Algorithm
- Nonlinear Programming by Quadratic Lagrangian (Gradient)
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)
- Adaptive Multiple Objective (Gradient)
- Adaptive Single Objective
- MATLAB

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 2026 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 Up 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

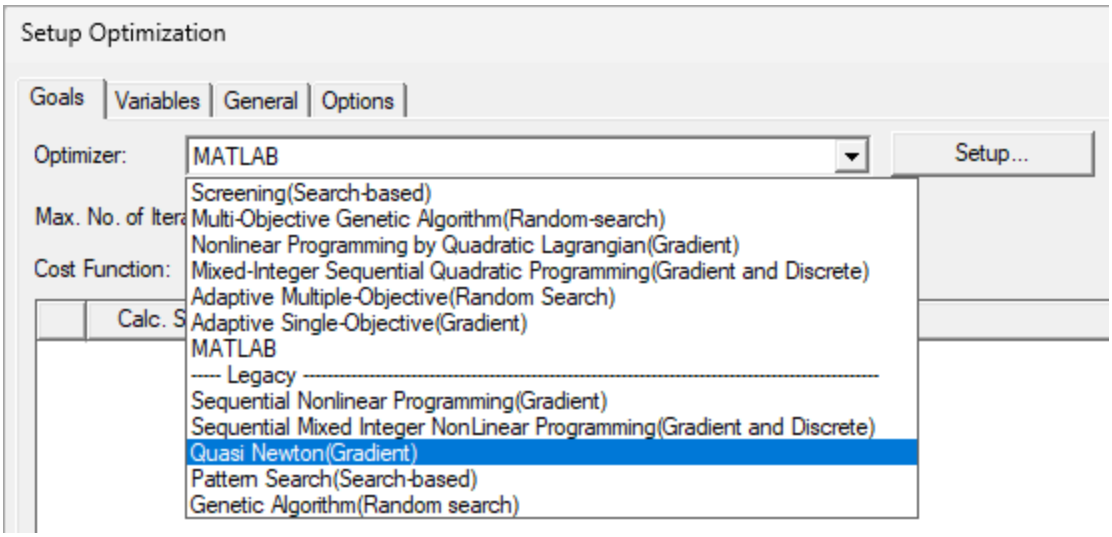
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **Icepak** > **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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

    > **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 the Pattern Search (Search-based) Optimizer

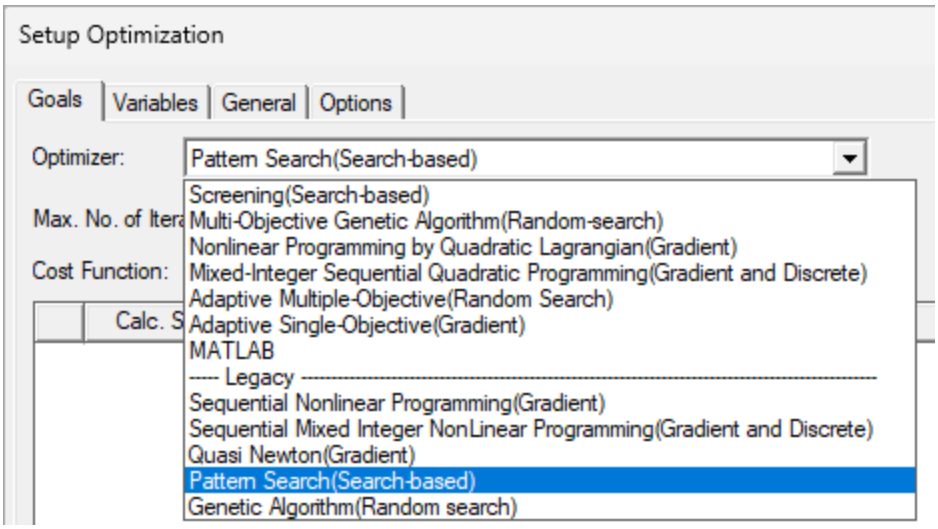
Following is the procedure for setting up an optimization analysis using the Pattern Search (Search-based) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **Icepak** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
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** drop-down menu appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

    > **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merit-Based Sequential Quadratic Programming(Gradient) Optimizer Setup

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.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Icepak** > **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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. 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.

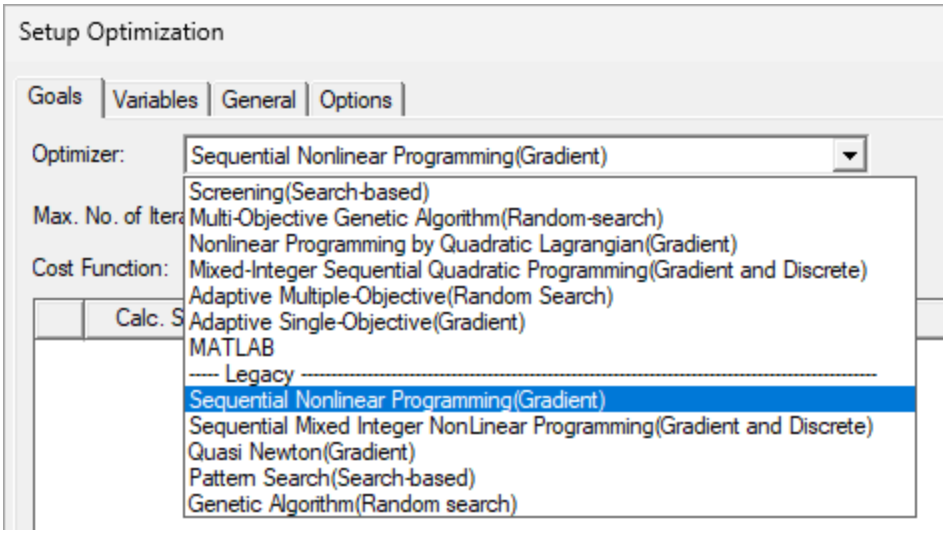
Setting Up the Sequential Nonlinear Programming (Gradient) Optimizer

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 **Icepak** > **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.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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**.

Ansys Electromagnetics Suite 2026 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.

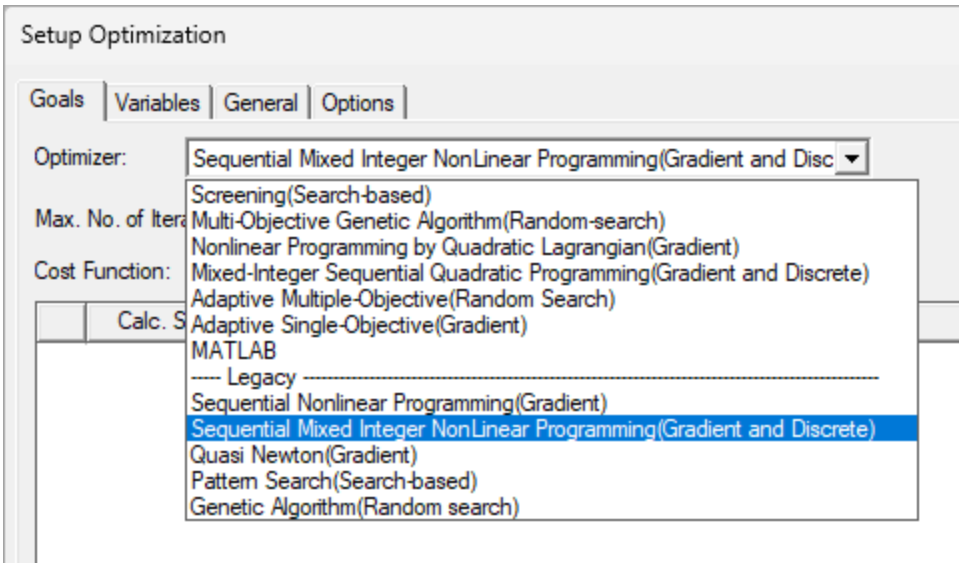
Setting Up the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer

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 **Icepak** > **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 2026 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.
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**.

Ansys Electromagnetics Suite 2026 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.

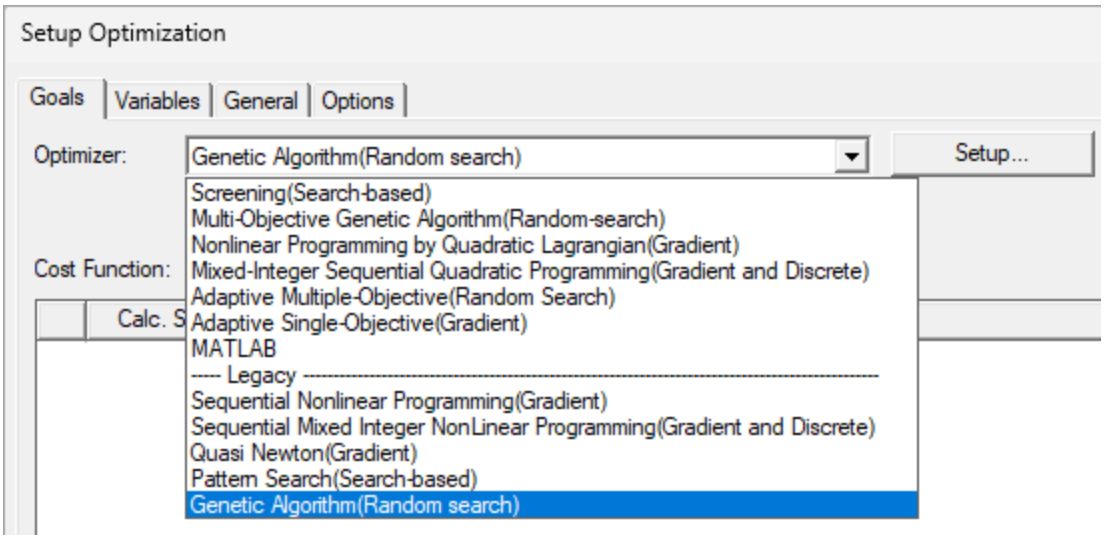
Setting Up the Genetic Algorithm (Random Search) Optimizer

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 **Icepak** > **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** drop-down list.



4. Click **Setup...** to modify the Advanced Genetic Algorithm Optimizer Options.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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** drop-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.
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**.

Ansys Electromagnetics Suite 2026 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.

Setting Up 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.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Icepak** > **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** drop-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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

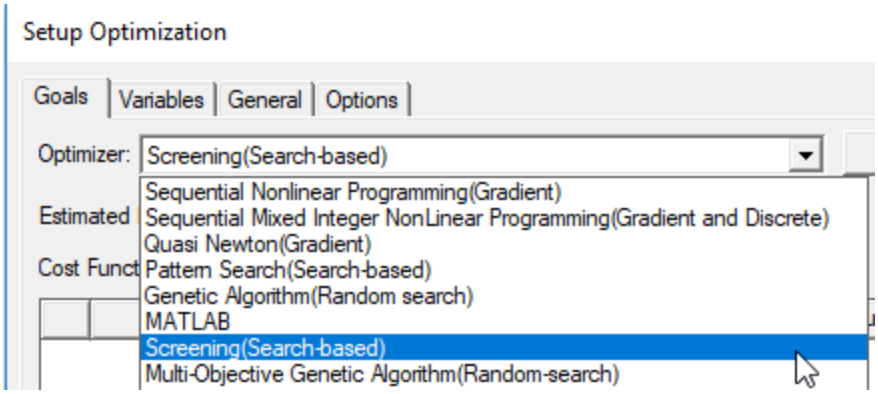
Screening (Search Based) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Screening 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.

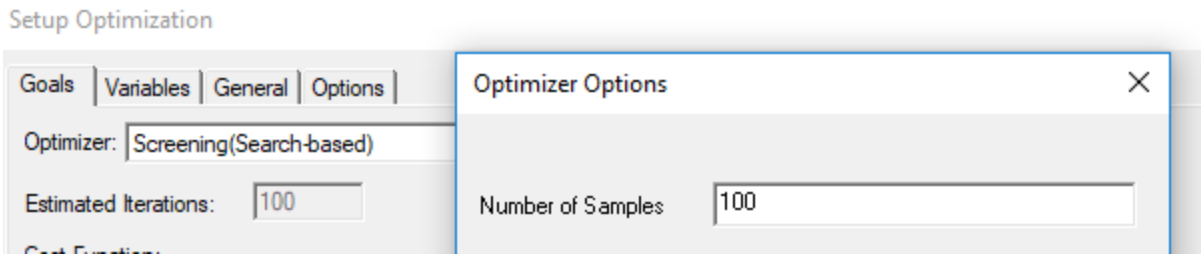
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is 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. Then, if you want to refine, these candidate points are used as starting points for gradient methods.

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 **Icepak** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* window appears.
3. Under the **Goals** tab, select the optimizer by selecting **Screening (Search Based)** from the **Optimizer** drop-down menu.



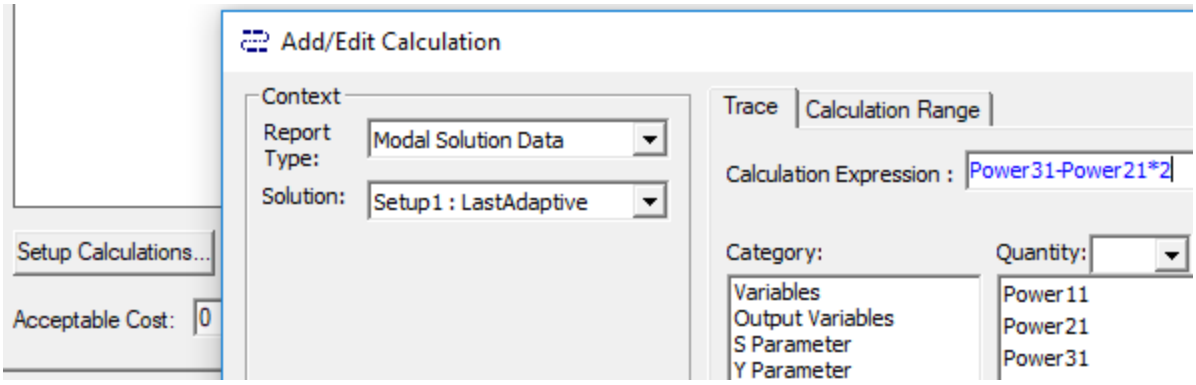
4. Optionally, press the **Setup** button to open the **Optimizer Options window** to change the default number of samples from 100.



The number of samples must be greater than the number of enabled input parameters. 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 10,000. The default is 100 for a Direct Optimization system.

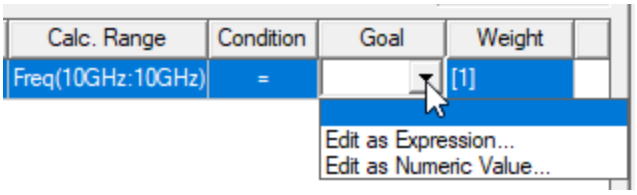
5. Add a cost function by selecting the **Setup Calculations** button to open the **Add/Edit Calculation** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

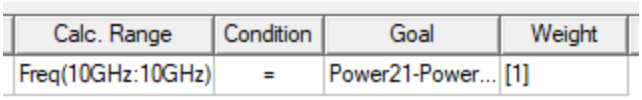


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** window.

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** window.



7. If you are satisfied with the expression or value displayed, click **Done** to close the window. 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** drop-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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Setting Up the 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

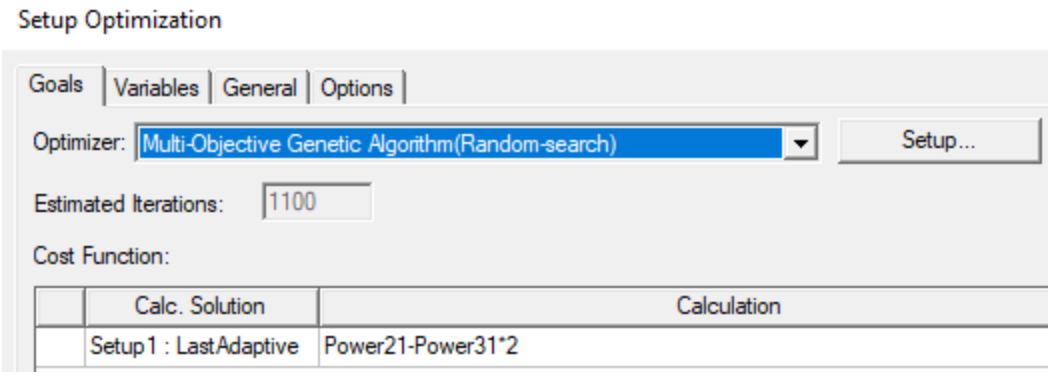
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

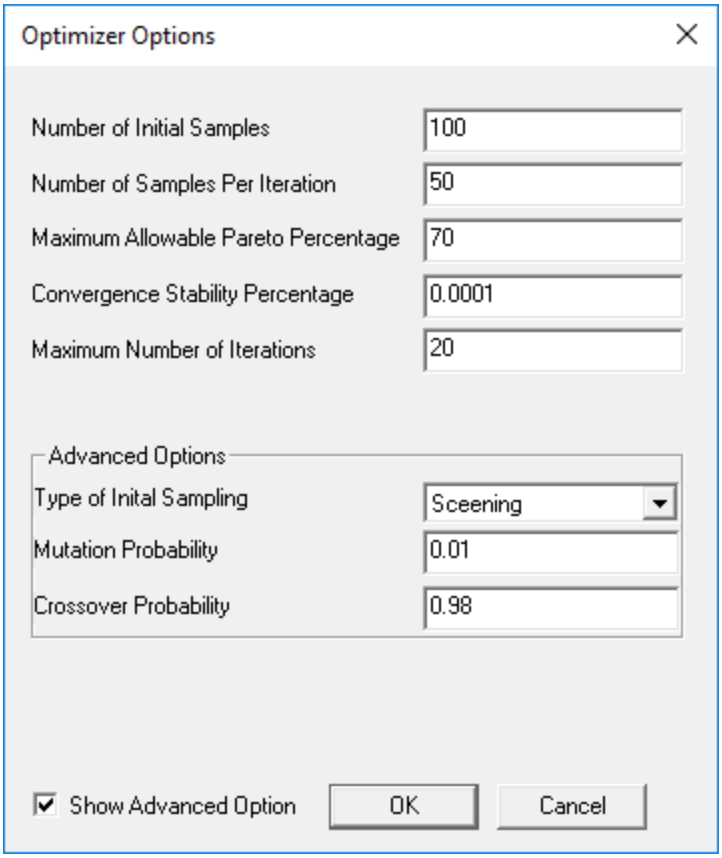
The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of 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 **Icepak** > **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 2026 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 2026 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

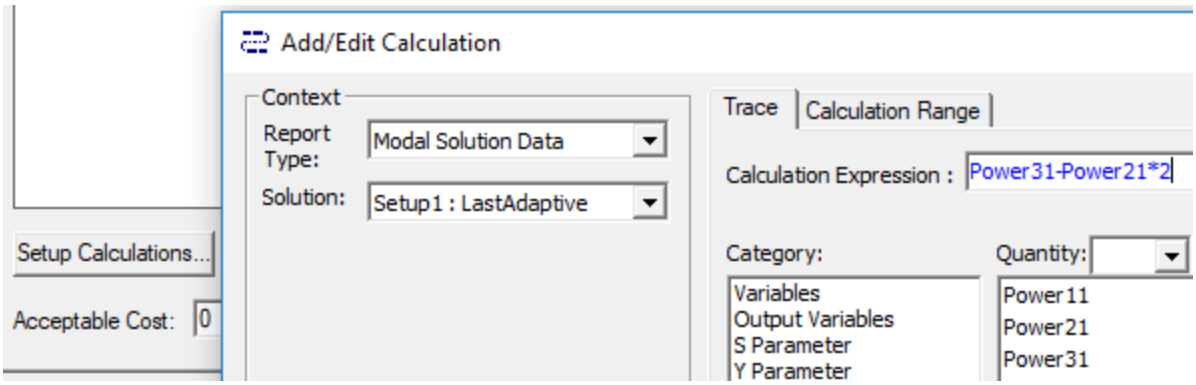
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

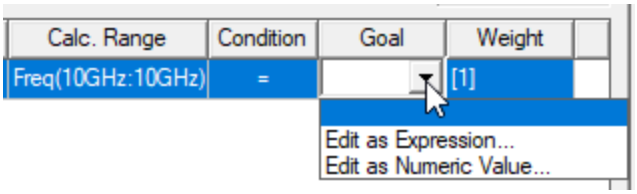
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the 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.

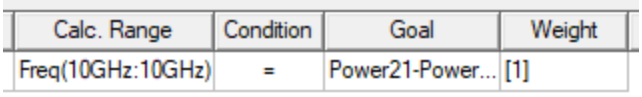
6. In the Optimization setup, in the drop-down menu for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.



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 into the **Goal** column.



Ansys Electromagnetics Suite 2026 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** drop-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 2026 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.

Setting Up the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer

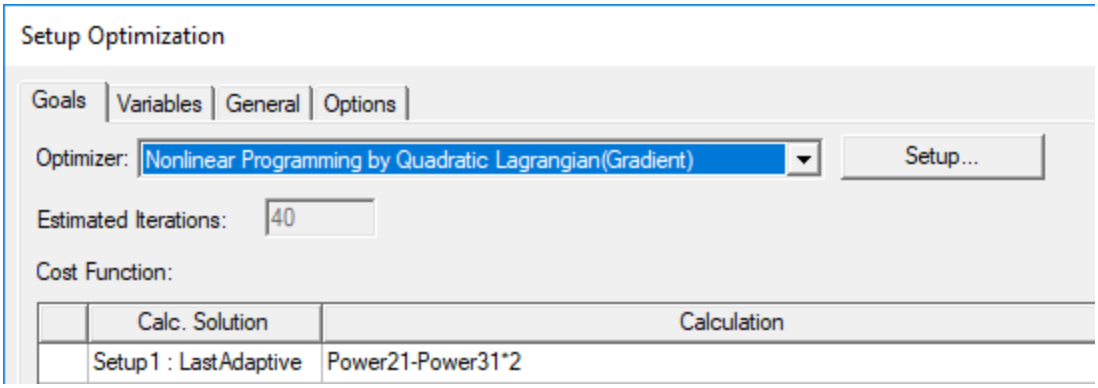
Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

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).

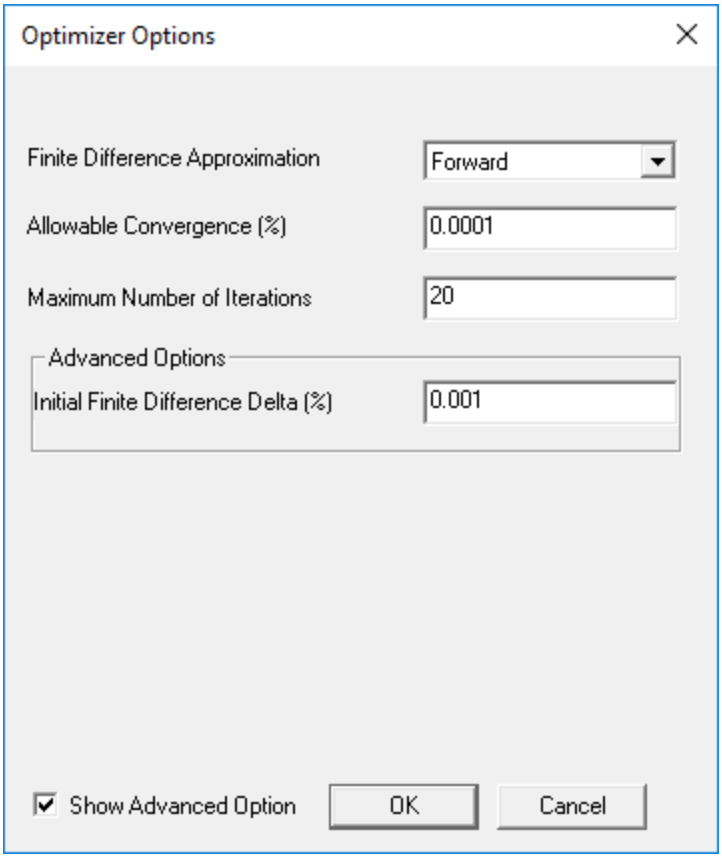
To generate samples and perform an NLPQL optimization:

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 **Icepak** > **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 2026 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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:

  For **Central**: number of iterations * (2*number of inputs +1)

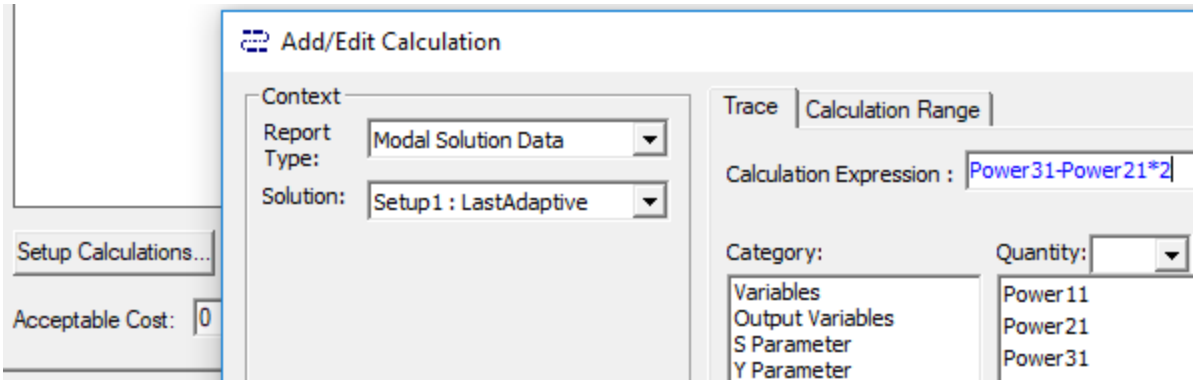
  For **Forward**: number of iterations * (number of inputs+1)

- **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.

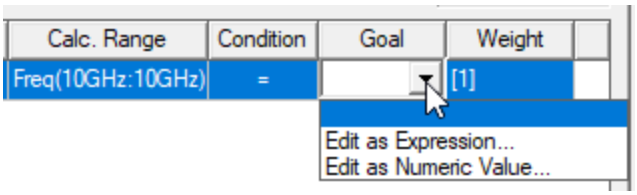
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



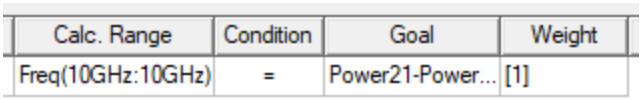
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

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** drop-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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)

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

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$

$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

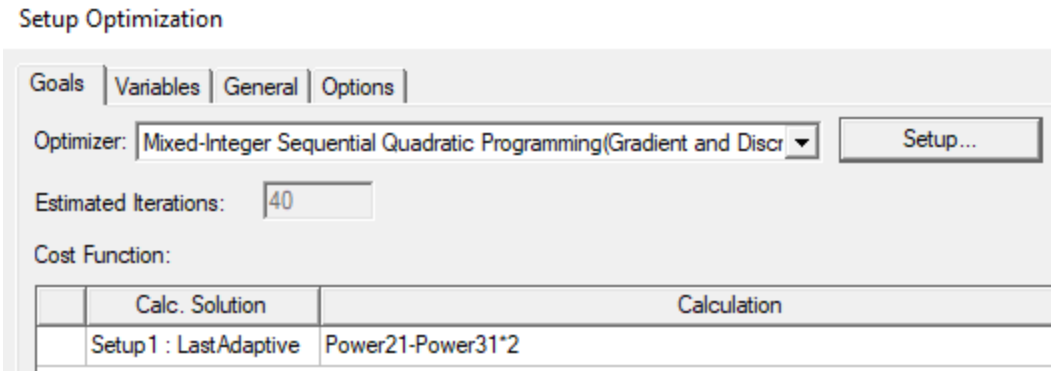
The Mixed-Integer Sequential Quadratic Programming (Nonlinear Programming by Quadratic Lagrangian) method (MISQP)can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. MISQP is available for both continuous 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:

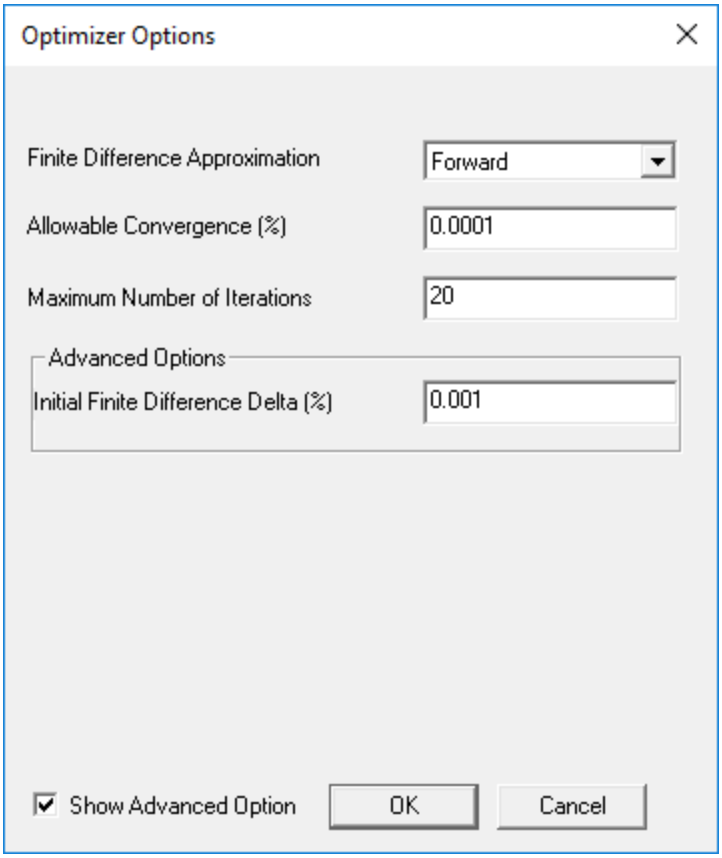
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Icepak** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu.



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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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 Approximation gradient calculation method, as follows:

  For **Central**: number of iterations * (2*number of inputs +1)

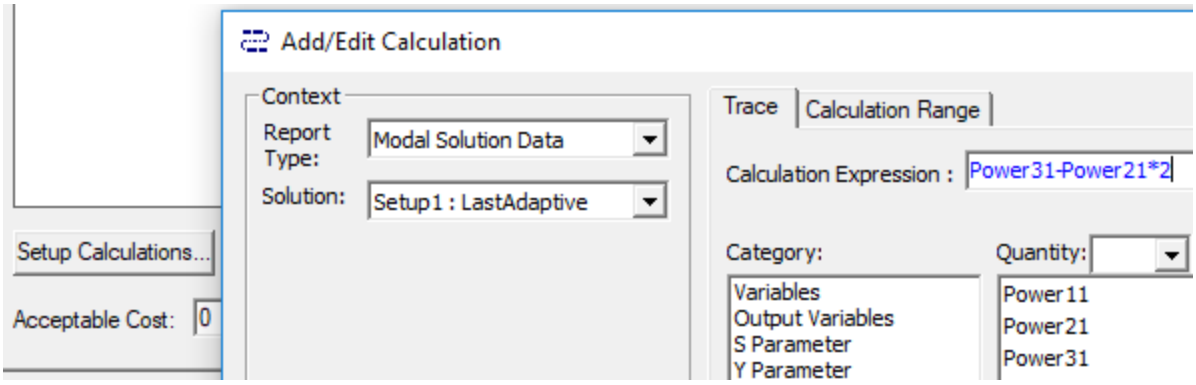
  For **Forward**: number of iterations * (number of inputs+1)

- **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.

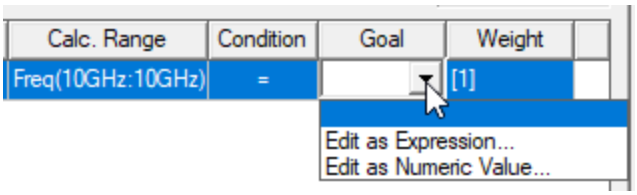
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



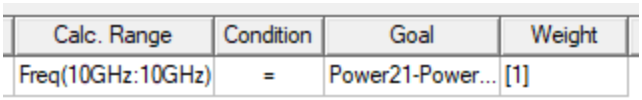
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

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** drop-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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

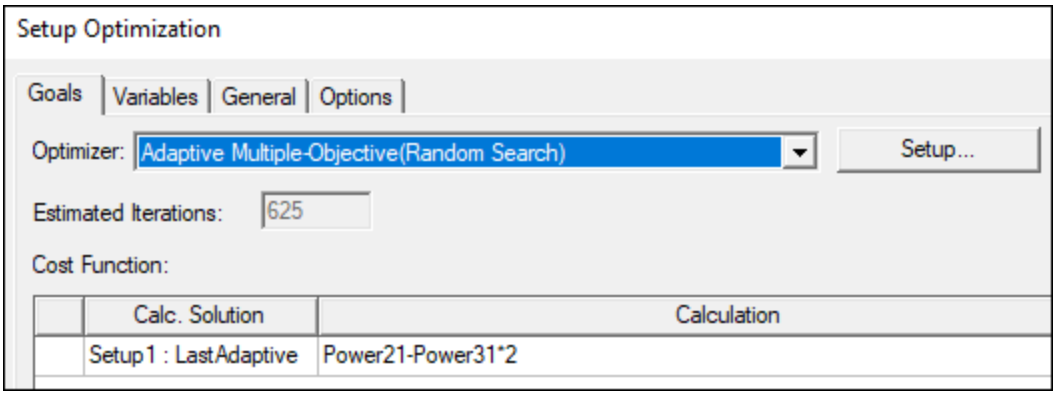
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adaptive Multiple-Objective (Random Search) Optimizer Setup

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.

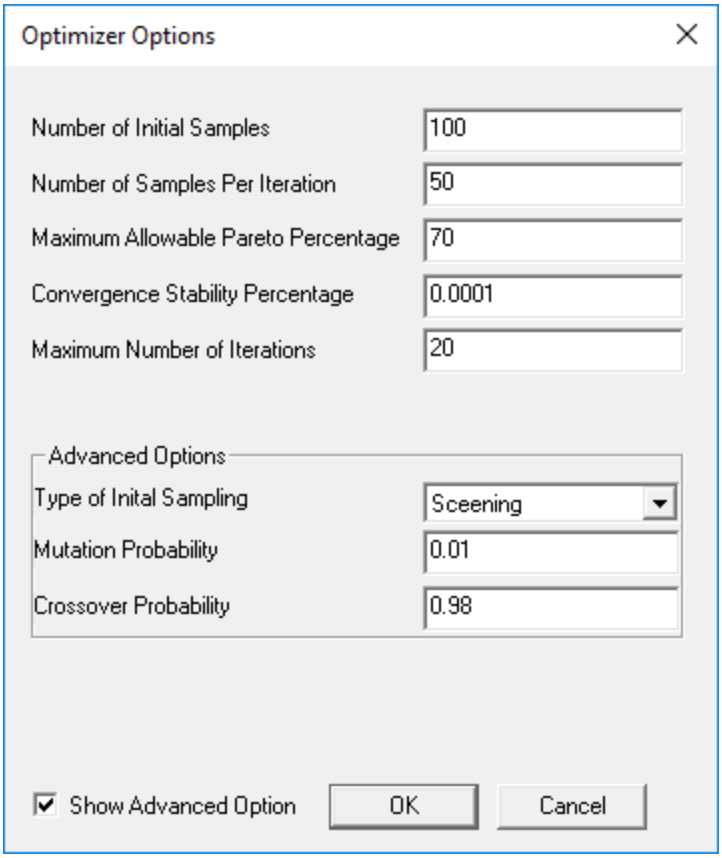
The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce 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 **Icepak** > **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 2026 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 2026 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:

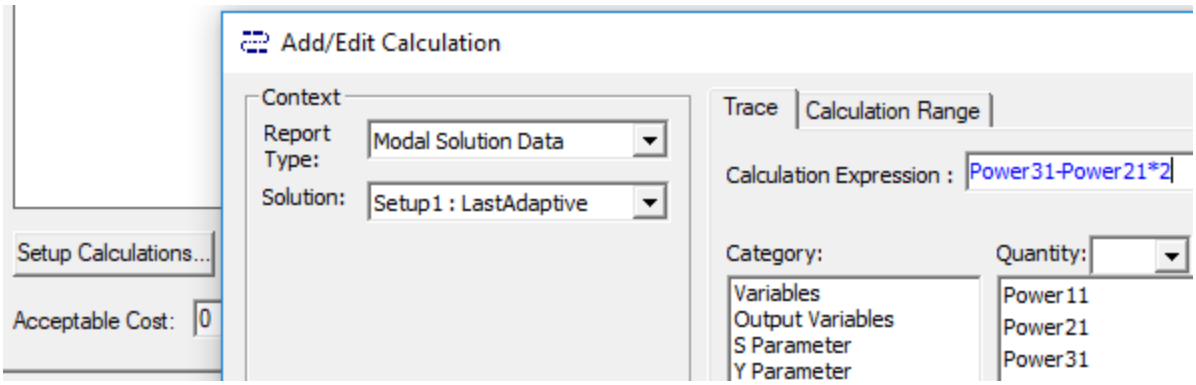
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

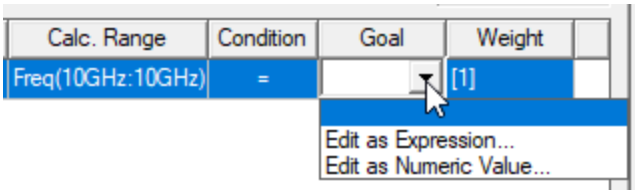
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the 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.

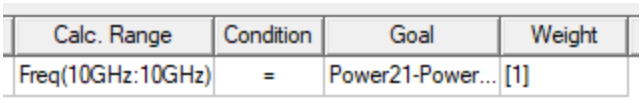


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

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 2026 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* drop-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 2026 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.

Adaptive Single-Objective (Gradient) Optimizer Setup

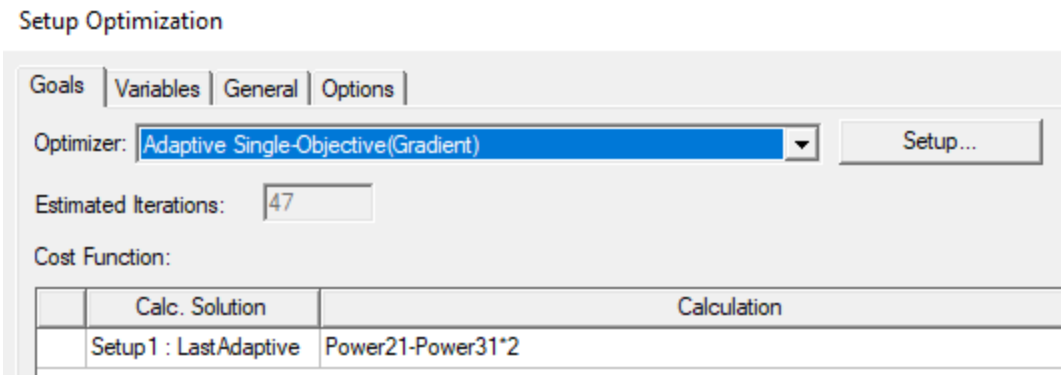
Following is the procedure for setting up an optimization analysis using the Adaptive Single-Objective (OSF + Kriging + MISQP) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

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.

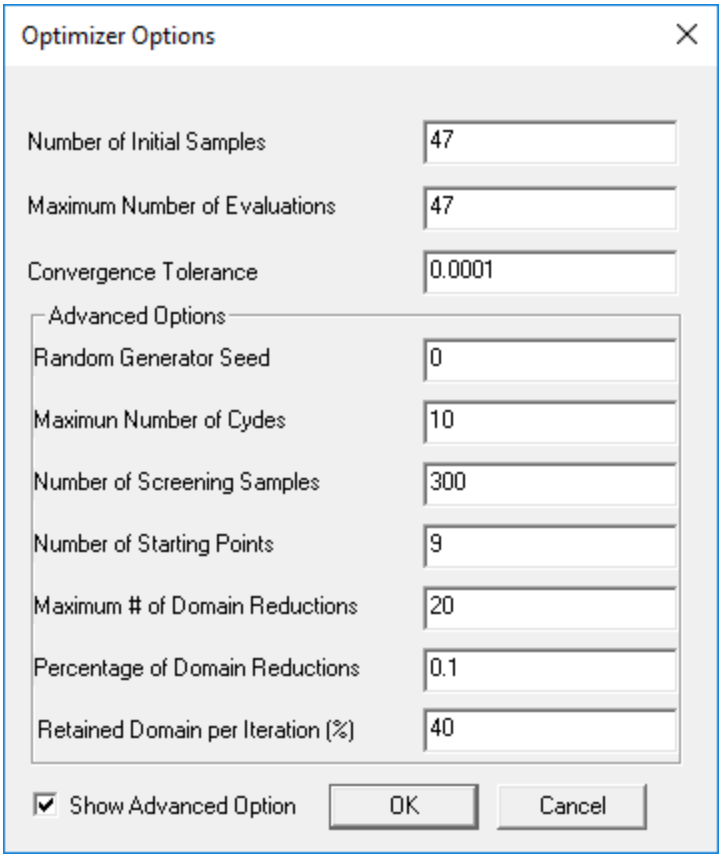
The Adaptive Single-Objective method is available for input parameters that are continuous, including those with manufacturable values. It can handle only one output parameter goal, although other output parameters can be defined as constraints. It does not support the use of parameter relationships 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 **Icepak** > **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.

Ansys Electromagnetics Suite 2026 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 and check **Advanced Options**.



- **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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

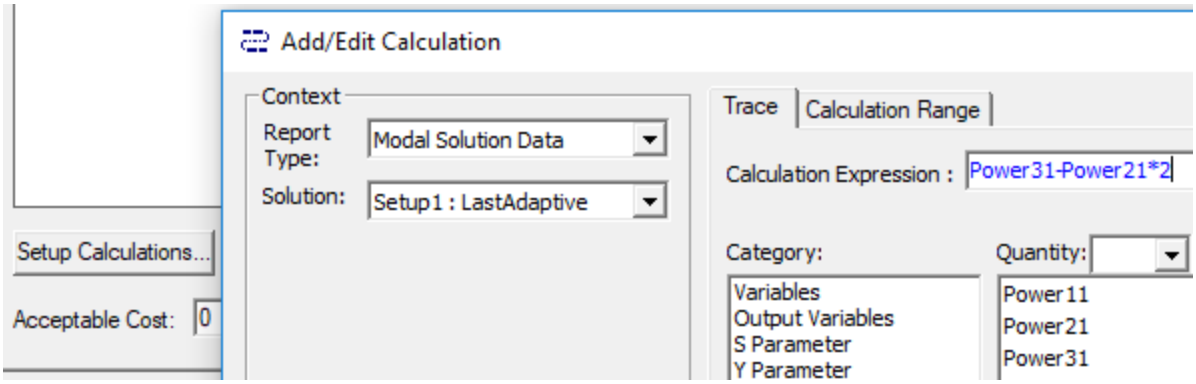
- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

samples are selected in this sample. The default is the value for **Number of Initial Samples**.

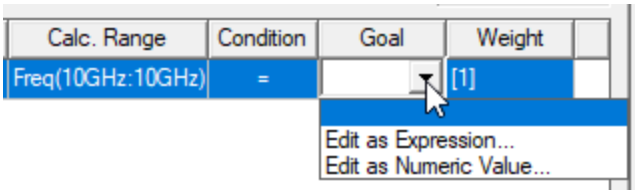
- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according 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 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.



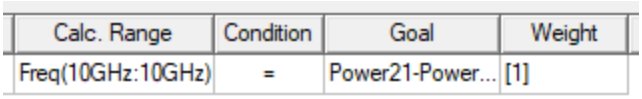
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



Calc. Range	Condition	Goal	Weight
Freq(10GHz:10GHz)	=	Power21-Power...	[1]

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** drop-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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

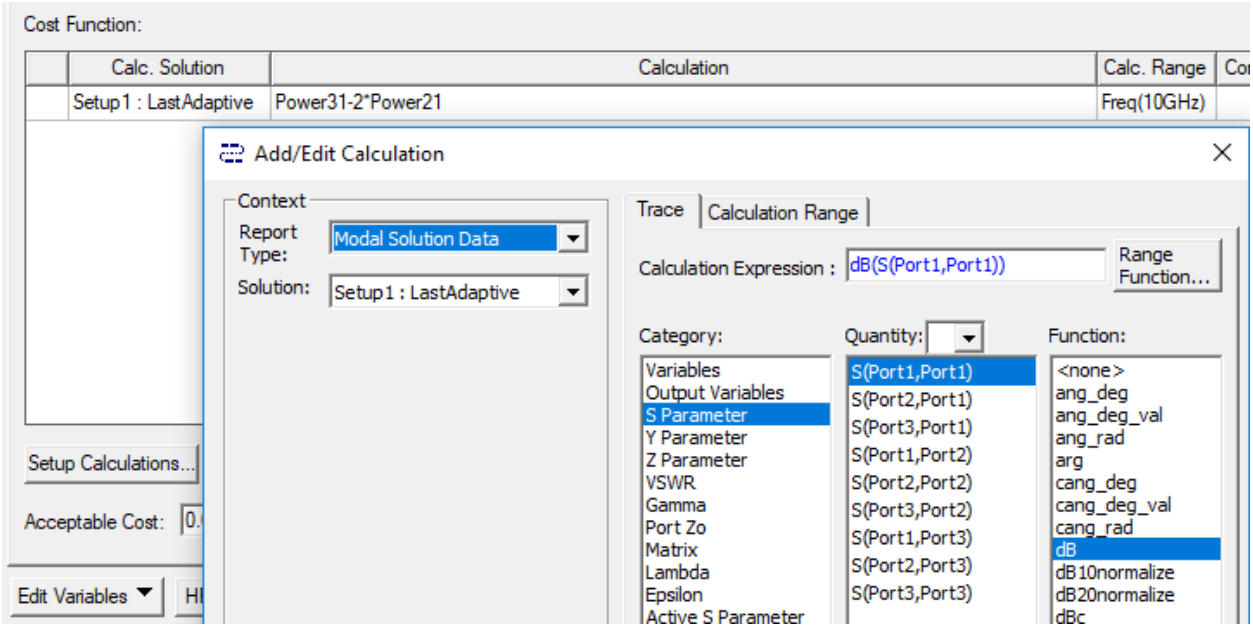
Cost Function

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 2026 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 2026 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.

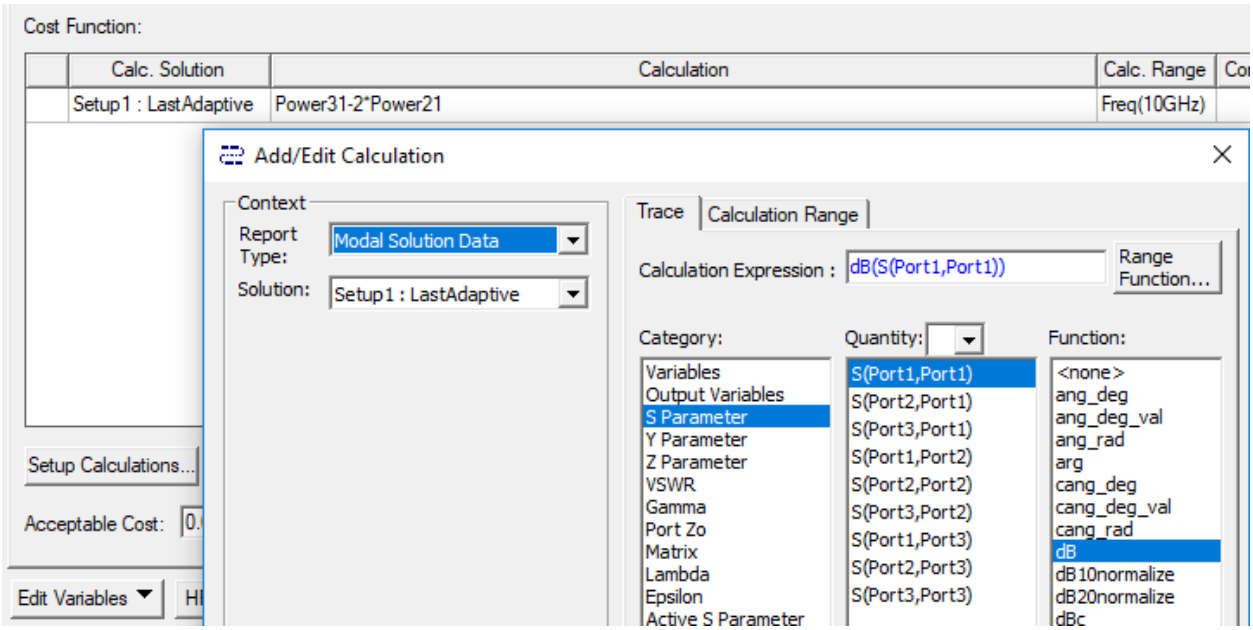
Adding a Cost Function

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 2026 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 2026 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 2026 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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

$$\sum_{j}^{G} \frac{W_j}{N_j} \cdot \sum_{i}^{N_j} e_i$$

where

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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:

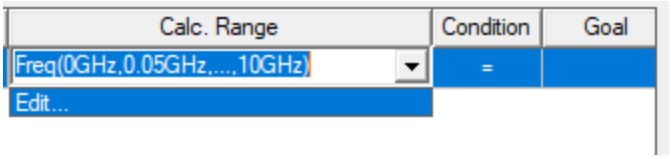
Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any Optimetrics analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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, Freq(0GHz....0GHz)

  min/max range, for example, Freq([0GHz, 10GHz])

- editable sweep, which allows you to customize values (that is a sweep that has an enabled "edited" radio button in sweep selection dialog):

  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

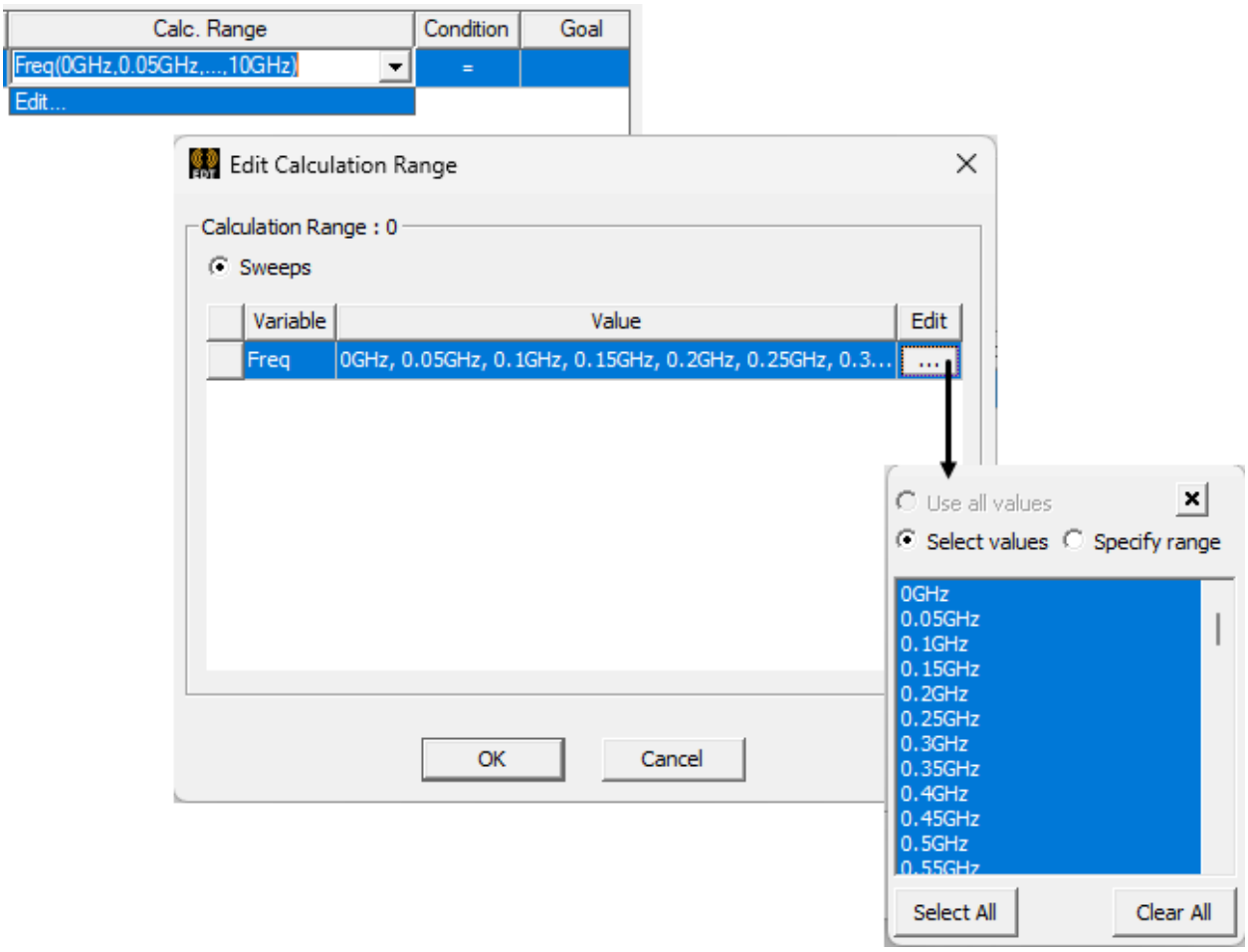
- You solve 0 to 10 GHz with a step value of 0.05.
- You can specify multiple sweep values by separating those with comma ","

  For example, F(1GHz), cap(1pf, 1.2pf)

  For example, Distance(All), Freq([1ghz,2ghz]), Phase(0 deg)

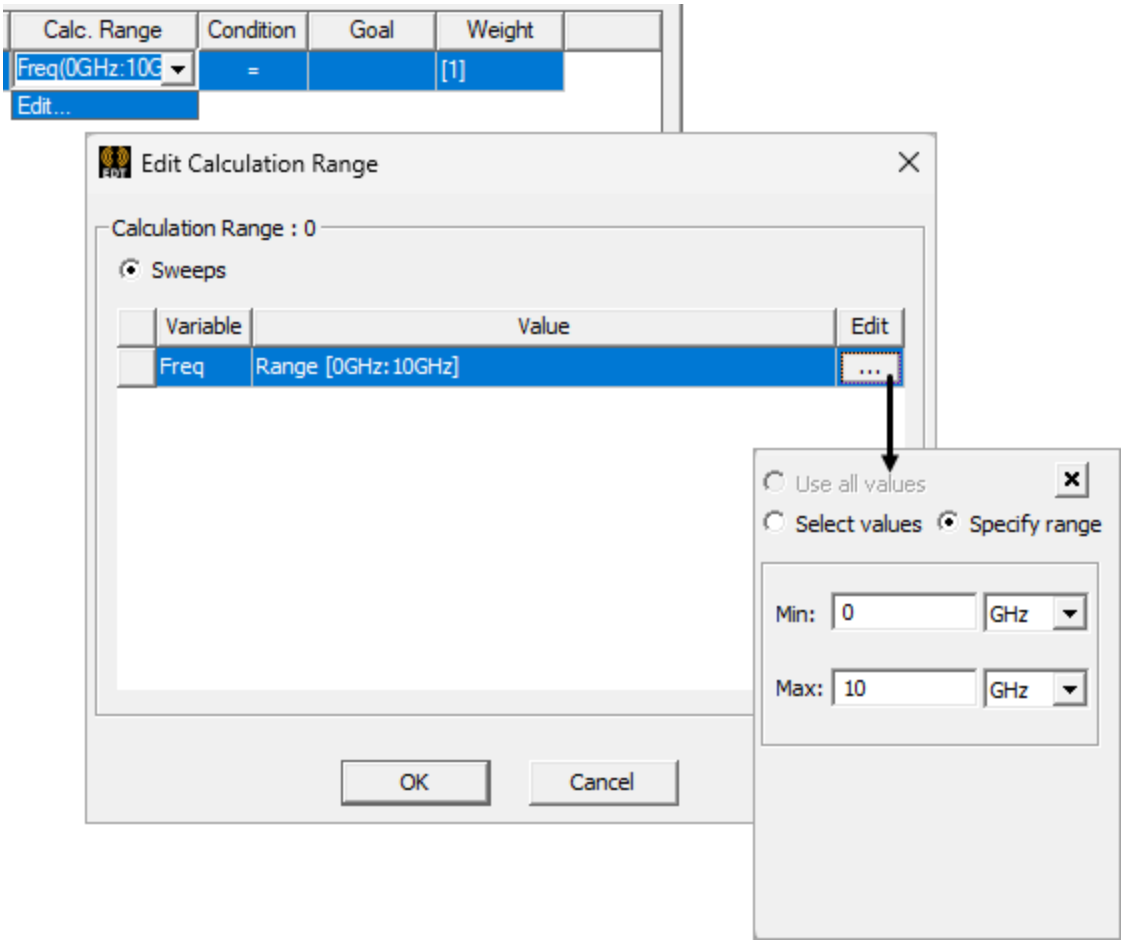
If you click **Edit..** on the menu, you see the **Edit Calculation Range** dialog box. Click on the ellipsis [...] button to select radio buttons for Use all values, Select Values or Specify range.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This example shows that when you specify a range, how the range appears in the Calc. Range field.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 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.

To revert to the default minimum and maximum values, clear the **Override** option.

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 **Icepak**> **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:

$$\boldsymbol{s_i} = \boldsymbol{x_{i+1}} - \boldsymbol{x_i}$$

The natural distance measure is

$$\|\boldsymbol{s_i}\| = \sqrt{\boldsymbol{s_i^T}\,\boldsymbol{s_i}}$$

which is the Euclidean norm.

Ansys Electromagnetics Suite 2026 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.

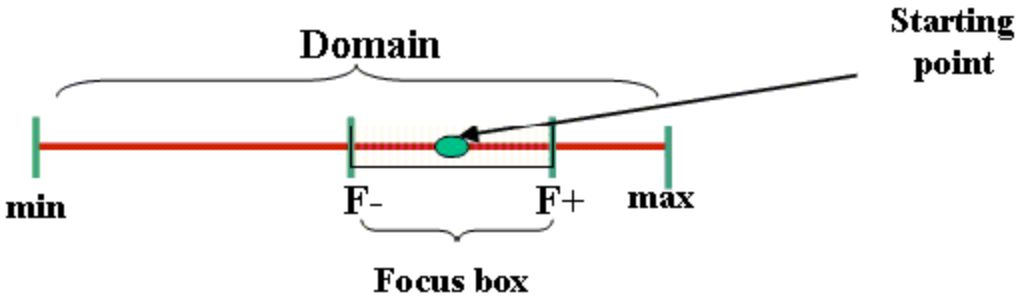
1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min Step** and **Max Step** columns, with default values for each variable to be optimized.
3. In the **Min Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** 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.
5. Click **OK**.

> **Tip:**
>
> A value of zero is recommended for the minimum step size.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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**.
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:

$Calculation_{(i)}\ Condition_{(i)}\ Goal_{(i)}\ Weight_{(i)}$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

$$Cost = \sum_{1}^{N} w_i \varepsilon_i^2$$

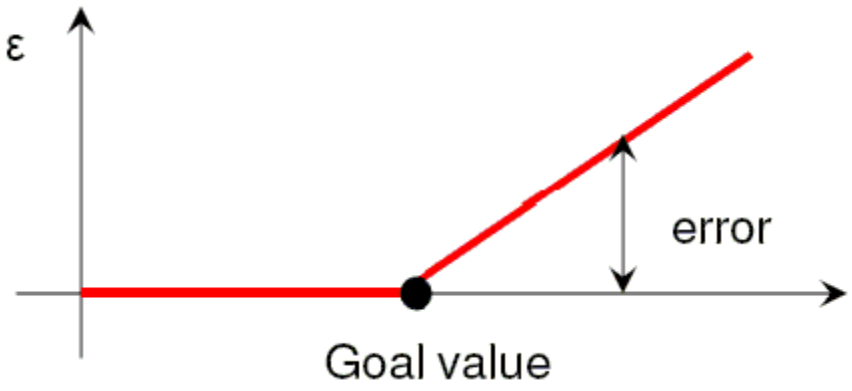
For the **Maximum** norm, the cost function uses the maximum among all the weighted goal errors, which means that cost is always less than zero:

$$Cost = \underset{1}{\overset{N}{Max}} W_i \cdot \varepsilon_i$$

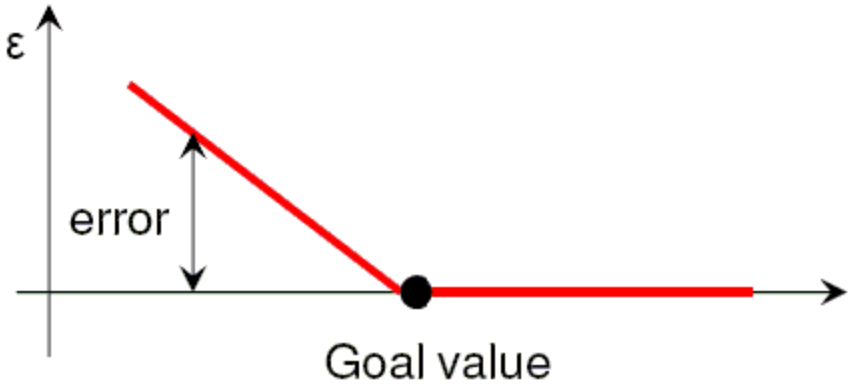
For all the above situations, N is the number of individual goals $w_i \varepsilon_i$ are individual weighting factors and residual error respectively. A minimization of the cost function is performed during optimization since it makes sense to minimize the error in the sense of the chosen norm type.

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:

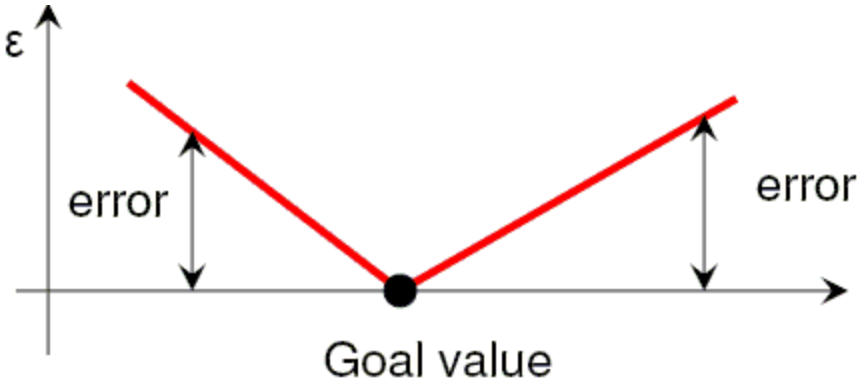
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:



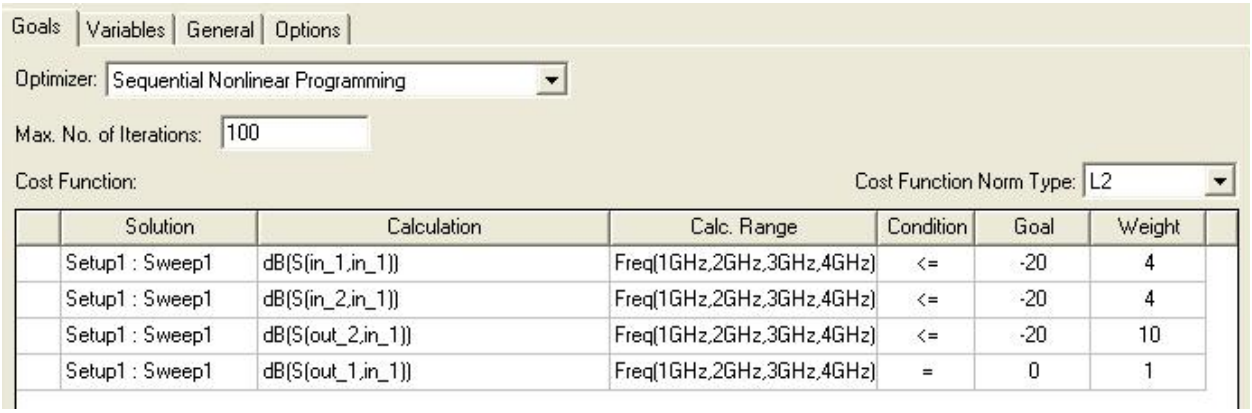
The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios. Note that when using "minimize" or "maximize" settings for the condition there should be a single goal setting which in this case coincides with the cost function.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



	Solution	Calculation	Calc. Range	Condition	Goal	Weight
	Setup1 : Sweep1	dB(S(in_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(in_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(out_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	10
	Setup1 : Sweep1	dB(S(out_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	=	0	1

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 probability Variable crossover probability
One point	Individual crossover probability
Two point	Individual crossover probability
Simulated binary crossover	Individual crossover probability Variable crossover

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

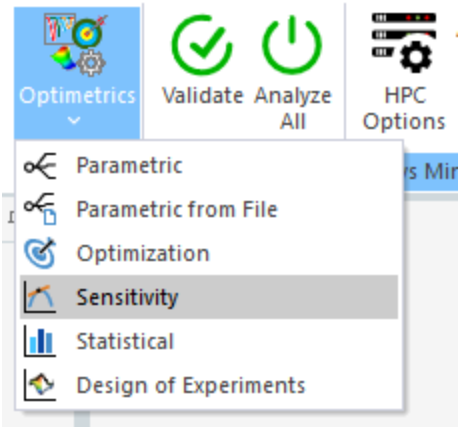
During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **Icepak** > **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:



   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**.

Ansys Electromagnetics Suite 2026 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.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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:

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   - 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 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.

   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 **Icepak**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.
3. Select **Sensitivity**.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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:

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:**

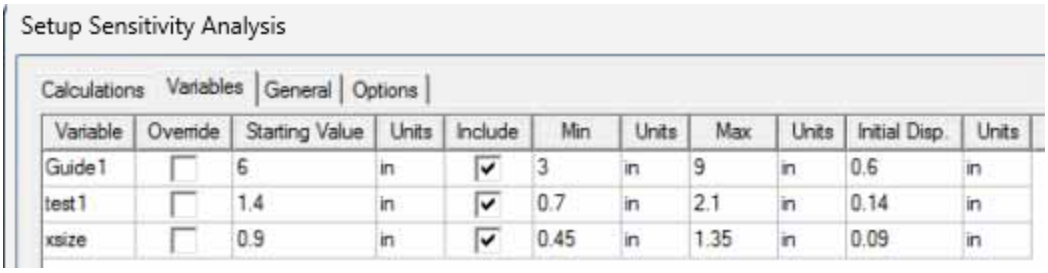
   This is one of the most popular methods to estimate worst-case performance. To start, a sensitivity analysis is performed. The results (sensitivities/first derivative) allow us to pick an extreme value (upper or lower bound) for each variable. The corresponding simulation result is used to predict upper 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**.

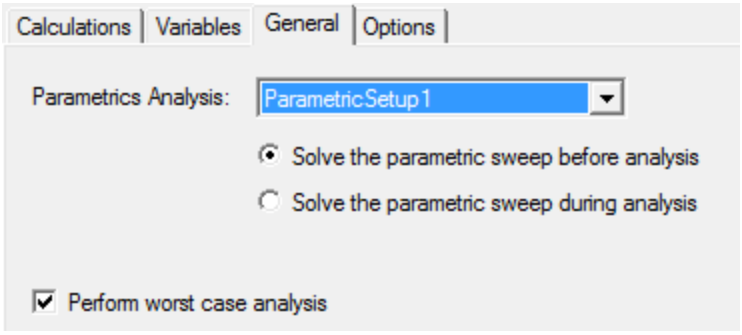
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.



> **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

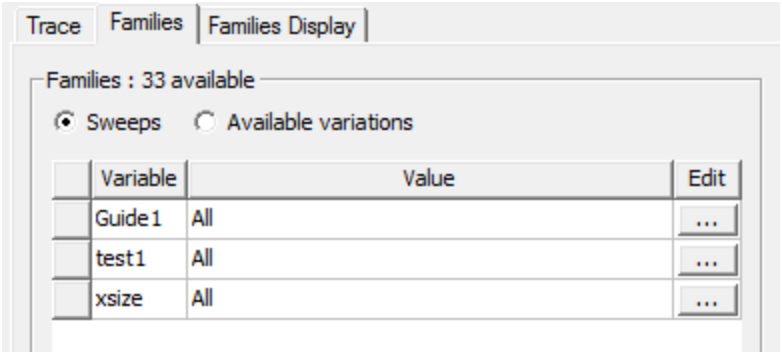
- After setup, analyze your Sensitivity Setup under Optimetrics.
- To view worst-case result (upper bound only):

  Create a Data Table report.

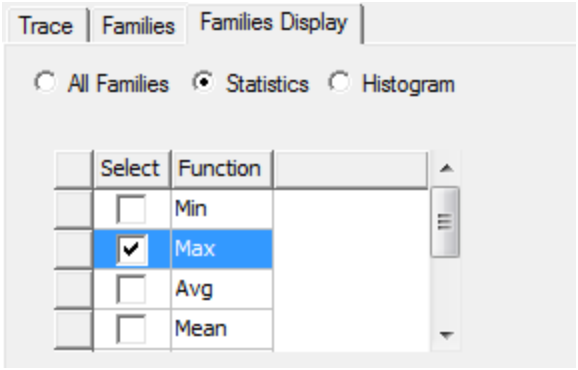
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



Under the **Families Display** tab, select **Statistics**and then check **Max**.



The associated data table shows the Max values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

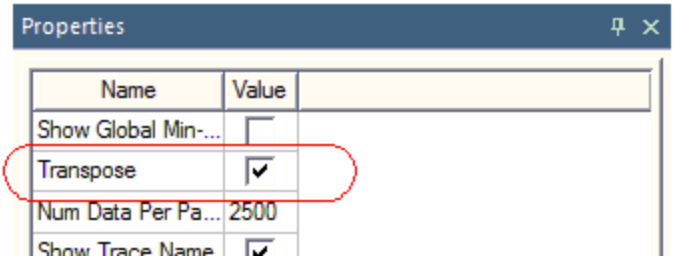
	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Max
1	3.000000	-0.000000
2	4.000000	-0.000000
3	5.000000	-0.105323
4	6.000000	-0.248322
5	7.000000	-0.267220
6	8.000000	-0.202039
7	9.000000	-0.298840

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

Tip:

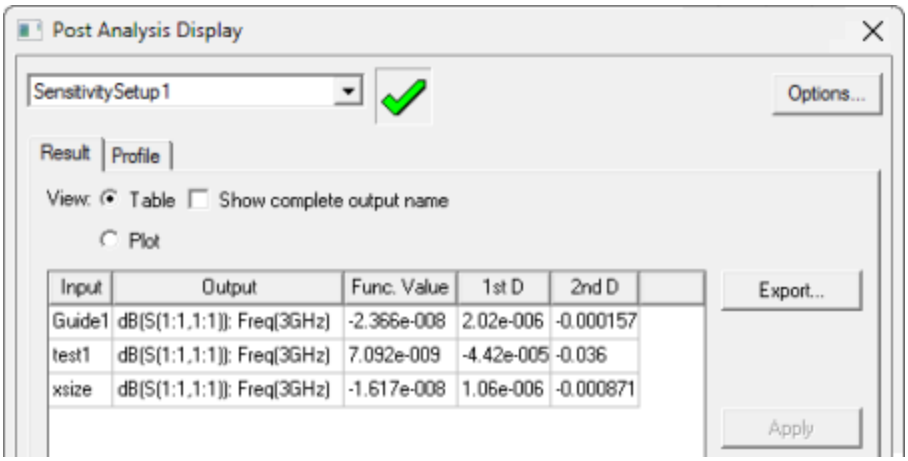
Transpose the table for an alternate view (double-click data table to view the **Properties** window, and on the **Data Table** tab check **Transpose**).



To estimate lower bound of performance:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> A statistical analysis is limited to using a maximum of thirty 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**.

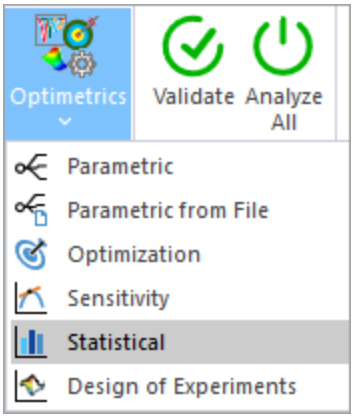
> **Note:**
>
> Statistical Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

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 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 **Icepak** > **Optimetrics Analysis** > **Statistical** , or select the **Simulation** tab of the ribbon and, under the Optimetrics icon, select **Statistical** from the drop-down menu:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Statistical Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations you want Icepak 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.
   - 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For swept variables, the Range Function button opens the **Set RangebFunction** dialog to apply functions to the expression that apply over the 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.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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**, Enter a tolerance value in the text box.

   The design variations are solved within the tolerance range of the starting value, using an even distribution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 drop-down menu.
    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 drop-down menu.
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 **Icepak** > **Design Properties**, select **Statistics**.

    If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

    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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

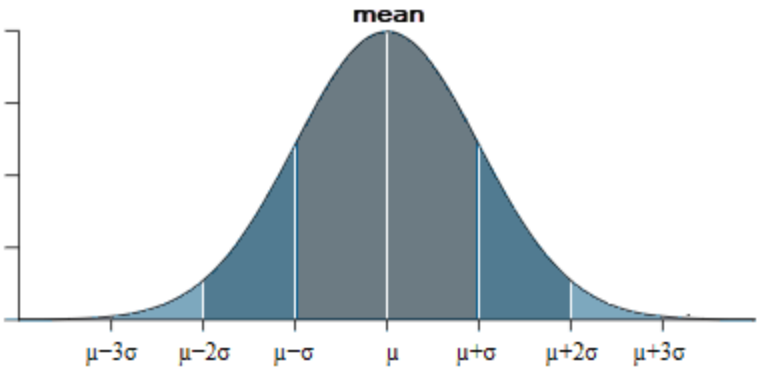
7. If the distribution type is **Uniform**, do the following:
    a. Type a cutoff probability value in the **Cutoff Probability** text box.
    b. Type mean and tolerance values in the corresponding text boxes.

    The design variations are solved within the tolerance range of the starting value, using an even distribution.

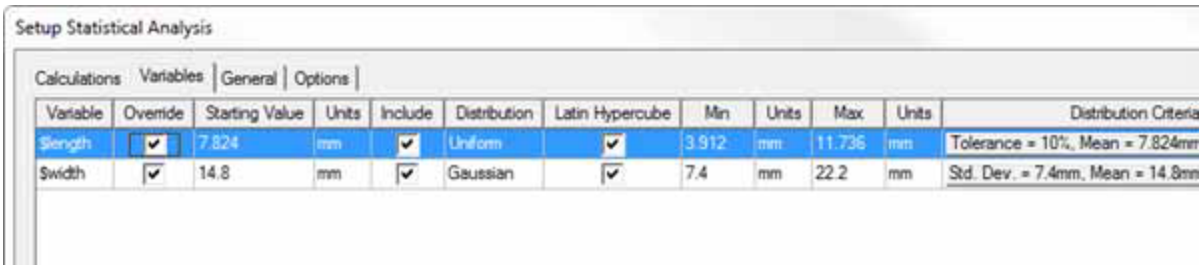
8. If the distribution type is **Lognormal**, do the following:
    a. Type a cutoff probability value in the **Cutoff Probability** text box.
    b. Type values for Sigma, M, and Theta in the corresponding text boxes.
9. If the distribution type is **User Defined**, do the following:
    a. Type a cutoff 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



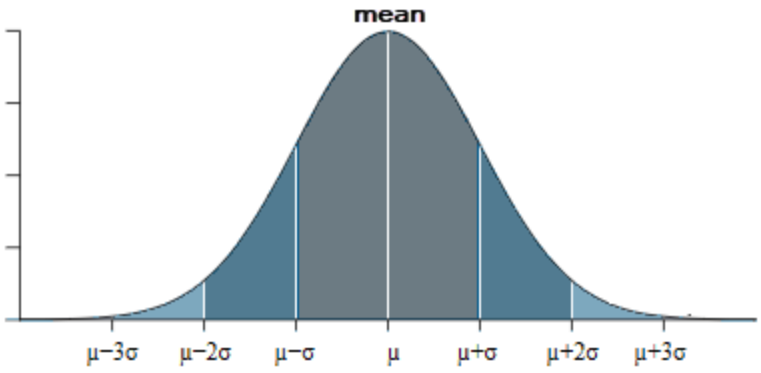
Uniform distributions such as **length** variable above use only the **Tolerance** value and do not have a cutoff probability.

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.

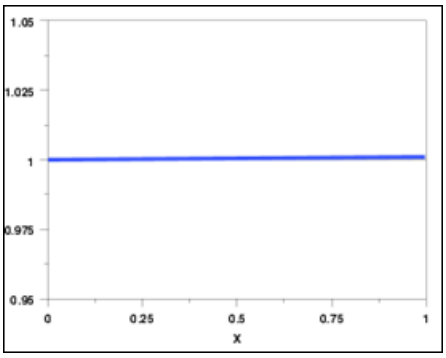
   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.



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.

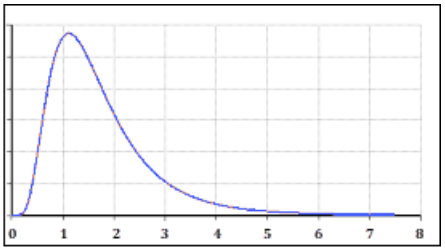
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

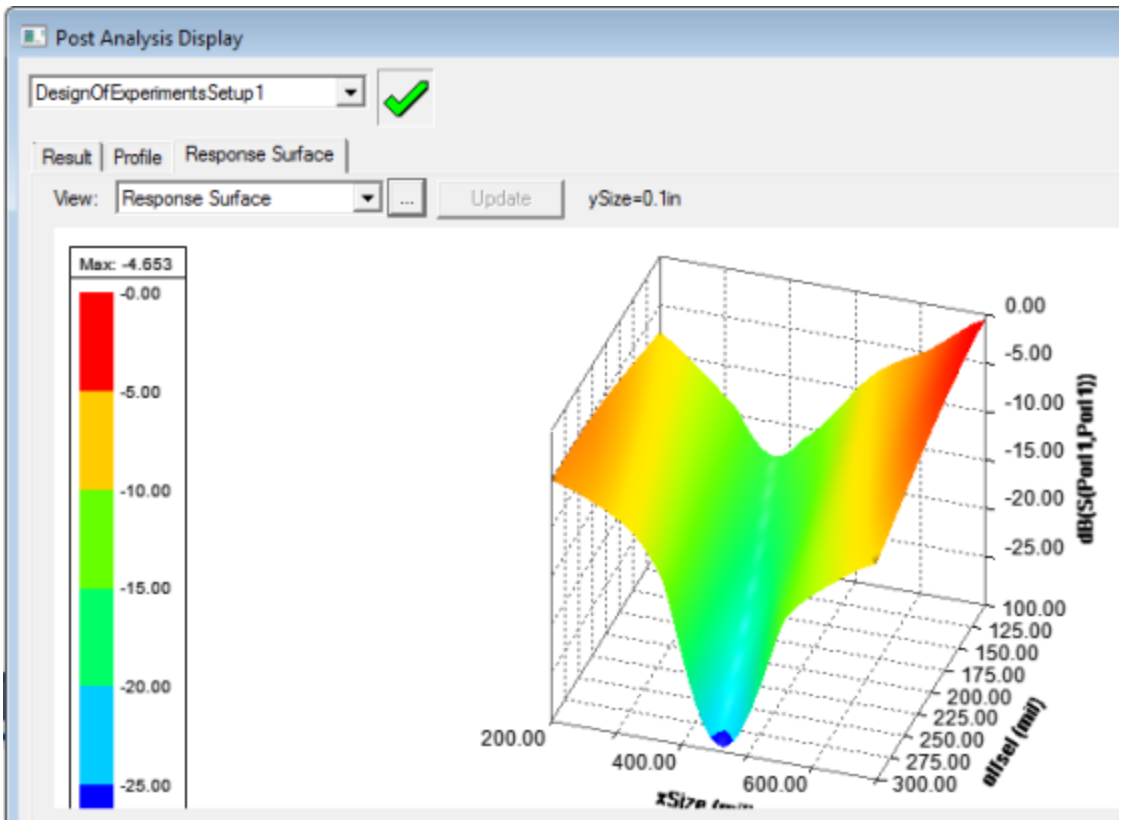
Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems. Design of Experiments plus a mathematical approximation of output parameters lets you

- reduce the number of simulations
- interactively explore the design space before running optimization

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

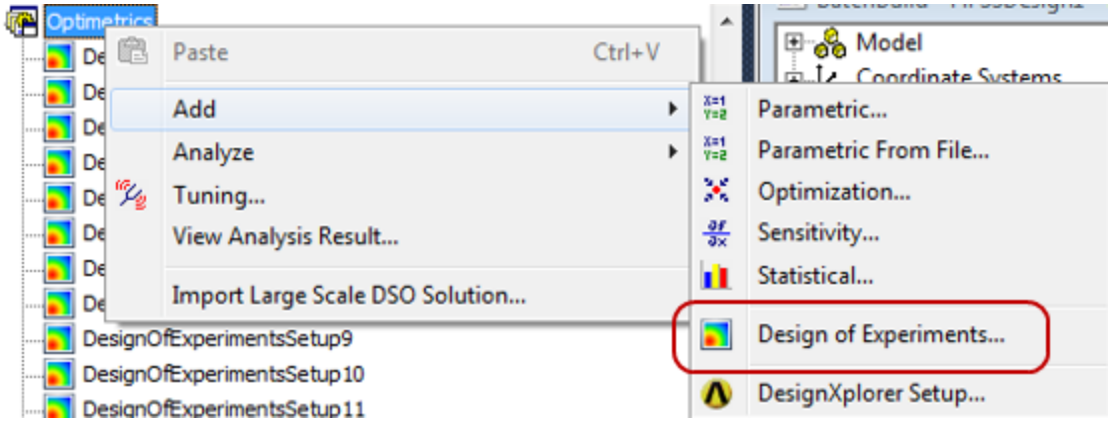
Design of Experiments describes the relationship between the design variables and the performance of the product by using Design of Experiments (DOE), combined with response surfaces. DOE and response surfaces provide all of the information required to achieve Simulation Driven Product Development. Once 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.

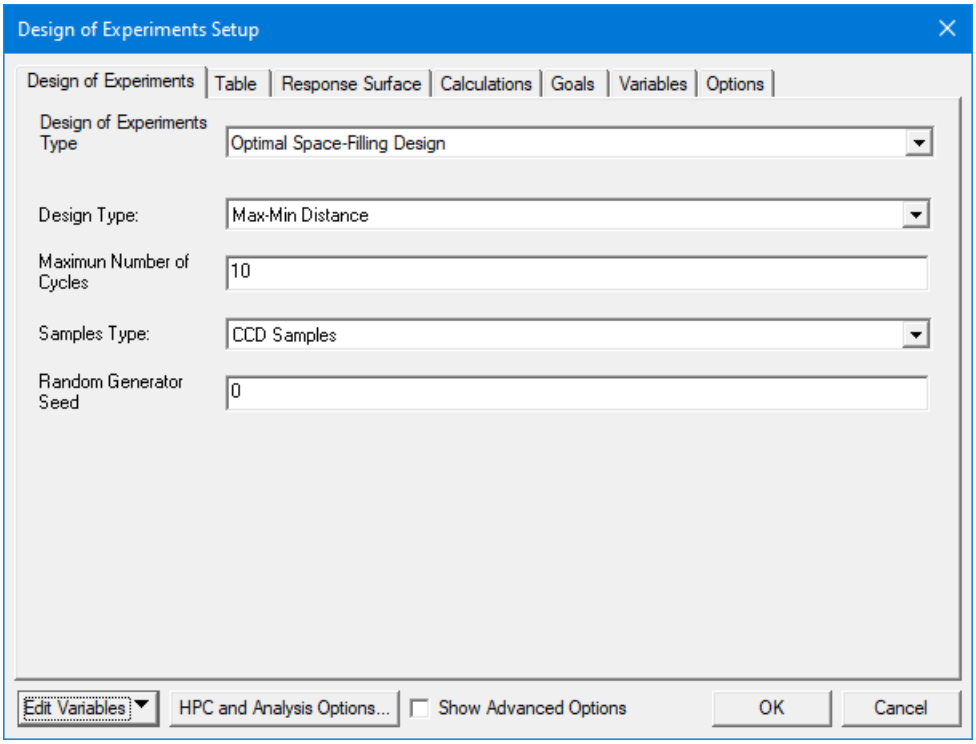


The Design of Experiments feature is integrated inside Electronics Desktop. Combined with Electronics Desktop‘s distributed solve feature, you can build the response surfaces from the DOE variation table much faster.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting **Design of Experiments** under **Optimization** opens a window with several tabs:



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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.

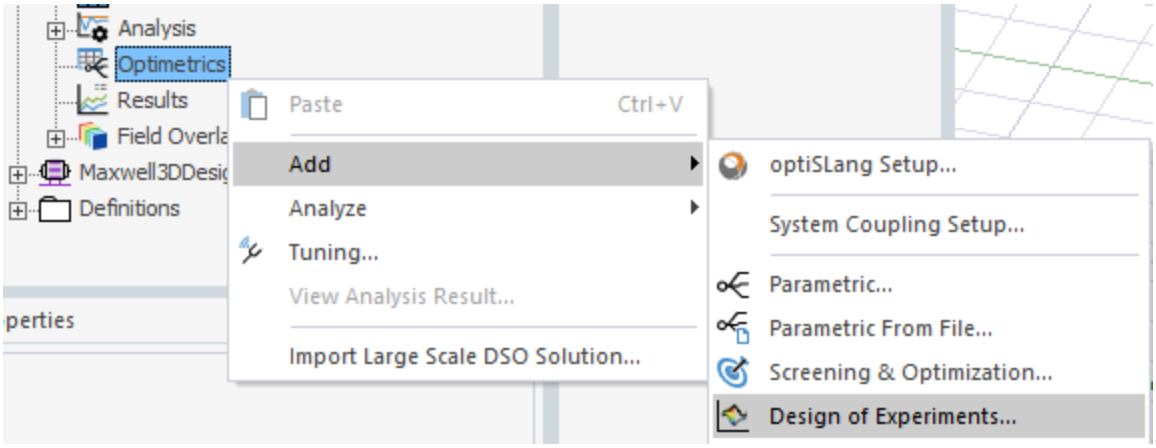
Setting Up Design of Experiments in Ansys Electronics Desktop

The process for setting up a Design of Experiments analysis in Ansys Electronics Desktop follows:

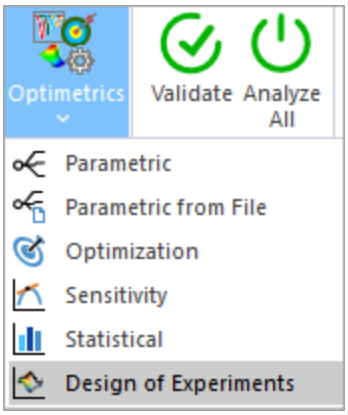
Add the Design of Experiments Setup

Right-click the Optimetrics icon in the Project tree and select **Add** > **Design of Experiments**. You can also use **Icepak** > **Optimetrics Analysis** > **Add Design of Experiments**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

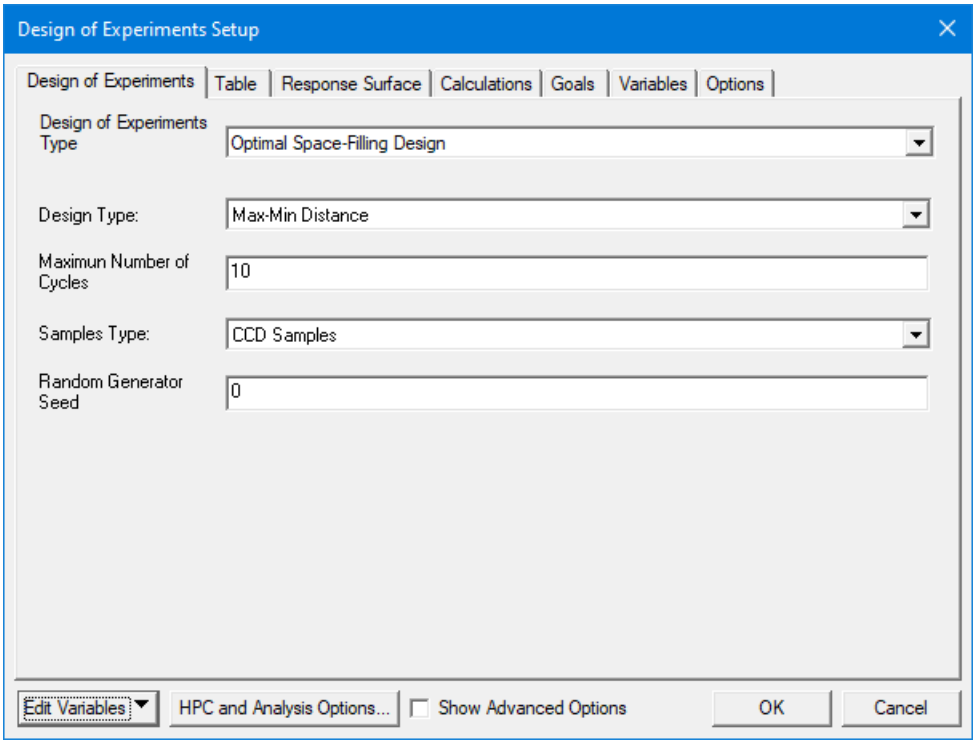


You can also select the **Simulation** tab and under the Optimetrics icon, select Design of Experiments from the drop-down menu:



The **Design of Experiments Setup** window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

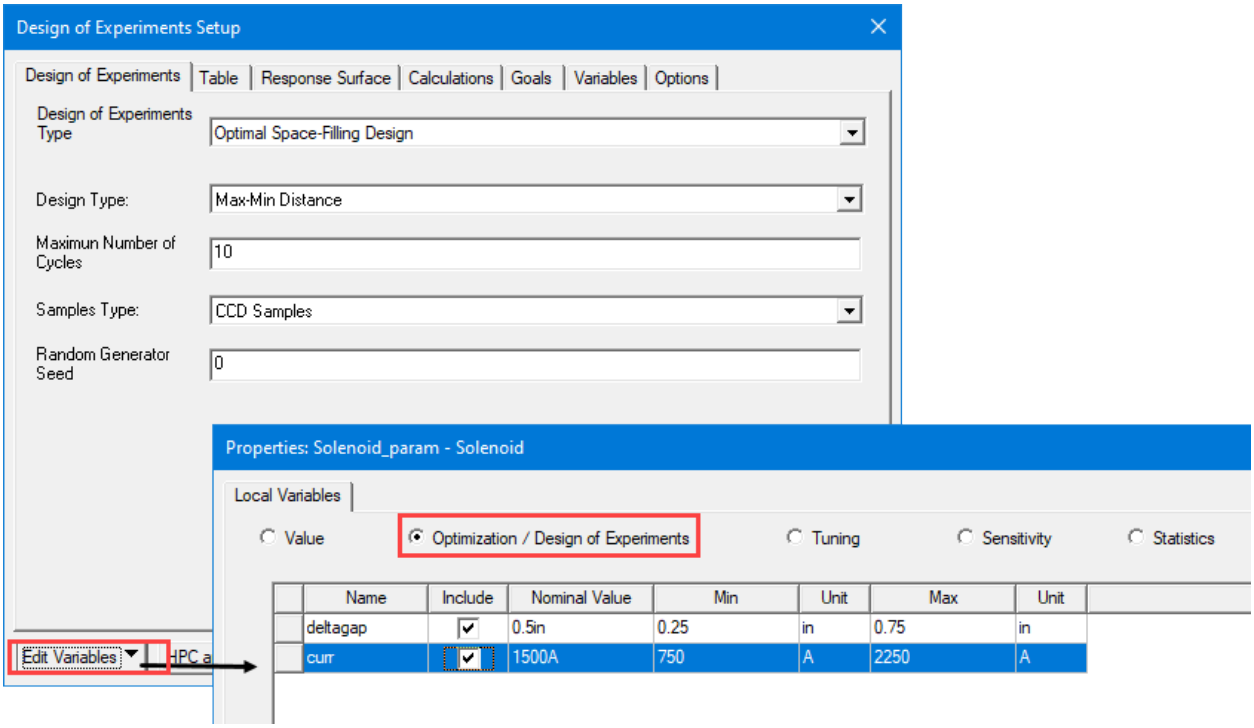


It has tabs for

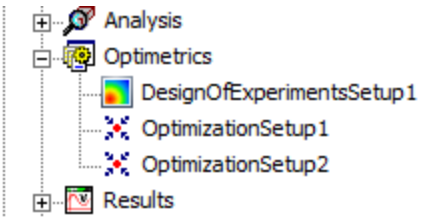
- Design of Experiments, where you specify the sampling points and associated parameters.
- Table, which shows the sampling points defined by the Design of Experiments settings you provide. If you select Custom and the Design of Experiments type, you can import data files as well as add or edit rows. 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

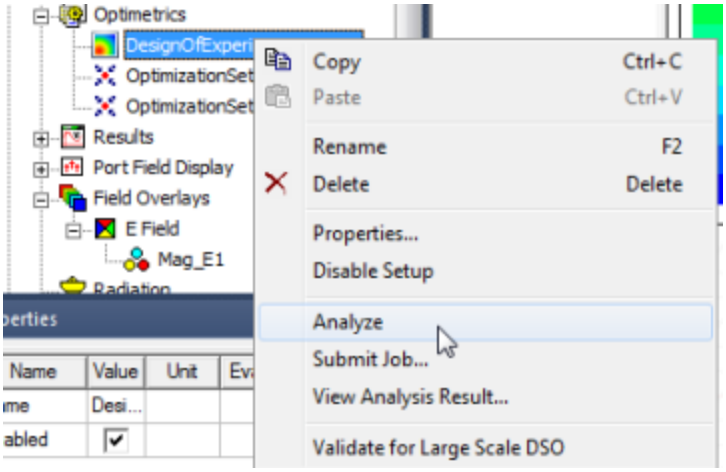


Once you have set parameters and click **OK**, the Design of Experiments setup appears under the Optimetrics icon in the Project tree.



From here you can right-click the DesignOfExperiments setup from the shortcut menu to run **Analyze**, **Submit Job...** or **Validate for Large Scale DSO**.

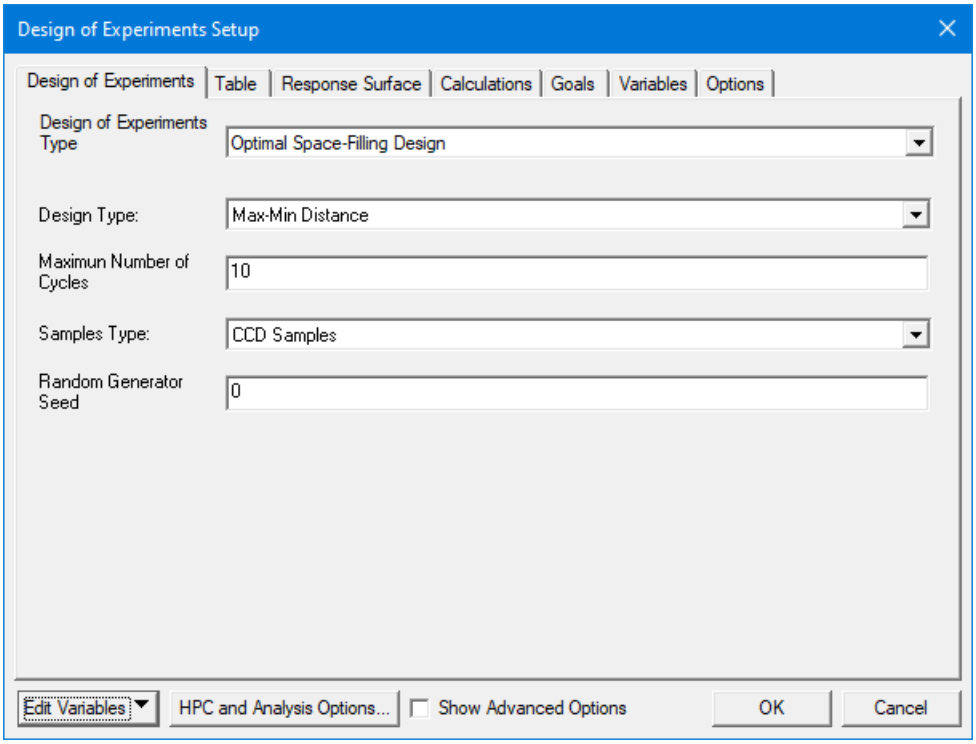
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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 • 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

Ansys Electromagnetics Suite 2026 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)
	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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **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).

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

- 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:

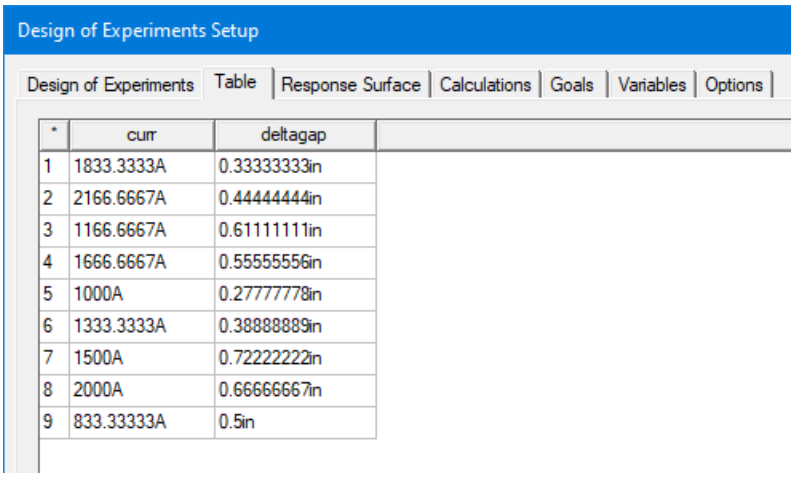
- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **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

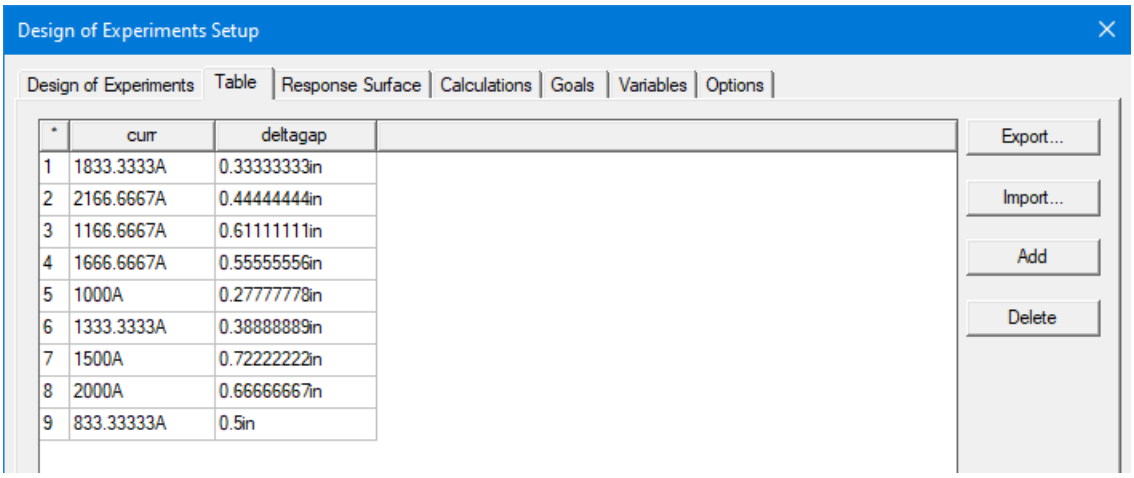
The **Table** tab for the **Design of Experiments Setup** window displays a preview of the design points designed by your selections on the **Design of Experiments** tab. There is one column for each defined variable. The **Export** button lets you create a file of the table in a format you specify.



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 input parameter

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



*	curr	deltagap
1	1833.3333A	0.33333333in
2	2166.6667A	0.44444444in
3	1166.6667A	0.61111111in
4	1666.6667A	0.55555556in
5	1000A	0.27777778in
6	1333.3333A	0.38888889in
7	1500A	0.72222222in
8	2000A	0.66666667in
9	833.33333A	0.5in

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

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

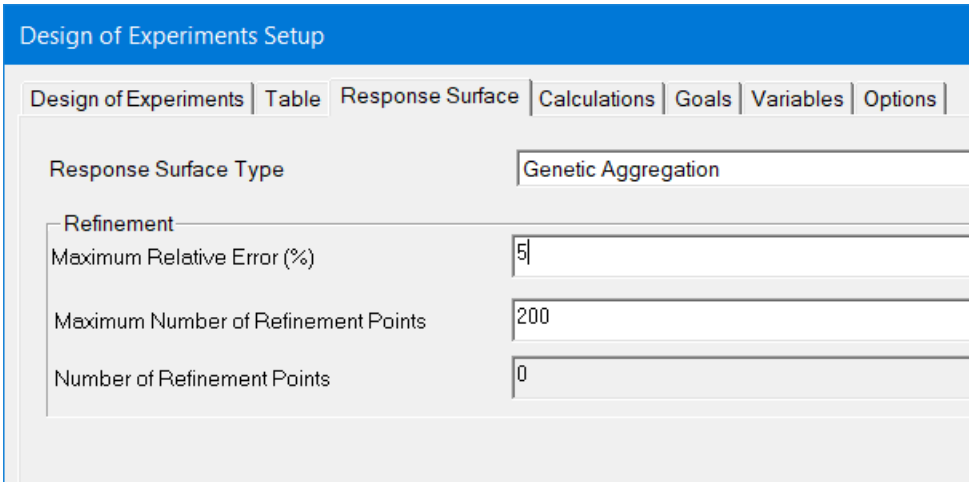
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files, (*.txt)

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** window lets you select the Response Surface type as **Genetic Aggregation** or **Standard Response Surface**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



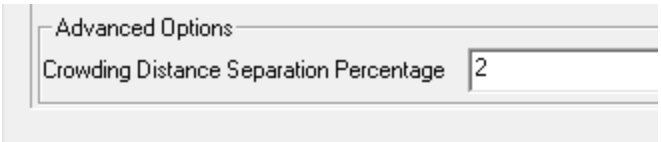
The Response Surface type selection specifies the refinement applied to the initial Design of Experiments. The Genetic Aggregation setting finds the best possible response surface for each output automatically by combining

- 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

- **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.

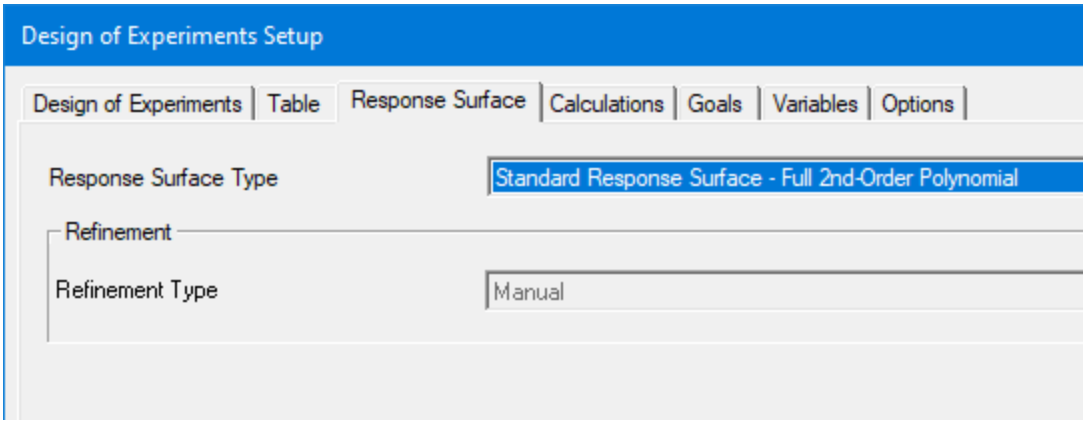
If you check **Show Advanced Options** at the bottom of the **Setup** window, you can also specify **Crowding Distance Separation Percentage**:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This setting defines the minimum allowable distance between new refinement points; it is implemented as a constraint in the search for refinement points.

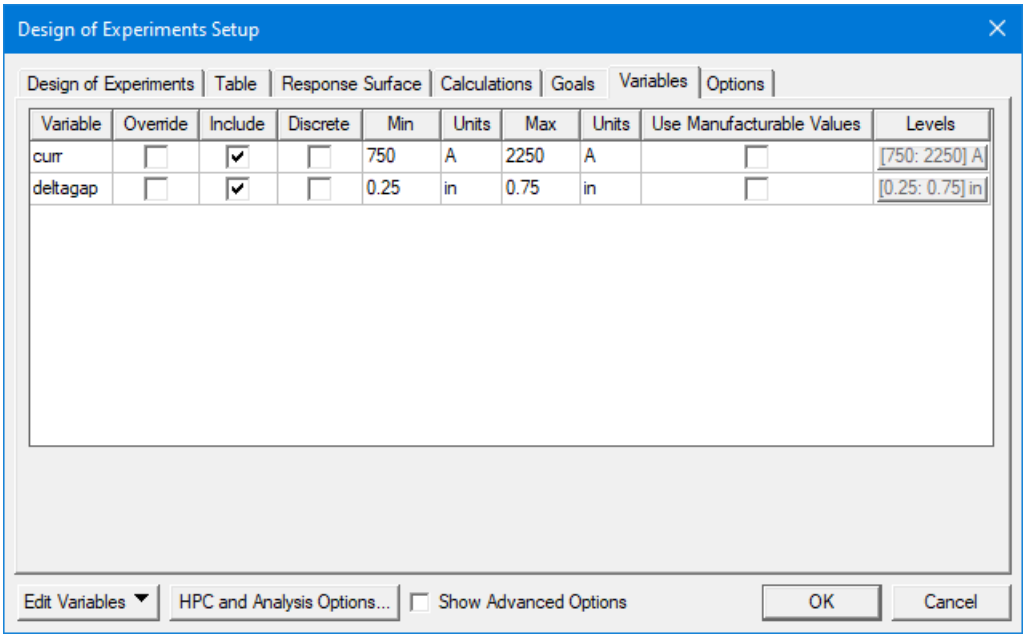
You do have the option of selecting **Standard Response Surface- Full 2nd Order Polynomial**:



After you have completed an analysis, you can view the generated plot.

Variables Tab for Design of Experiments

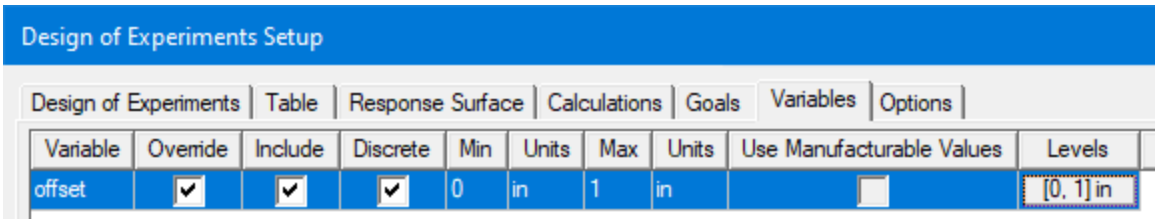
The Variables tab for the **Design of Experiments Setup** window displays list of the variables defined for the Design and Project as Optimetrics/Design of Experiments variables.



Ansys Electromagnetics Suite 2026 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:

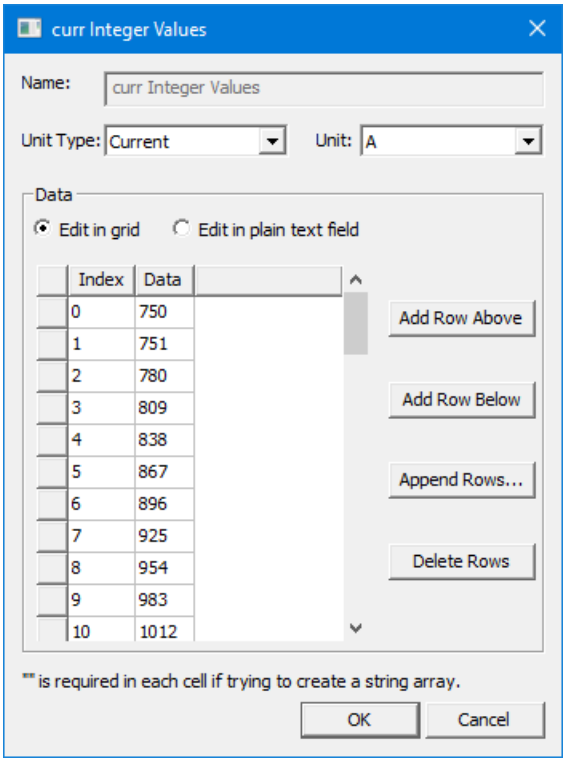
- **Override** – check this to override the current design value. Unchecking this causes a dialog to appear asking you to confirm the return to the design value.
- **Include** – whether to include the variable in an analysis.
- **Discrete** – **Discrete Variables** physically represent different configurations 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:

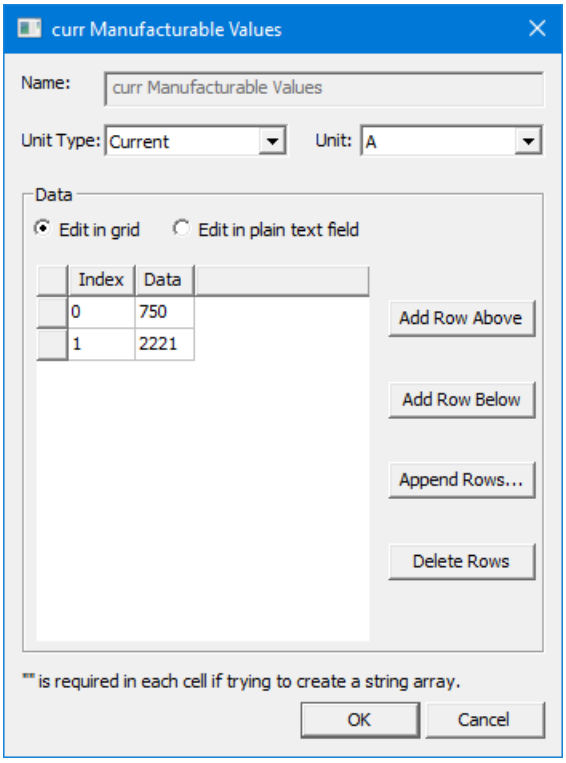
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

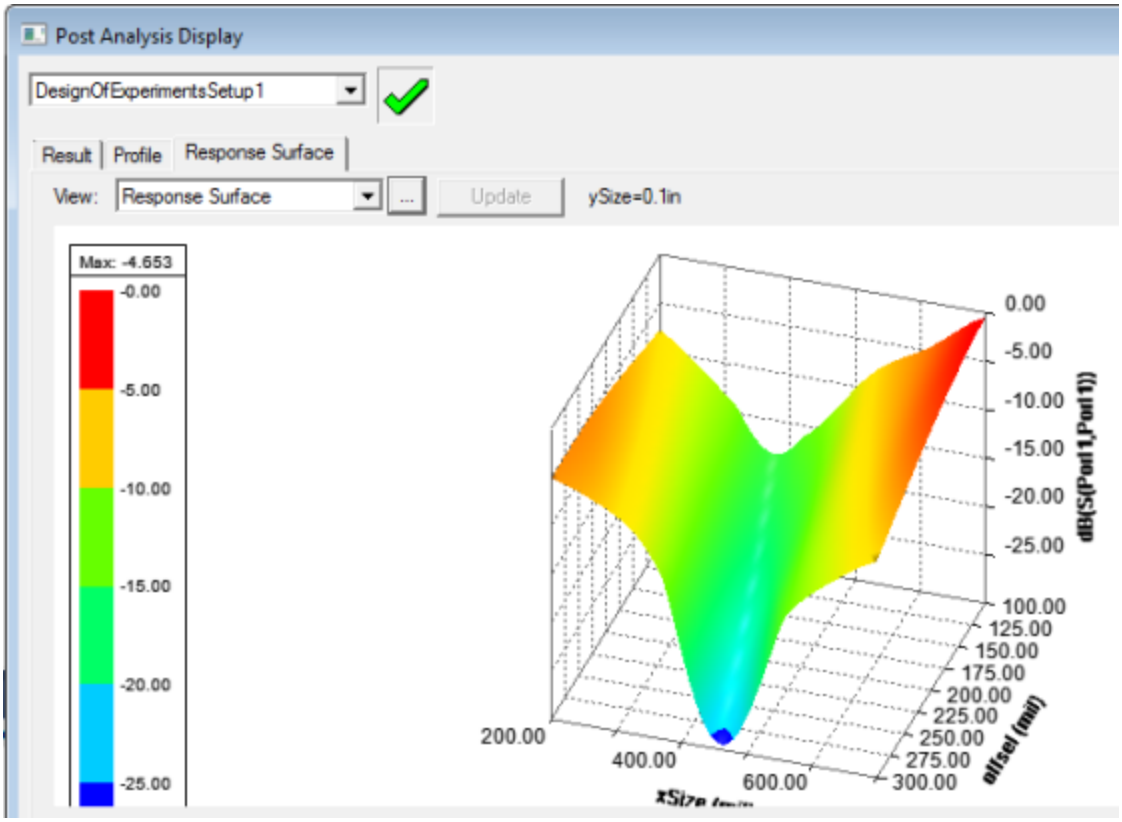
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



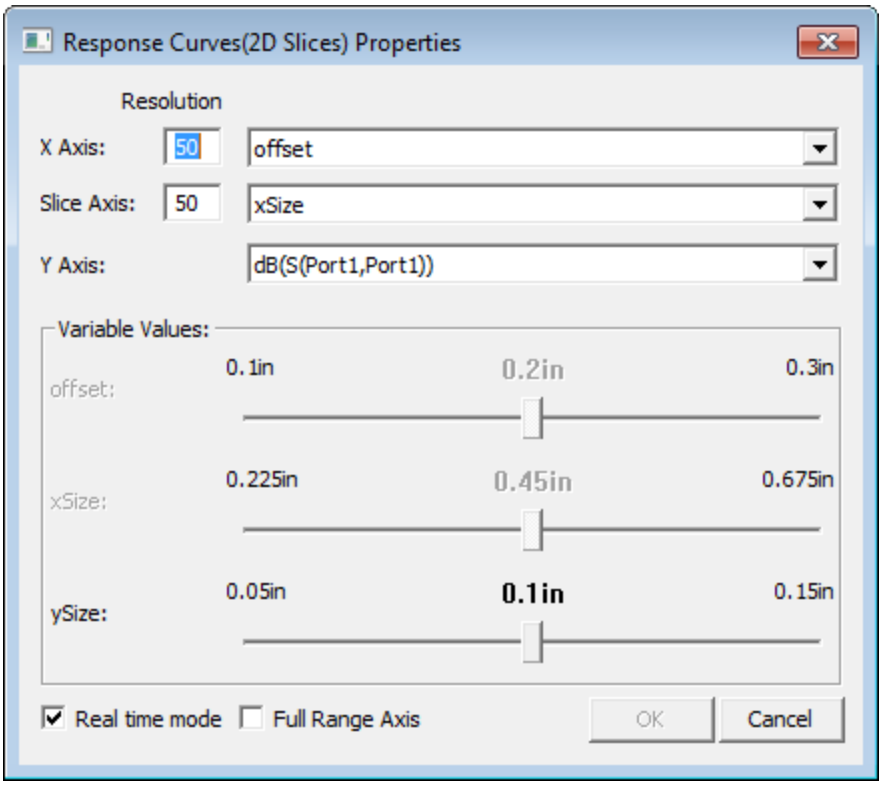
From the **Response Surface** tab, the “View” list box lets you select all available views of the selected response-surface-setup.

- 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

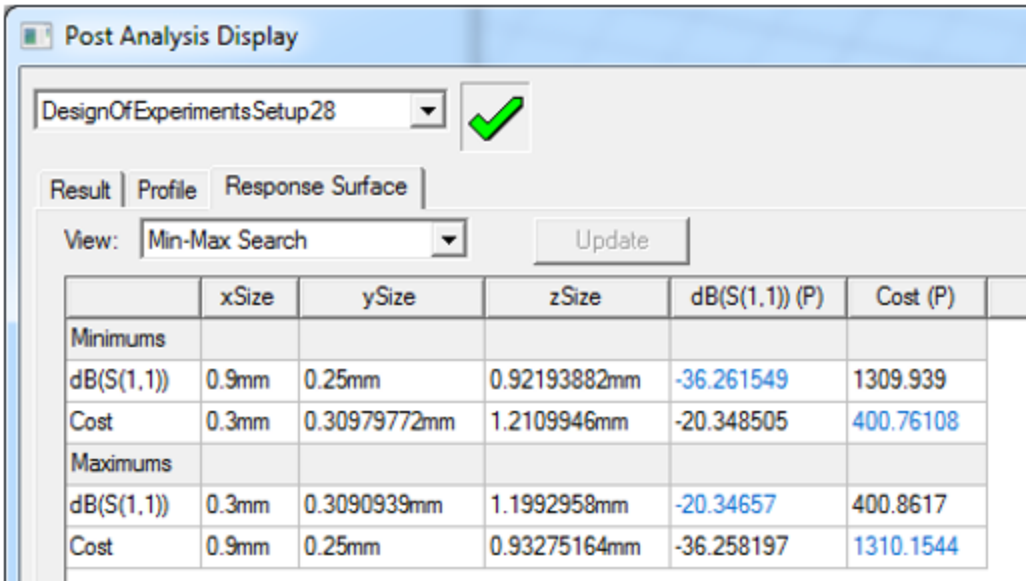
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

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

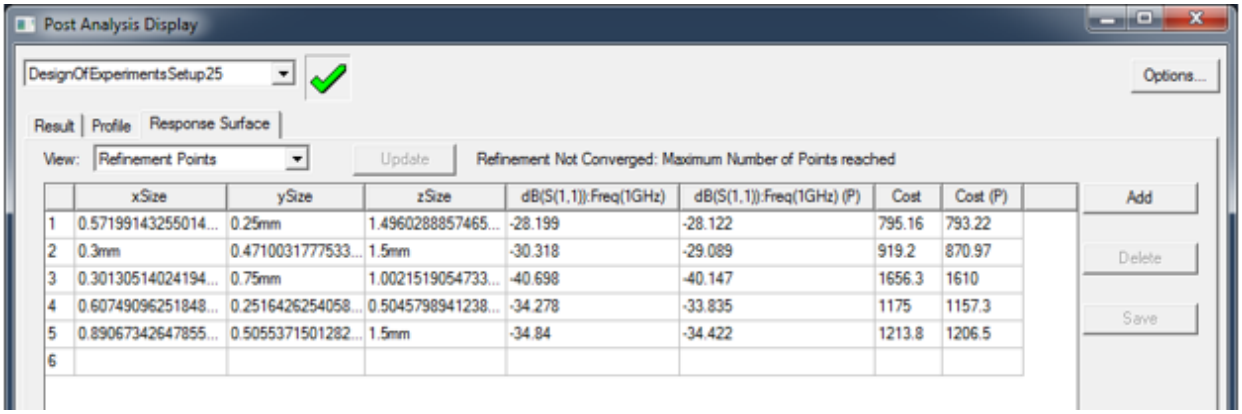
You can Export the Table in the following formats:

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	xSize	ySize	zSize	dB(S(1,1)):Freq(1GHz)	dB(S(1,1)):Freq(1GHz) (P)	Cost	Cost (P)
1	0.57199143255014...	0.25mm	1.4960288857465...	-28.199	-28.122	795.16	793.22
2	0.3mm	0.4710031777533...	1.5mm	-30.318	-29.089	919.2	870.97
3	0.30130514024194...	0.75mm	1.0021519054733...	-40.698	-40.147	1656.3	1610
4	0.60749096251848...	0.2516426254058...	0.5045798941238...	-34.278	-33.835	1175	1157.3
5	0.89067342647855...	0.5055371501282...	1.5mm	-34.84	-34.422	1213.8	1206.5
6							

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.

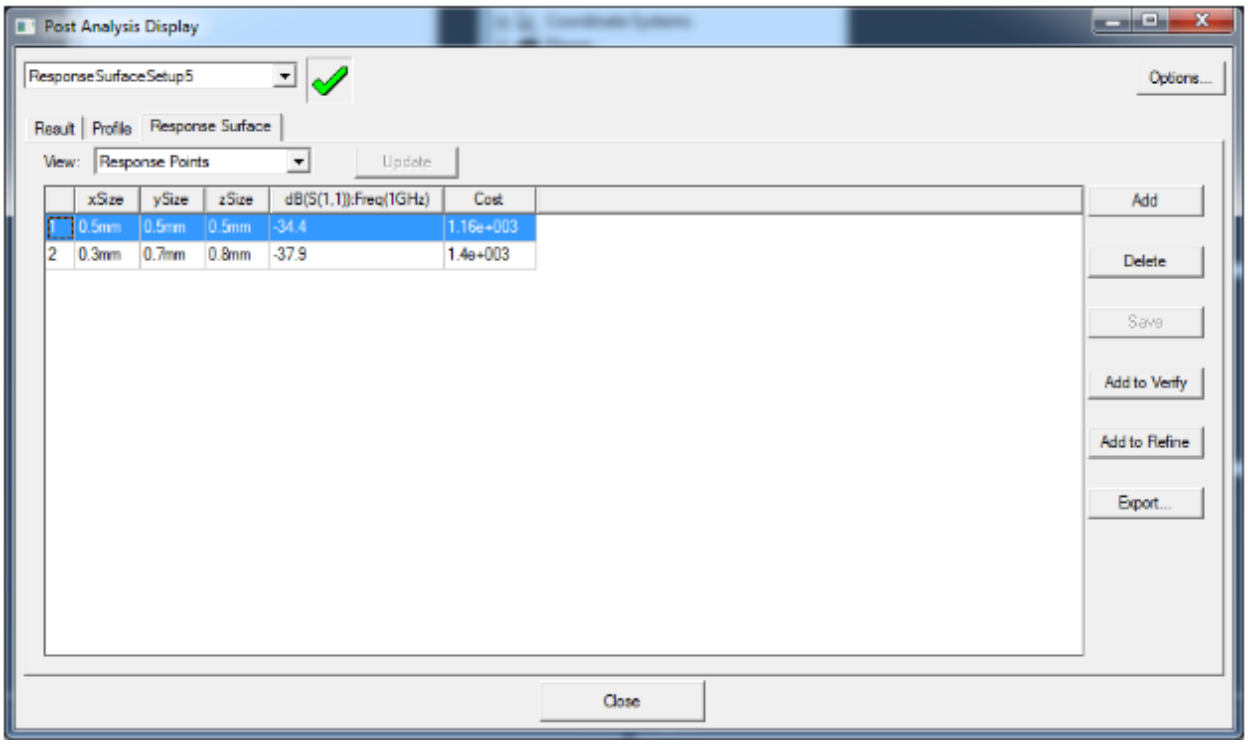
You can add, delete, or modify refinement points by clicking the **Add** or **Delete** buttons, or by modifying point values directly in the grid.

Manual Refinement Point can be inserted from the Response point table and Verification Points Table.

Response 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

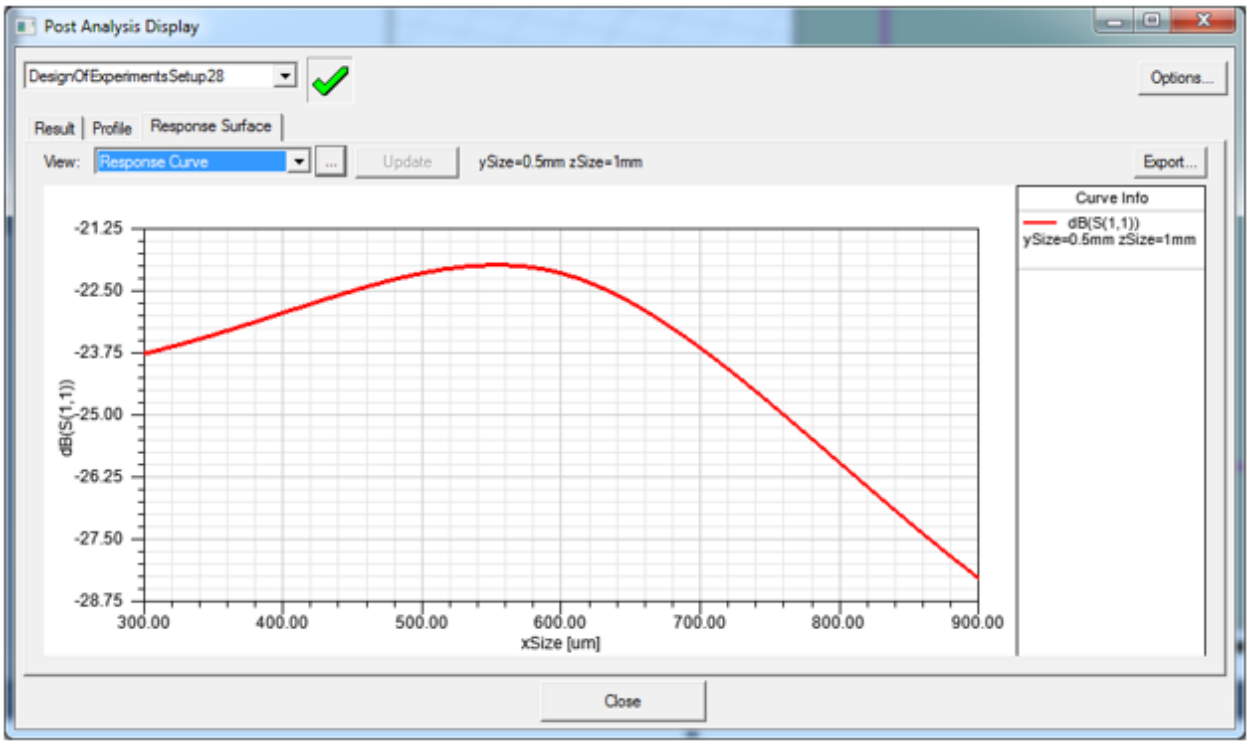
You can add, delete, save, or export response points by using the command buttons in the dialog or you can modify response points manually by modifying point values directly in the grid.

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.

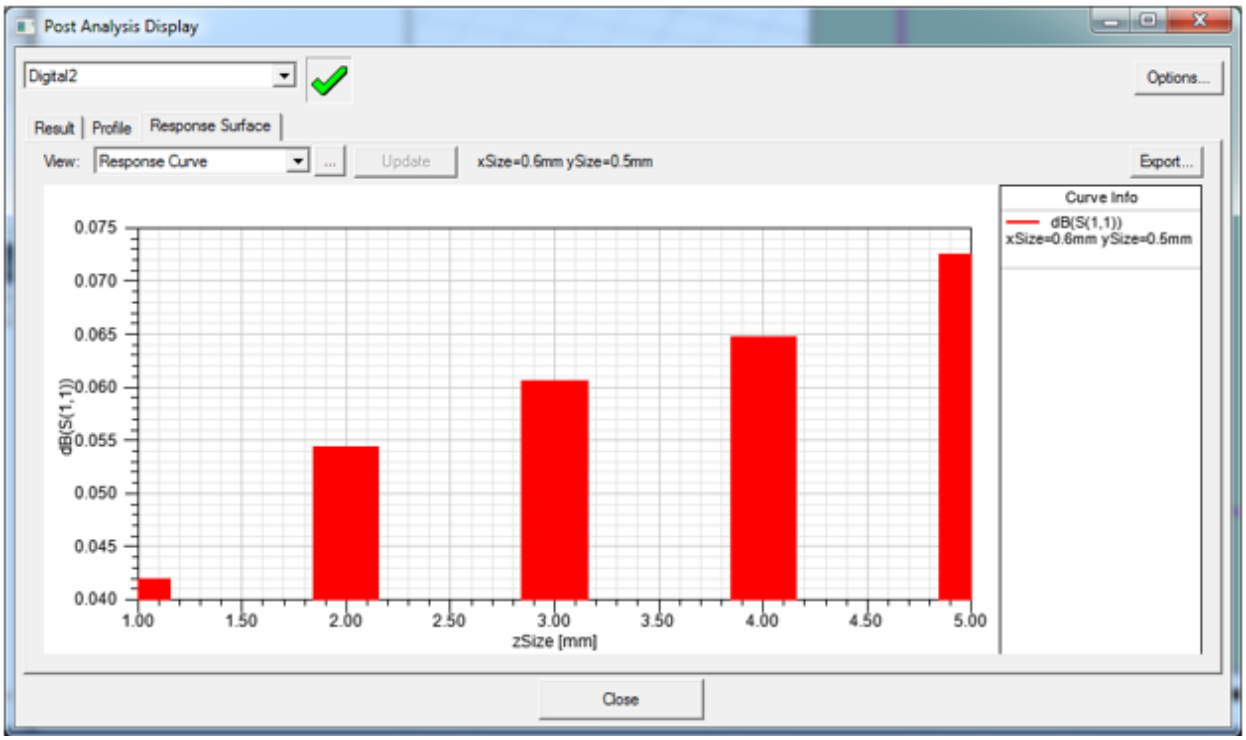
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Discrete Variable

If you choose Discrete Variable as the X-axis, a bar chart plot is shown:

Ansys Electromagnetics Suite 2026 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:

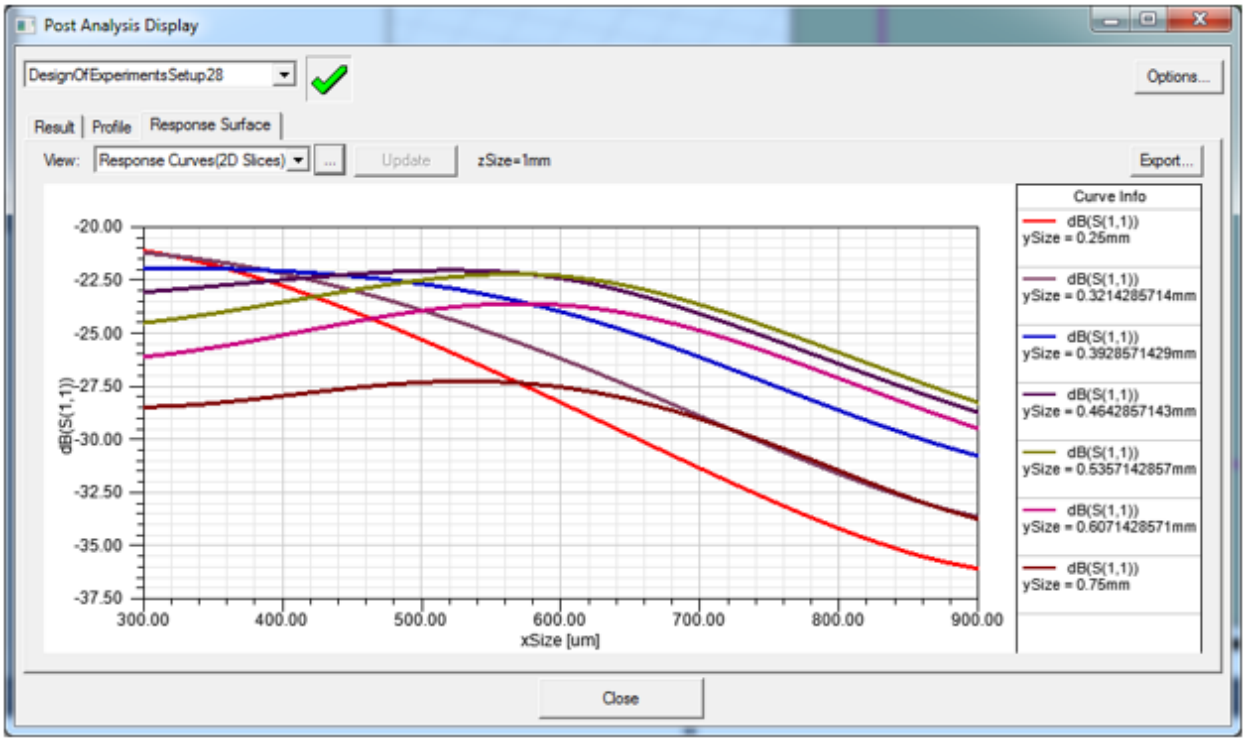
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

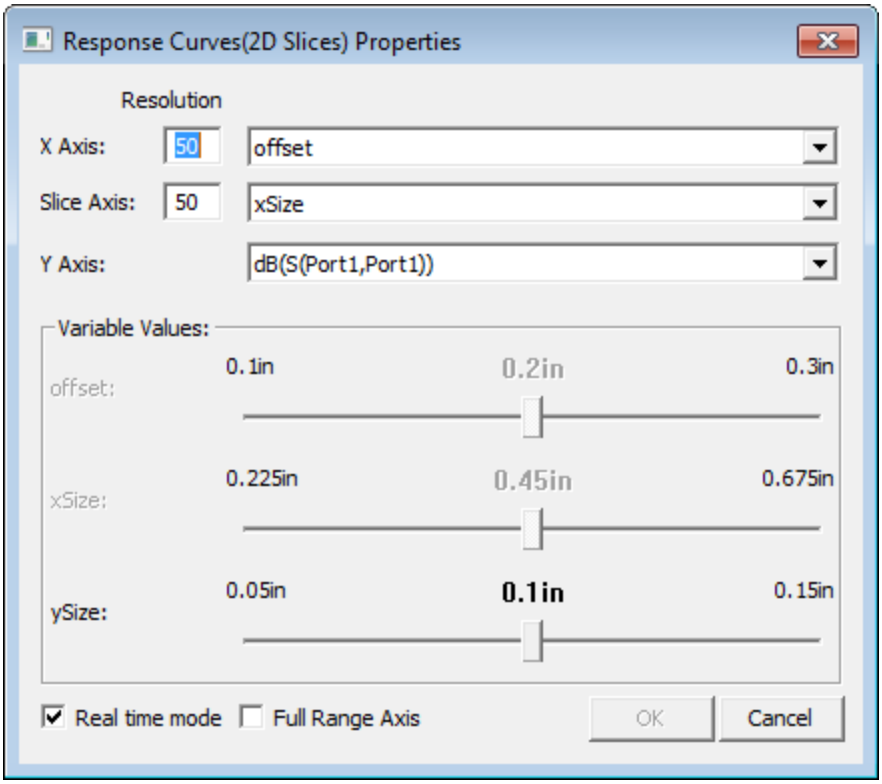
Response Curve (2D Slices)

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 2026 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:

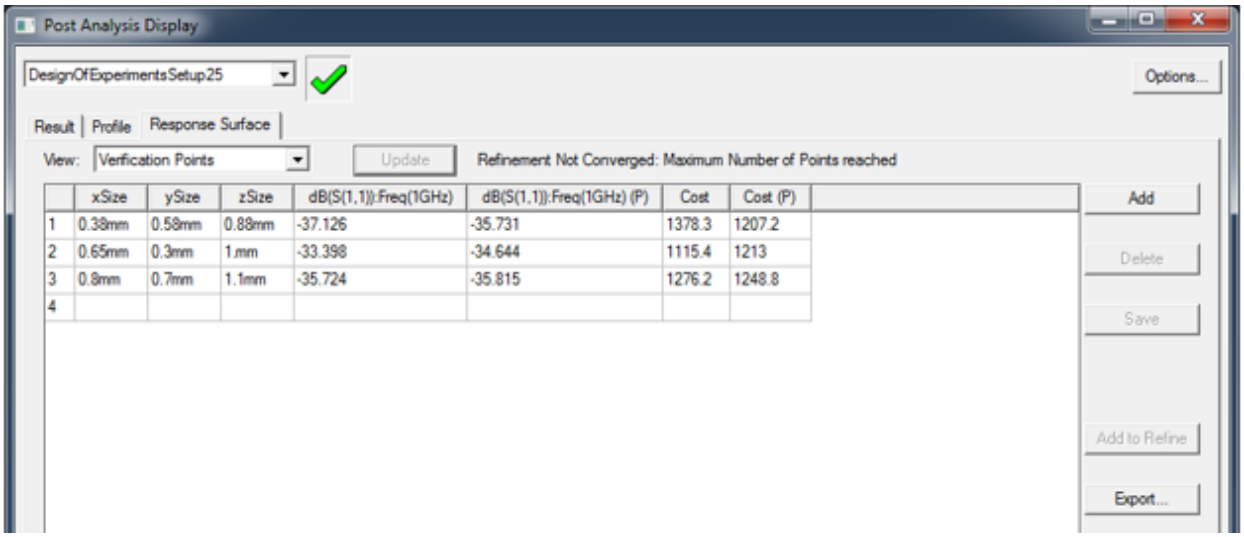
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Ansys Electromagnetics Suite 2026 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.

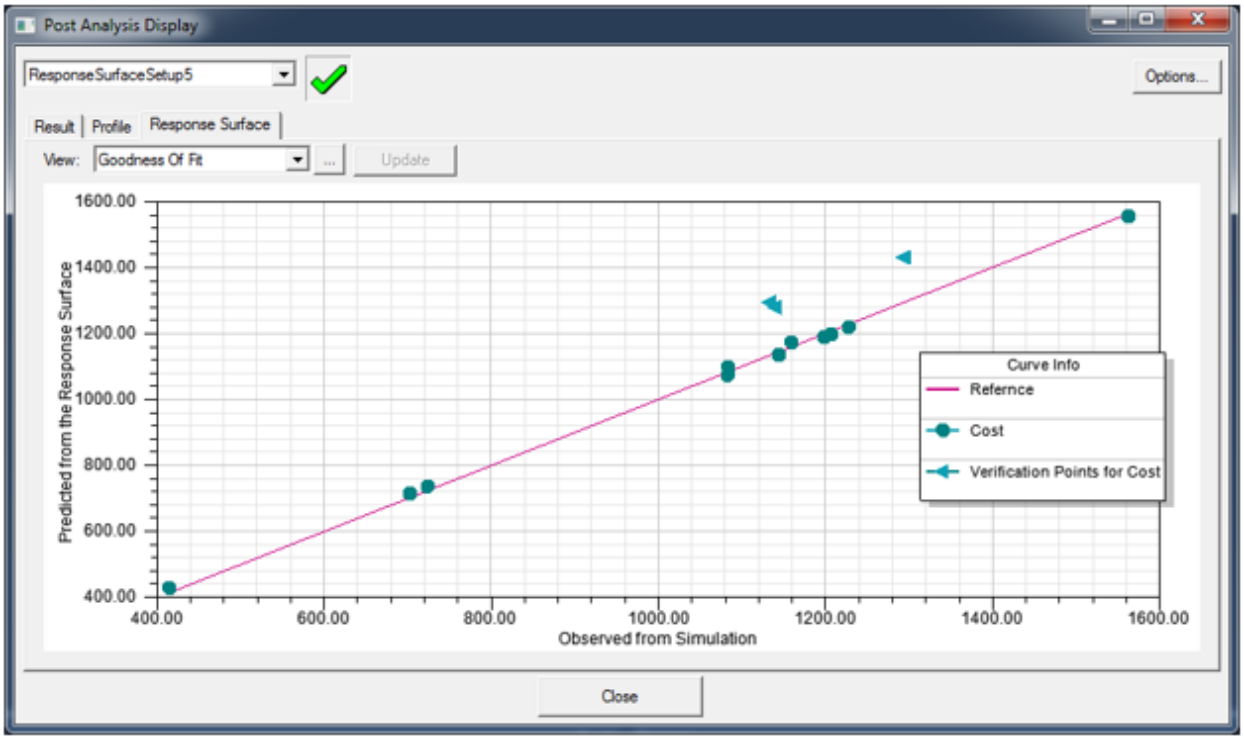
Verification points are useful in validating any type of response surface. In particular, however, you should always use verification points to validate the accuracy of interpolated response surfaces, such as Kriging or Sparse Grid.

Ansys Electromagnetics Suite 2026 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)

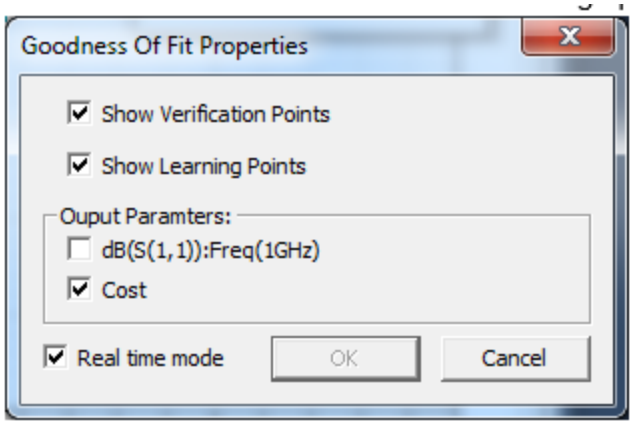
From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. Response surfaces are built from design points in the Design of Experiments (DOE) and refinement points (collectively, called "learning points"). The Goodness of Fit calculations 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

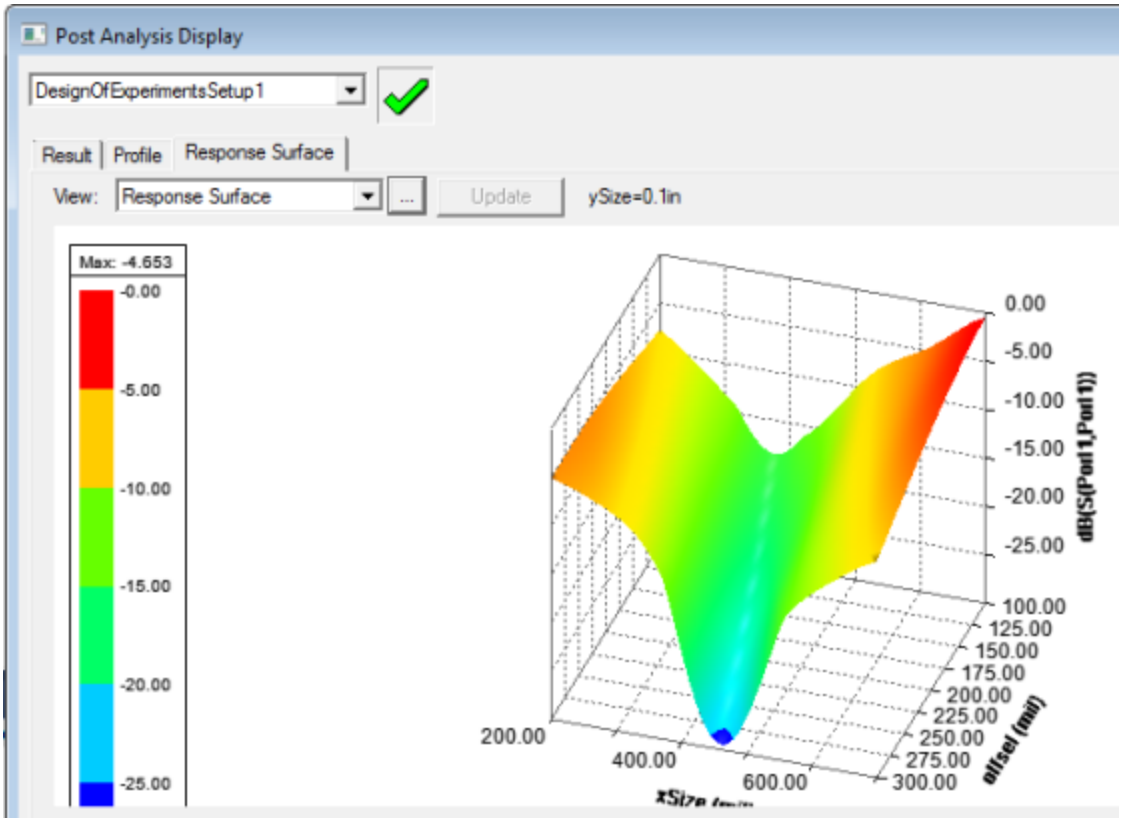


Response Surface Results

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.

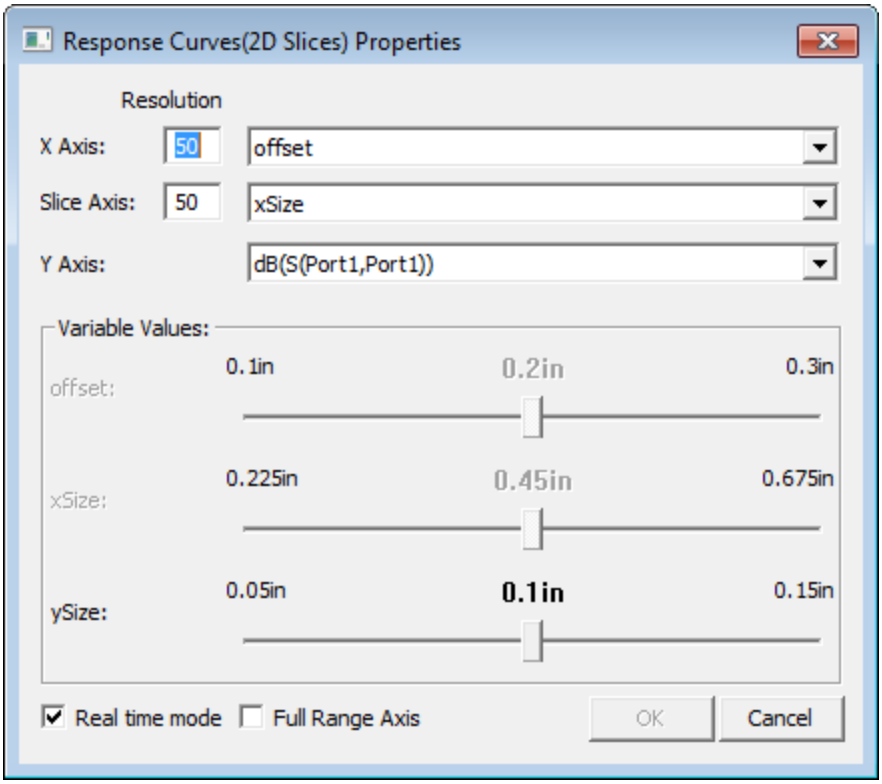
Response surfaces are functions of varying natures in which the output parameters are described in terms of the input parameters. Built from the Design of Experiments (DOE), they quickly provide the approximated values of the output parameters throughout the design space without having to perform a complete 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.

Ansys Electromagnetics Suite 2026 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 that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2026 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:

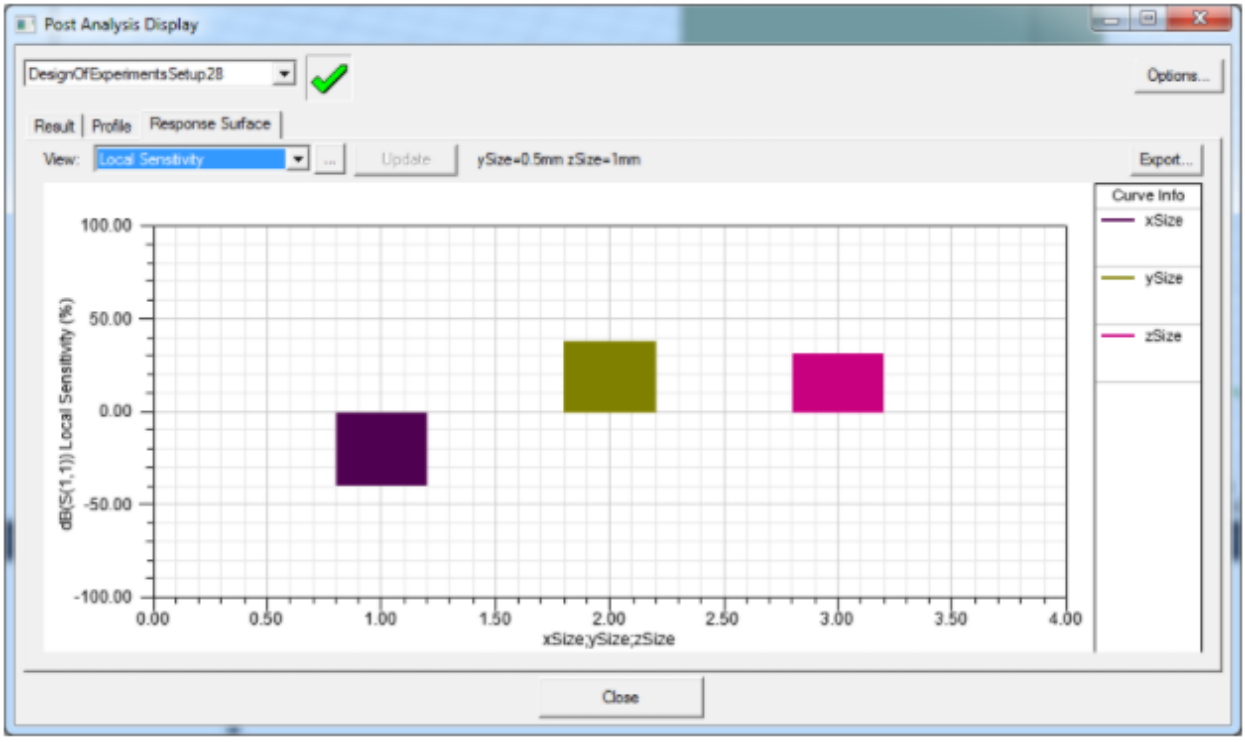
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Exported files can be imported into a report.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

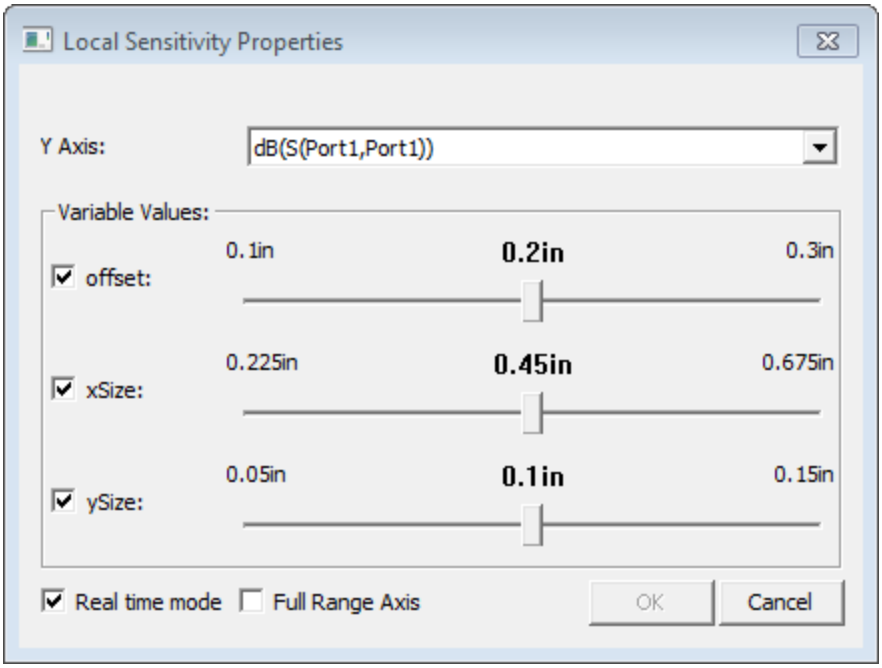
Local Sensitivity Charts

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.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog that lets you adjust the variables selected and the values applied.

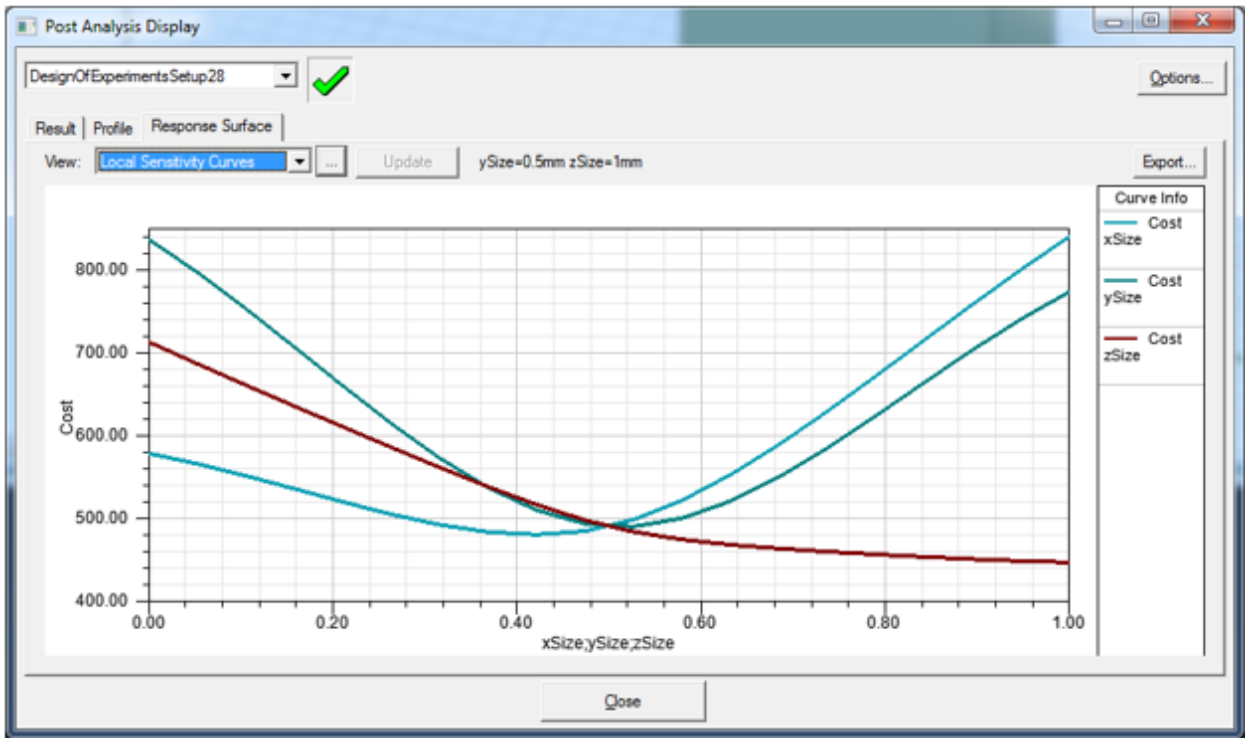
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Local Sensitivity Curves

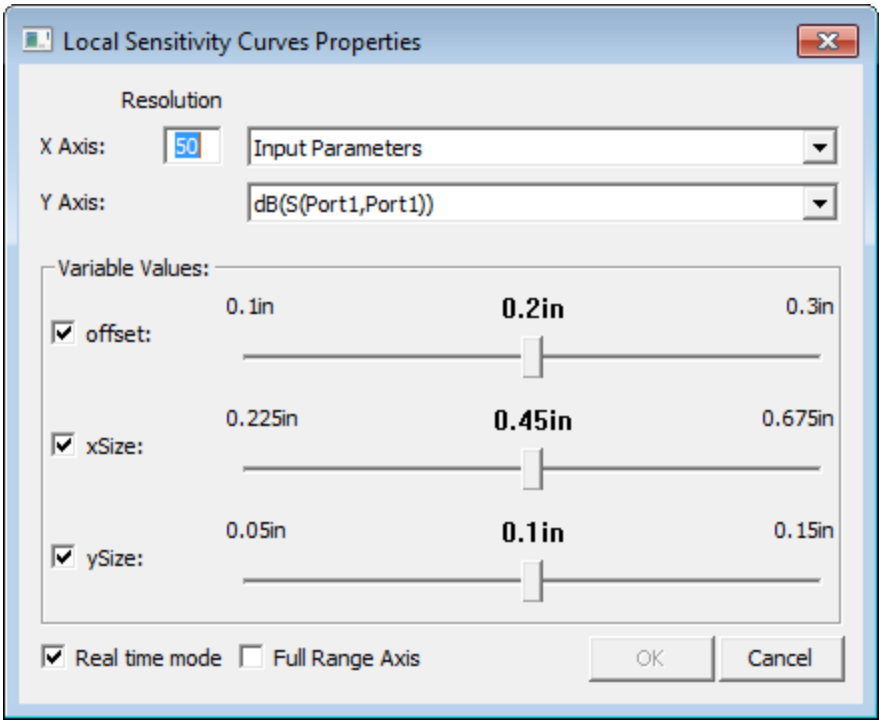
From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Curves allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity Curves chart helps 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

Tuning a Variable

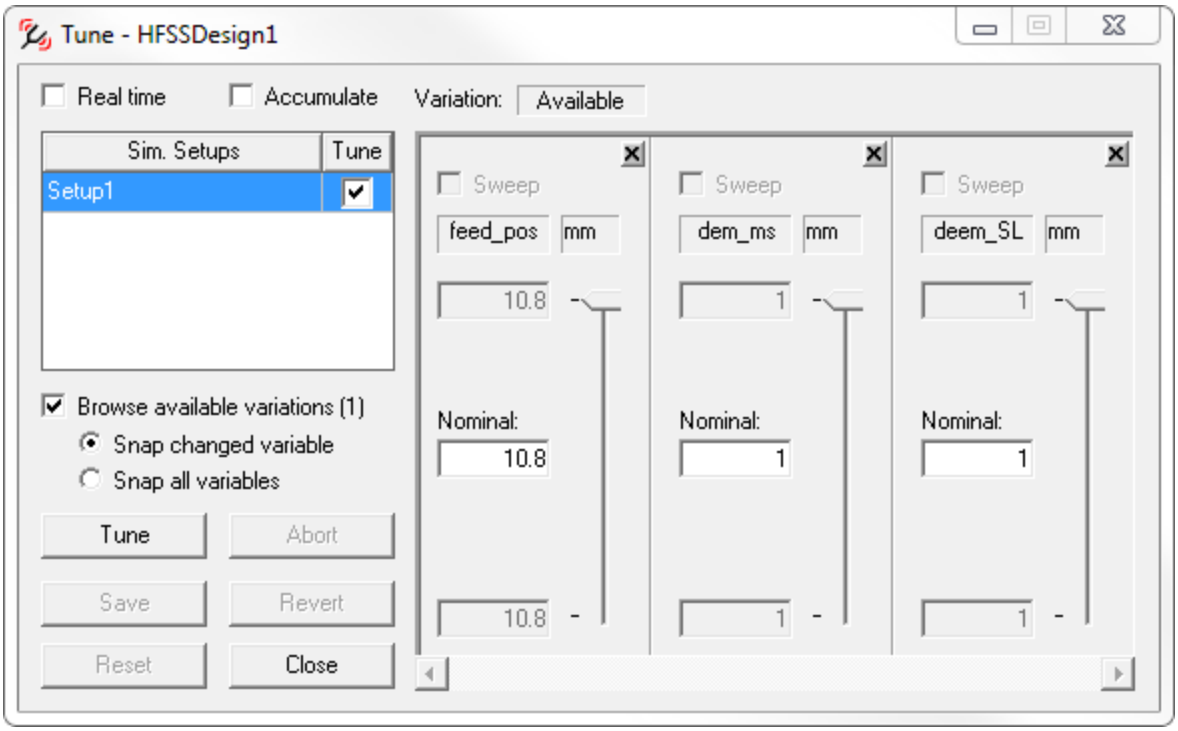
If you want to ensure that tuning does not resolve variations already solved by an optimization setup, you must check **Save Fields and Mesh** in the **Options** tab of that setup.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Tune** dialog box appears, listing the variables which have been included for tuning.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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.

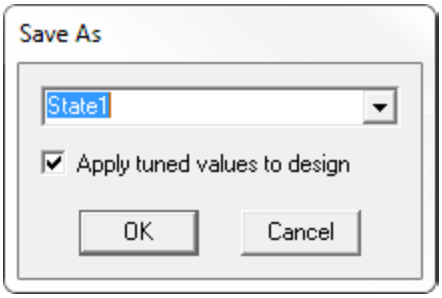
Saving a Tuned State

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.

1. After tuning a variable, click **Save** in the **Tune** dialog box.

   A **Save As** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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, IcepakFEA, 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 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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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.

Ansys Electromagnetics Suite 2026 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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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.

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.

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 2026 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**.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

$$cost(x): R^n \rightarrow R$$

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.

> **Important:**
>
> Linear constraints may only be applied to the following optimizers:
>
> - MATLAB
> - Sequential Nonlinear Programming (Gradient)
> - Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
> - Quasi Newton (Gradient)
> - Pattern Search (Search-based)
> - Genetic Algorithm (Random Search)

To set a linear constraint:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Variables** tab one of two ways:
   - 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 2026 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.

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.

Viewing Analysis Results for Optimetrics Solutions

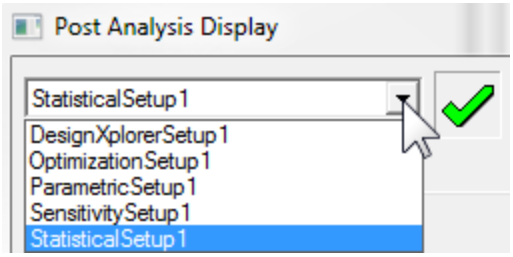
To view data specific to an Optimetrics solution:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 convergence information 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 **Icepak** 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.
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 Icepak as it solved each design variation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Manager 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.
   - 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Manager 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 Manager 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Manager 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.

Link to DesignXplorer

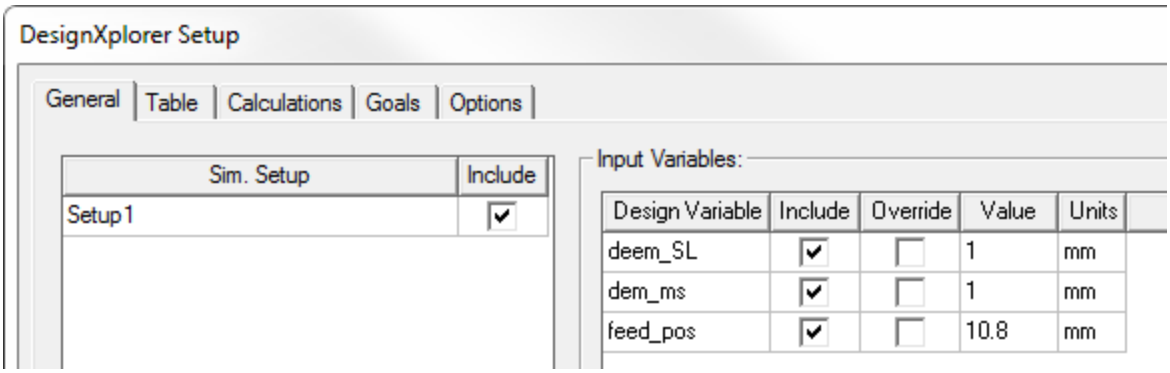
You can export a .xml file containing information on an Icepak setup, optimization variables, and output variables that enables Ansys Design Xplorer to manage the simulations (for example, for design of experiments and optimization). Design Xplorer will launch Ansys Electronics Desktop simulations of 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

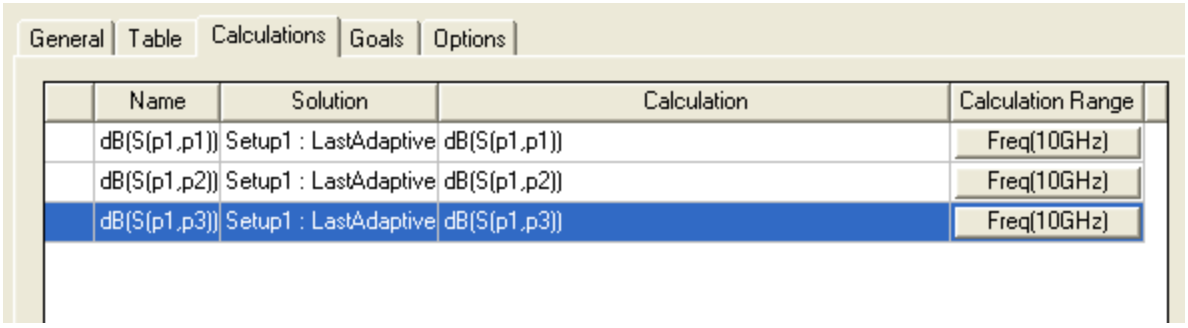
This opens the **Design Xplorer** dialog box with the **General** tab selected. It lists the setups available in the current project, and the input variables it contains.



2. Select **Include** for the simulation setups you want to use.
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.

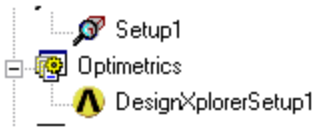
   Use the **Add Calculation** button to add expressions to the Calculations table.



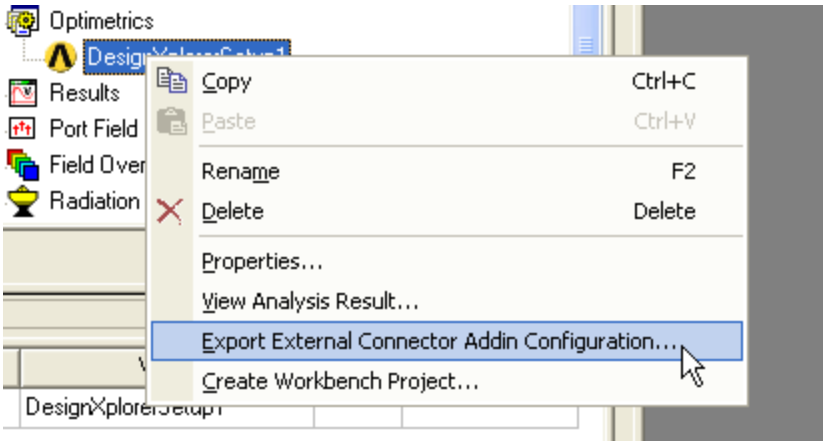
5. When you have added the calculations of interest, click **OK** to save the setup.

   An icon for the Design Xplorer setup appears under Optimization in the Project tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. To create a .xml file with the setup information for Design Xplorer, first Save your project.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - 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. Icepak 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 Burst Computing Support for HPC
- Large Scale DSO for Parametric Analysis
- Interactive Scheduler Jobs
- HPC Integration
- Multi-Step Job Submission
- Distribution Command Line Options
- GPU Acceleration

High Performance Computing in Icepak

> **Note:**
>
> The GPU feature appears in the user interface by default as an open beta feature. To use the GPU feature, you must enable the beta option. For information on enabling beta options, see General Options: Desktop Configuration.

The Ansys Electronics Desktop has different options and settings to improve the efficiency of various Icepak processes.

- Pre/post processing option
- High Performance Computing (HPC) tasks
- HPC cores

> **Note:** The number of tasks specified must be less than or equal to the number of cores.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre/post Processing Option

The **Desktop Performance** settings in the **General Options** contains the Pre/post processing option (see General Options: Desktop Performance). For Icepak designs, the number of processors specified affects the following Icepak processes:

- Viewing the mesh with the Mesh Viewer
- Mesh quality calculations with the Mesh Viewer
- Plotting field overlay data
- Field summary calculations
- Fields calculator calculations

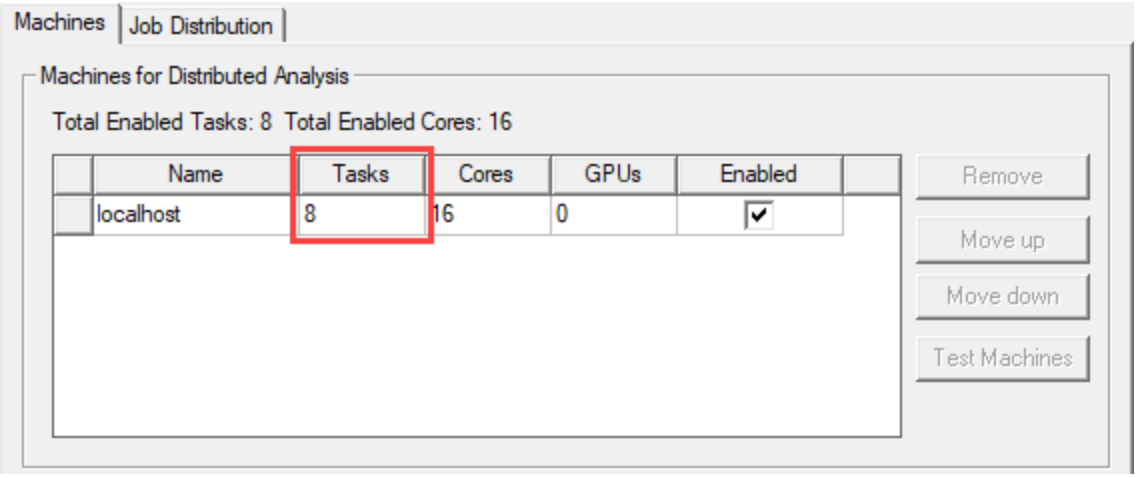
HPC Tasks

When configuring HPC settings for distributed analysis (see Editing Distributed Machine Configurations), the number of tasks you specify affects the following Icepak processes:

- Meshing a design with multiple mesh regions
- Solver initialization
- Nominal solve (Ansys Fluent process)
- Parametric setup analysis

Meshing

When an Icepak design contains multiple mesh regions, specify multiple tasks to mesh multiple regions concurrently.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a design contains fewer than or equal to the number of tasks specified, all mesh regions are meshed concurrently. For example, if a design contains 10 mesh regions and 12 tasks are specified, all 10 mesh regions are meshed concurrently.

When a design contains more mesh regions than tasks specified, the number of mesh regions meshed concurrently is equal to the number of tasks specified. For example, if a design contains 10 mesh regions and 8 tasks are specified, 8 mesh regions are meshed concurrently, and the remaining 2 mesh regions will be meshed concurrently as computing resources become available.

The following table contains more examples of task usage for Icepak meshing scenarios.

Tasks	Mesh Regions	Concurrently Meshed Regions
4	8	4
16	12	12

Parametric Analysis

One Level Distribution

When a parametric setup contains multiple variables and, therefore, multiple variations of a simulation, one level distribution uses the specified number of tasks to solve multiple variations concurrently.

When a parametric setup contains fewer variations than tasks specified, the number of variations solved concurrently is equal to the number of tasks specified. For example, if a parametric setup contains 10 variations and 12 tasks are specified, all 10 variations are solved concurrently.

When a parametric setup contains more variations than tasks specified, the number of variations solved concurrently is equal to the number of tasks specified. For example, if a parametric setup contains 16 variations and 12 tasks are specified, 12 variations are solved concurrently, and the remaining 4 variations will be solved concurrently as computing resources become available.

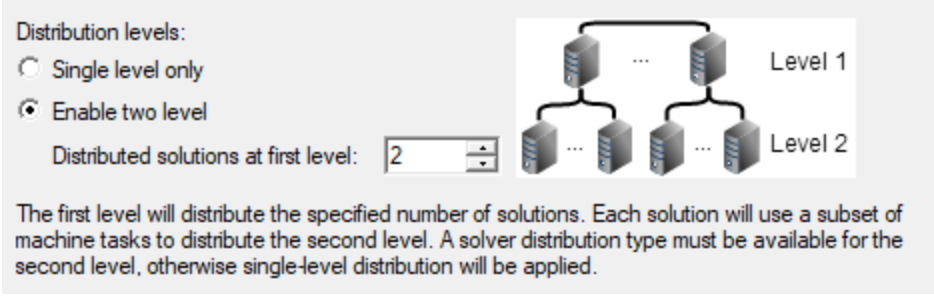
The following table contains more examples of task usage for Icepak one level parametric analysis scenarios.

Variations	Tasks	Concurrent Runs	Tasks Per Concurrent Run
10	8	8	1
5	10	5	1
16	4	4	1
4	16	4	1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two Level Distribution

When you enable two level distribution for a parametric setup, specify a number of distributed solutions (variations) at the first level. Level two distribution uses the number of distributed solutions to solve multiple variations concurrently.



The tasks specified are divided equally among the distributed solutions. For example, if a parametric setup contains 10 variations and 2 distributed solutions are specified, 2 variations are solved concurrently. In this scenario, if 8 tasks are specified, 4 tasks are used per variation.

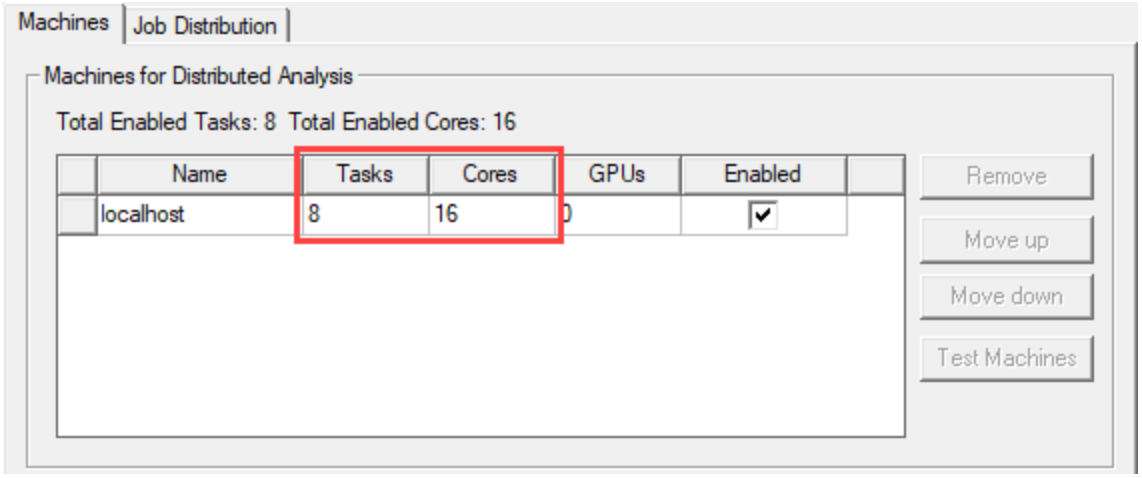
The following table contains more examples of task usage for Icepak two level parametric analysis scenarios.

Variations	Tasks	Distributions at First Level	Concurrent Runs	Tasks Per Concurrent Run
10	8	2	2	4
5	12	4	4	3
16	4	2	2	2
4	16	4	4	4

COM Engine CPU Usage

High performance computing in the Electronics Desktop impacts your overall CPU usage. The number of tasks and cores specified determines the maximum percentage of CPU usage.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



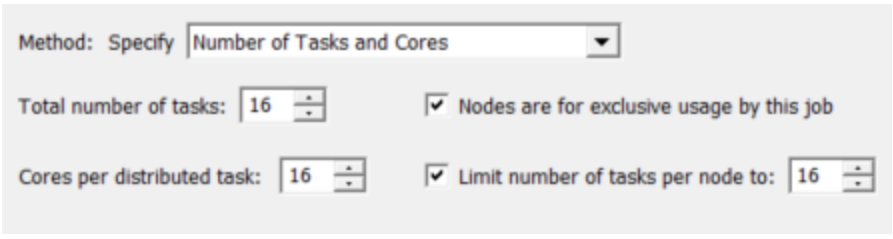
The following table contains examples of maximum COM engine CPU usage scenarios.

> **Note:** The number of cores your CPU has determines the maximum CPU usage percentage. The Max CPU Usage percentages in the table below are based on a test machine with 16 cores.

Tasks	Cores	Max CPU Usage
2	2	13.1%
2	4	25.6%
4	4	26%
8	8	35%
8	16	55%

Specifying Tasks and Cores for Job Schedulers

When configuring High Performance Computing with a job scheduler, the number of tasks and cores you specify determines how the cluster processes jobs. Also, other configuration settings impact the number of cluster nodes used for each job.

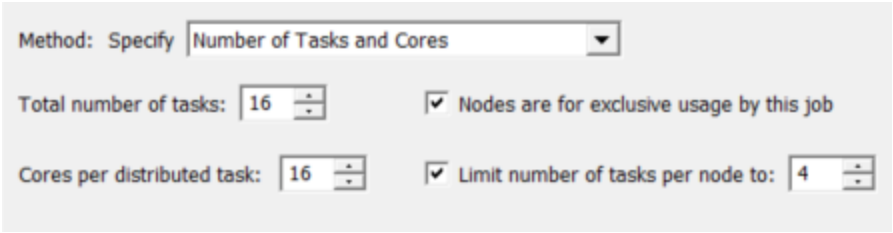


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When specifying the number of tasks and cores for a job scheduler in Icepak, the total number of cores used is equal to the **Total number of tasks** specified multiplied by the **Cores per distributed task** specified.

As defined in the image above, if you specify 16 tasks and 16 cores per task, the total number of cores is 256 (number of tasks * number of cores). In this example, if you enable **Nodes are for exclusive usage by this job** and **Limit number of tasks per node to** and enter 16 as the value for the latter, 16 cluster nodes are used to process 16 tasks each.

As defined in the image below, changing the value of **Limit number of tasks per node to** to 4 results in 64 cluster nodes used to process 4 tasks each.



Specifying Tasks and Cores from the Command Line

When specifying tasks and cores from the command line, there is no ability to specify the number of tasks per node. The number of cluster nodes used is determined by the total cores and cores available on each cluster node.

```
C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt -distributed
includetypes=default maxlevels -machinelist num=16 -monitor -ng -
batchoptions " 'Icepak/NumCoresPerDistributedTask'='16'" -batchsolve
IcepakDesign1 \\servername.host.com\filepath\projectname.aedt
```

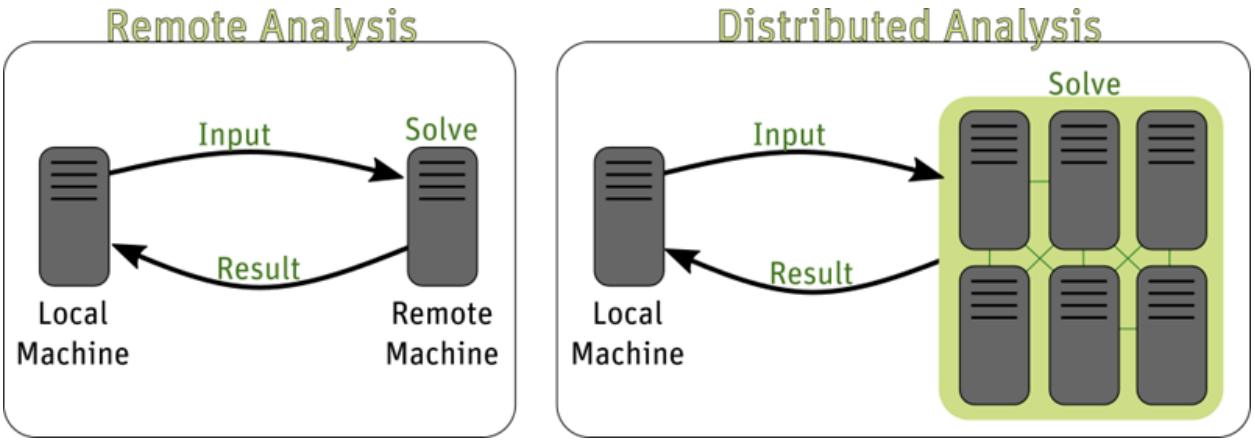
For example, if you specify 16 tasks and 16 cores, 256 cores are used. If each cluster node has 32 cores, 8 cluster nodes are used.

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 2026 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 2026 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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 2026 R1** > **Register with RSM**. You can also run **RegisterEnginesWithRSM.exe**, located in the product subdirectory (for example, C:\Program Files\ANSYS Inc\v261\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/v261/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.

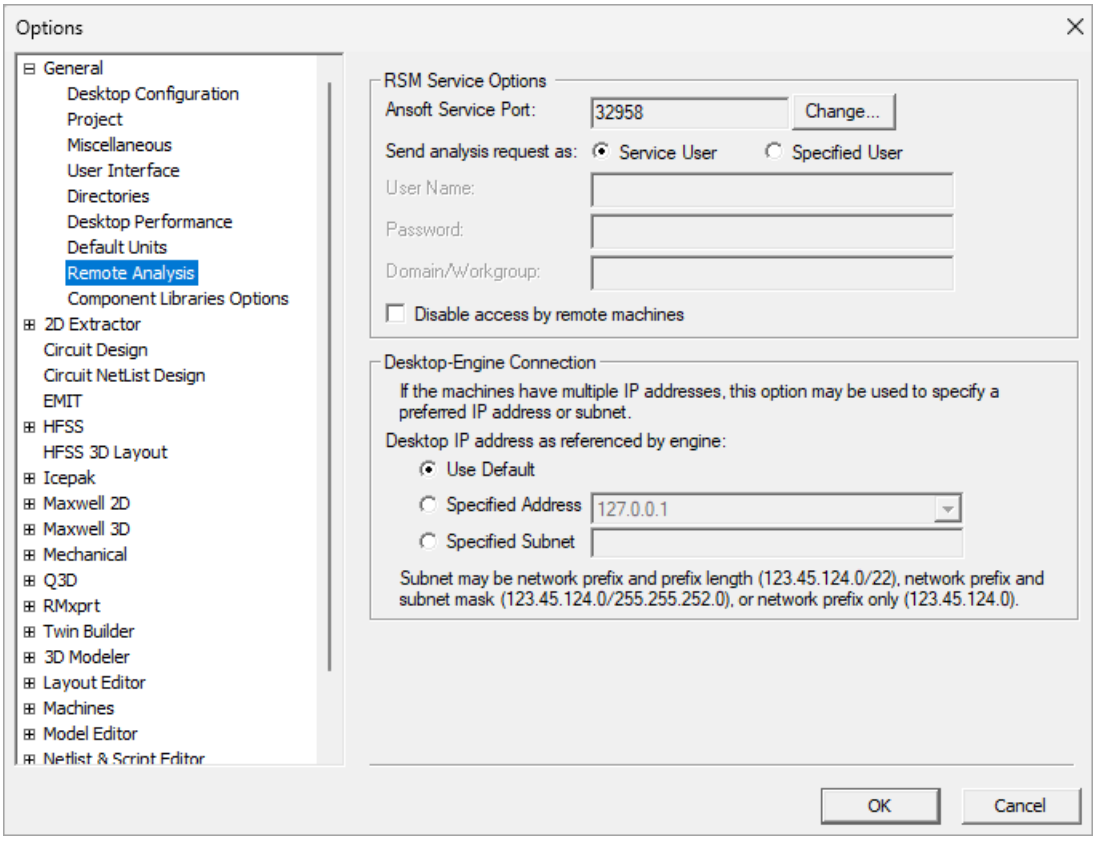
Configuring the Local Machine to Solve Remotely

To set Icepak 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 2026 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 Icepak

When running the Ansys Electronics Desktop on Linux, in order to solve Icepak designs using a remote Linux machine, you must configure the machines to share login credentials and enable an automatic login on the remote machine.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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 2026 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 2026 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 Start icon, search for **Register with RSM**, then select 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 icon, search for and then select **Register with RSM**.

Ansys Electromagnetics Suite 2026 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 2026 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.

> **Note:** When using distributed analysis, the Temp folder must be accessible by all machines set up in the HPC configuration.

Icepak 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.

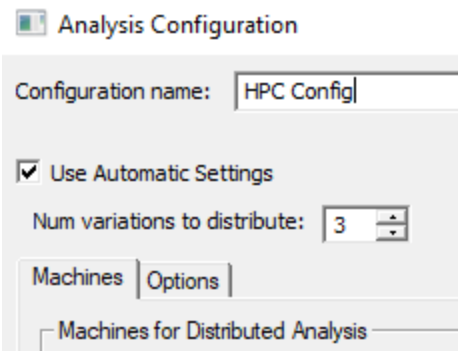
All types of distribution use MPI, except when only distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line. MPI may also be used if Auto mode is specified, and rows of a parametric table are distributed. MPI must be correctly configured if 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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

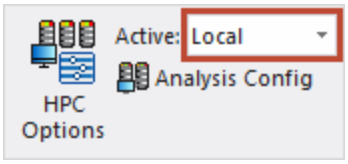
Configure a Distributed Analysis

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.



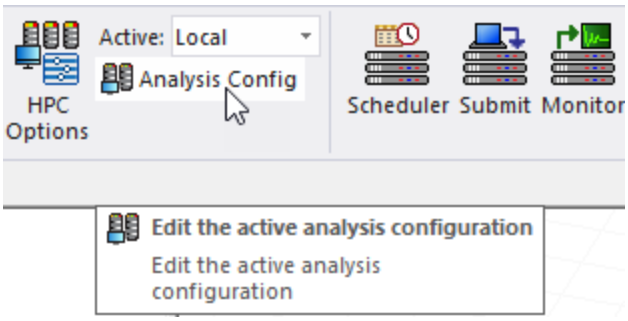
Editing Distributed Machine Configurations for Icepak

To create a new distributed machine configuration, or edit an existing machine configuration.

> **Note:**
>
> For more information about how HPC options and settings improve the efficiency of Icepak processes, see High Performance Computing in Icepak.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

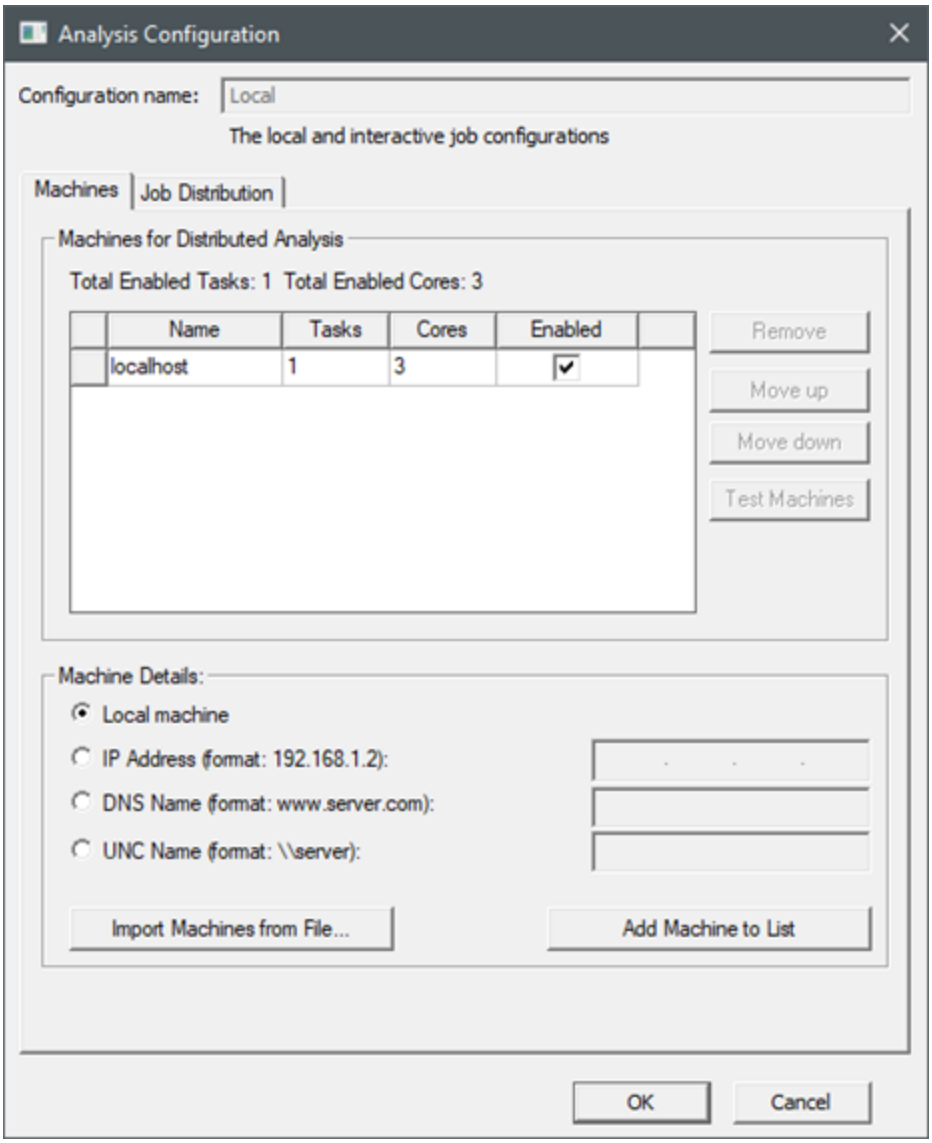
1. Click **Tools** > **Edit Active Analysis Configuration** to open the **Analysis Configuration** dialog directly or click **Analysis Config** on the **Simulation** tab of the ribbon toolbar.



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 2026 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 2026 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.

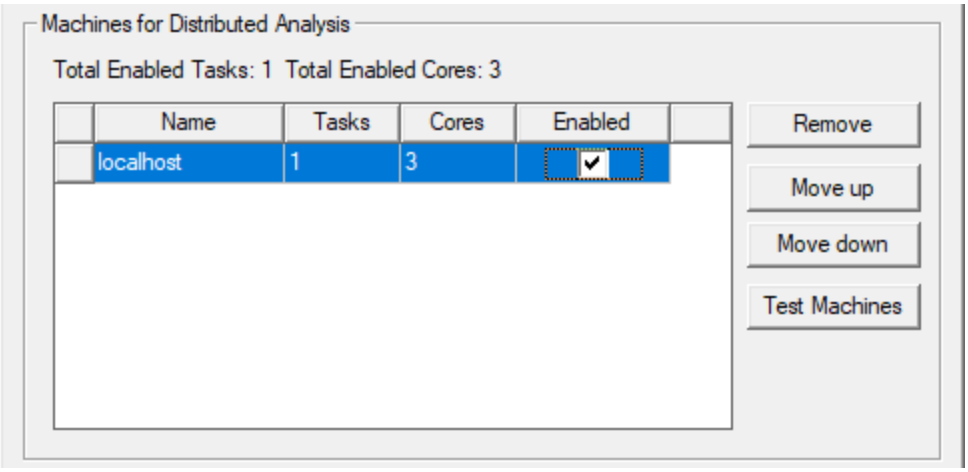
Local Machine Radio Button

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Icepak uses the **Tasks** setting to control the number of concurrent meshing processes for handling multiple mesh regions. For single-level job distributions, when Optimetrics is enabled under the *Job Distribution* tab, the number of Optimetrics variations to solve in parallel is also controlled by this setting.
- The **Cores** setting is used by the solver only during the **mesh processing** stage. Note that, during the **solution iterations** stage, the Icepak solver automatically chooses the number of cores to balance the computational resources on the machine. The *Cores* setting has no effect on this allocation.
- The number of cores you specify must be equal to or greater than the number of tasks you specify.

For distributed tasks in Icepak, 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. 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 to machine 1 and assign 1 task to machine 2, giving all tasks the same number of cores (4).

If you select a distributed configuration (rather than Local) from the Toolbar menu and you launch multiple analyses from the same UI. Icepak 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, Icepak chooses the first machine in the list. If another simulation is launched while the previous one is running, and this simulation requires two machines, Icepak chooses machines 2 and 3 from the list. If the first simulation then terminates and we launch another simulation requiring three machines, Icepak chooses 1, 4, and 2 (in that order).

Test Machines

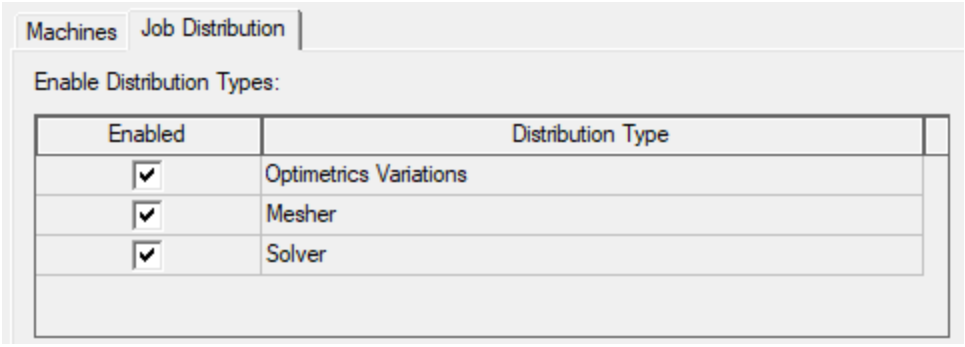
When multiple users on a network are using distributed solve or remote solve, they should check the status of their machines before launching a simulation to ensure no other Ansys EM processes are running on the machine. To do this, you can select one or more machines and click **Test Machines**. A **Test Machines** dialog box opens.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

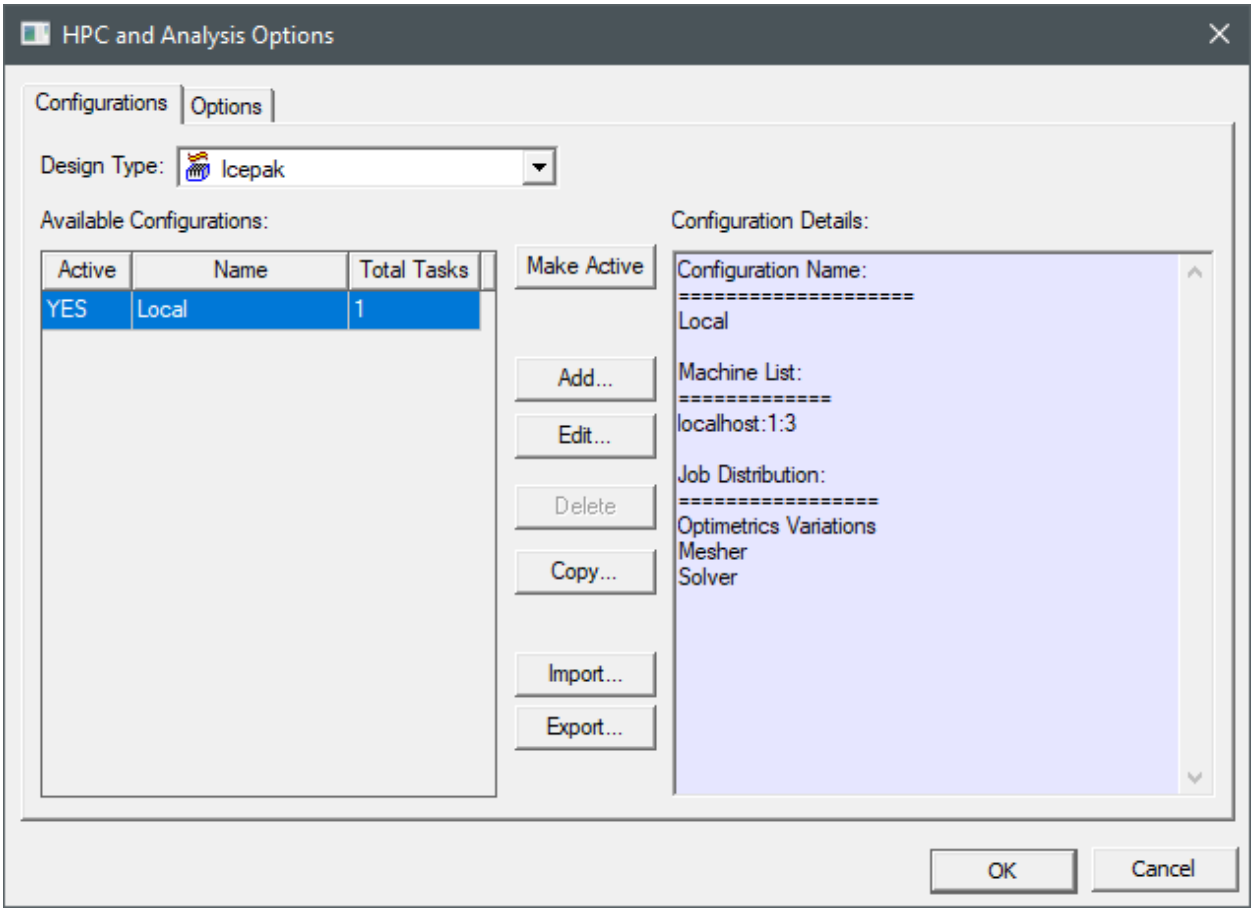
When using job distribution for meshing, you must specify the optimal number of **Tasks** on the **Machines** tab. The number of tasks required is the number of mesh regions (including the global mesh region) plus one task for the mesh controller. For example, if you've created three mesh regions in your model, specify at least five tasks: three for the mesh regions, one for the global mesh region, and one for the mesh controller. For this example, if you specify more than five tasks, the Electronics Desktop ignores the extra tasks. Conversely, if you specify fewer than five tasks, one or more mesh regions are meshed serially rather than concurrently.

> **Note:**
>
> Mesh times are reported on the Profile tab in the Solutions dialog box. See Monitoring the Solution Process for more information.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

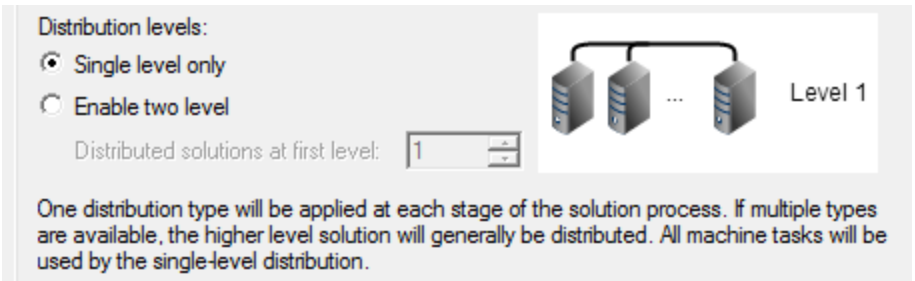
The distribution types that you enable here are listed in the **HPC and Analysis Options** dialog box in the *Configuration Details* section. To access this dialog box, click **Tools** > **Options** > **HPC and Analysis Options** from the menu bar or click the **HPC and Analysis Options** command on the **Simulation** ribbon tab.



For products that support two-level distribution, when the design is appropriate, you can turn on two level distributed solutions, and specify the number of engines to use for level 1. An example design that could use two level distribution would be an array with frequency sweeps.

Distribution Levels (for Icepak)

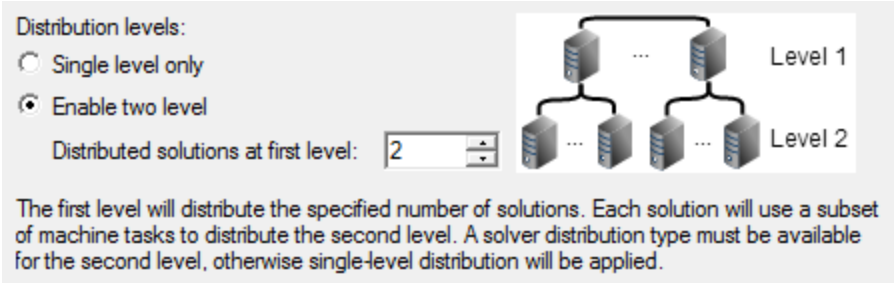
The radio buttons let you enable/disable two level distribution.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select single level, one distribution type will be applied at each stage of the solution process. If multiple types are available, the higher level solution will generally be distributed. All machine tasks will be used by the single-level distribution.

Selecting **Enable two level** enables the **Distributed Solutions at first level** selection box.



If you select **Enable two level**, the first level will distribute the specified number of solutions. Each solution will use a subset of machine tasks to distribute the second level. A solver distribution type must be available for the second level, otherwise single-level distribution will be applied.

The **Preview Job Distribution Setup** menu and field lets you view how the selected setup will be distributed.

Distributed Solutions at First Level 1

This control determines how many level 1 tasks to create during a two level distribution. This indirectly determines how many level 2 tasks for each level 1 task are used: the total number of tasks is specified by the list of enabled machines on the first tab, and the software evenly distributes resources among the L1 tasks which then are used to spawn off level 2 tasks.

Example:

- You create a machine list with 20 enabled tasks.
- You enable Variations and Solver.
- You enable 2 level distribution with distributed solutions at first level = 3.
- You solve a parametric analysis.

For the parametric analysis, the level 1 distribution type is Variations and the level 2 distribution type is Solver. Since the number of level 1 tasks = 3, three variations are launched simultaneously. The number of enabled tasks is 20. Those 20 tasks will be split as evenly as possible over the 3 variations. For the solver, Variations 1 & 2 will each get 7 tasks (processes) and Variation 3 will get 6 tasks (processes) to use for Solver distribution.

Ansys Electromagnetics Suite 2026 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.

> **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, including when to use two-level distribution. 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.

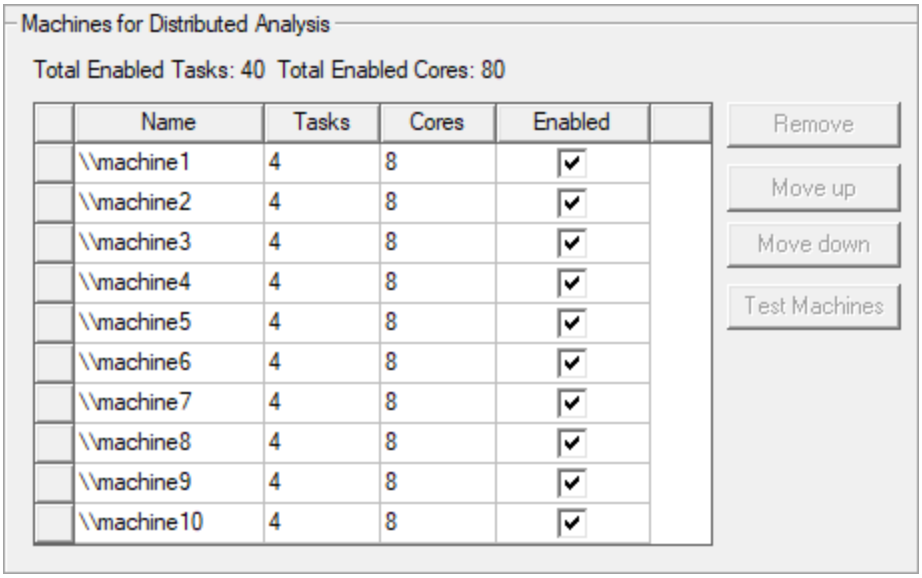
Two-Level Distribution Guidelines

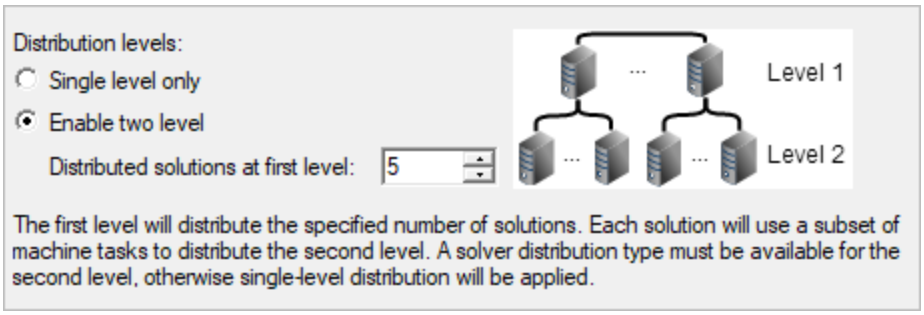
Ansys recommends that you focus resources for the highest level of distribution (level one). In other words, allocate the resources so that a maximum number of frequencies or variations are solved in parallel. First, determine the minimum number of machines required to solve one

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

frequency or variation based on your prior knowledge of the type of model being solved. Then allocate the resources so that a maximum number of solutions can be solved in parallel.

For example, suppose that a given model requires at least 100 GB of memory to solve using DDM. Assume that you have 10 machines, each with 60 GB and 8 cores. One solution will require two machines for the DDM solver. This means that, at most, 5 frequencies can be distributed. Each DDM solution requires at least 3 tasks, but since more cores are available you should set 4 tasks per machine, resulting in 2 cores for each Domain . A machine configuration based on the machines and number of tasks described above is shown in the figures below.





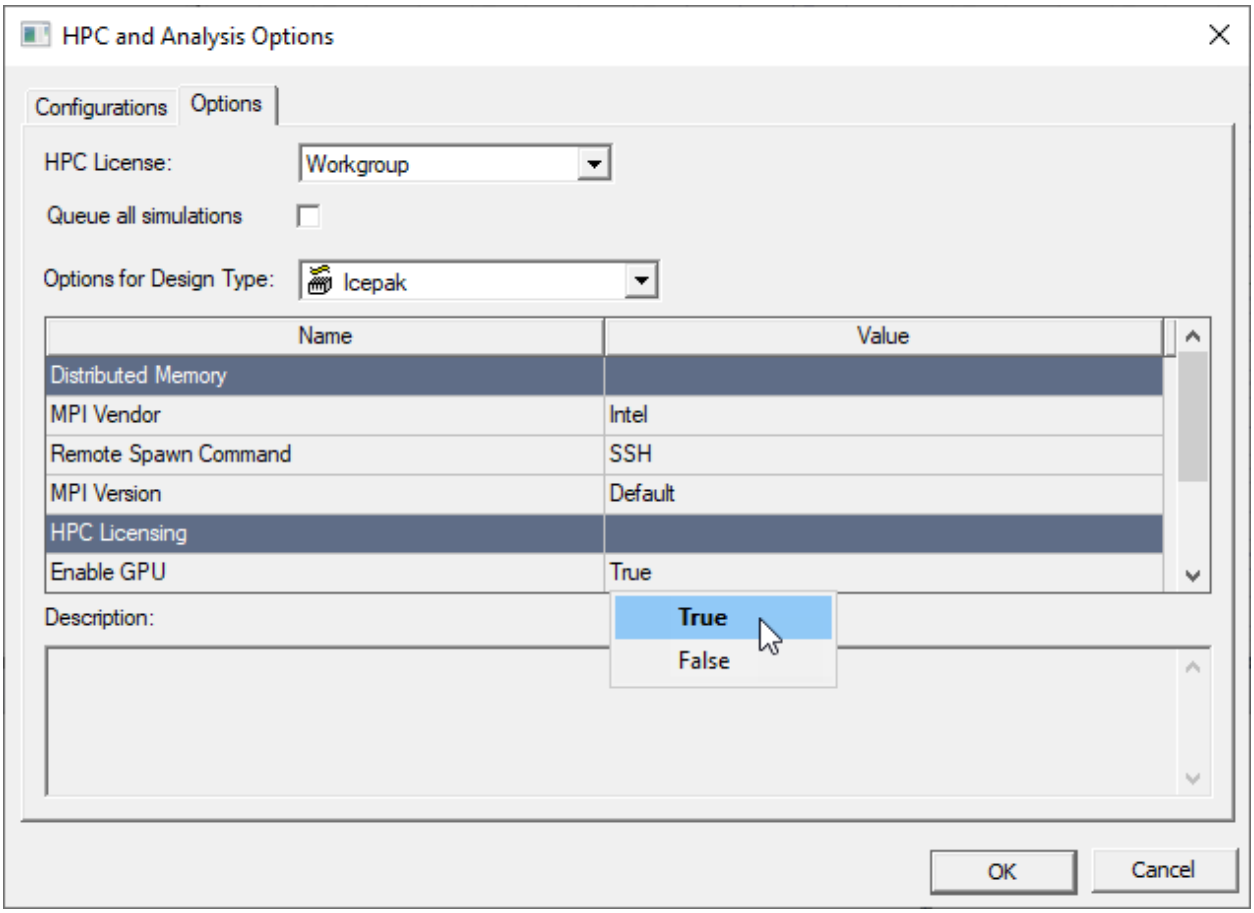
Enabling GPU (Beta)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The GPU feature appears in the user interface by default as an open beta feature. To use the GPU feature, you must enable the beta option. For information on enabling beta options, see General Options: Desktop Configuration.

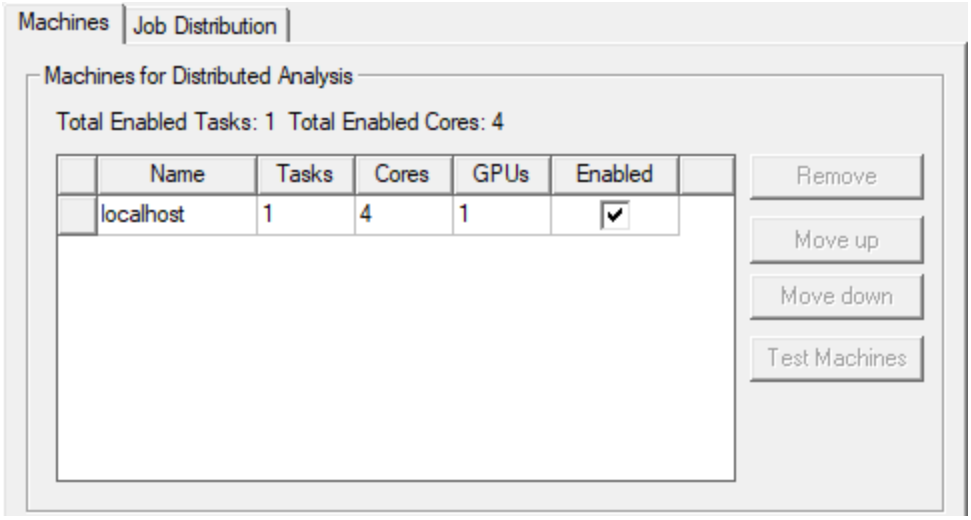
Ansys Electronics Desktop supports GPU acceleration for Icepak steady-state simulations.



To enable use of GPU, you can set **Enable GPU** to **True** and specify the number of GPUs in the **Analysis Configuration** window.

Note: If the number of GPUs specified is larger than the GPUs available on the machine, the solver automatically uses the maximum number of GPUs available. For example, if the machine has 4 GPUs, and the number of GPUs specified is 6, Icepak will use the maximum available GPUs on the machine (4 in this example).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: The GPU solver compute mode should be set to default when tasks greater than 0 are used for the GPU solver. To check the compute mode, open the Windows command prompt as admin and run `nvidia-smi.exe -q`.

To change compute mode, run the following command: `nvidia-smi -i 0 -c 0`

- `-i` = device id. Device id is an integer value. It correlates with number of GPUs present in the machine.
- `-c` = compute mode. Compute mode can be 0 or 1. 0 = default or 1 = exclusive mode.

GPU Limitations for Icepak Designs

Note:

CUDA version 12.8 or later is required to use the GPU solver to analyze designs with native joule heating enabled.

Certain features in Icepak do not support using a GPU. If the design has any features not supported by the GPU solver, Icepak automatically switches to the CPU solver. A warning message that includes the unsupported feature(s) is reported in the **Messages** window. Icepak designs containing the following objects or settings are automatically run using the CPU:

- PCB component
- PCB networks

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Internal fans with Swirl specified
- Network boundary condition
- No Reverse Flow option
- Shell conduction
- Joule heating
- Spalart Allmaras turbulence model
- Ray tracing radiation
- Solar loading
- UDFs

Troubleshooting GPU Acceleration for Icepak

In the **Message Manager**, if you receive a error stating that Electronics Desktop could not find the nvml.dll binary, resulting in CPU usage for the simulation, download and install the latest version of CUDA to obtain the drivers required for GPU acceleration.

To check GPU usage, open the Windows Command Prompt and enter nvidia-smi to display all processes currently running with the GPU.

To check GPU multi-processors and cores on your machine, open the command prompt and enter the following command to change the working directory:

cd C:\Program Files\NVIDIA GPU Computing Toolkit\CUDA*versionnumber*\extras\demo_suite

Open the command prompt in this folder and run deviceQuery.exe.

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 2026 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 an extreme case, some tasks may be hours behind the 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 2026 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 **Abort** from the **Abort Job** button drop-down menu on the Job Monitor dialog box.

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.

If the selected scheduler is **HPC Platform Services** or **Ansys Cloud Burst Compute**, the **Hard Abort** option will be available under the **Abort Job** button dropdown menu in the **Job Monitor** dialog. Use this option if the simulation is hanging and does not respond to **Abort**. Any intermediate files that can be collected will be placed in the downloaded results under archive **abort_intermediate_results.zip**.

Prerequisites for Large Scale DSO

General Prerequisites

- Ansys Electronics Desktop must be installed on the cluster that 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.

- 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`

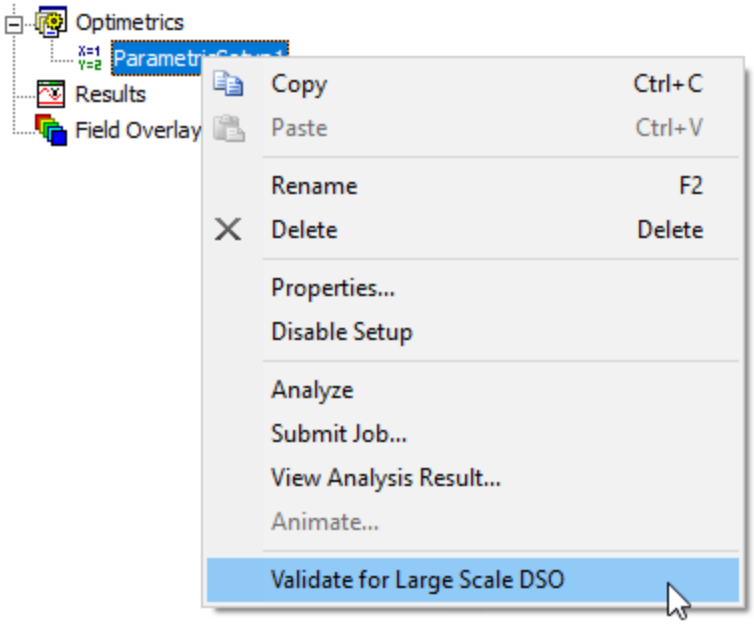
Job Management Interface for Large Scale DSO

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Correct any errors and save the project.
3. Close Electronics Desktop.
4. Copy the project (or folder, if the project references external files) from a personal workstation to a shared drive on cluster.

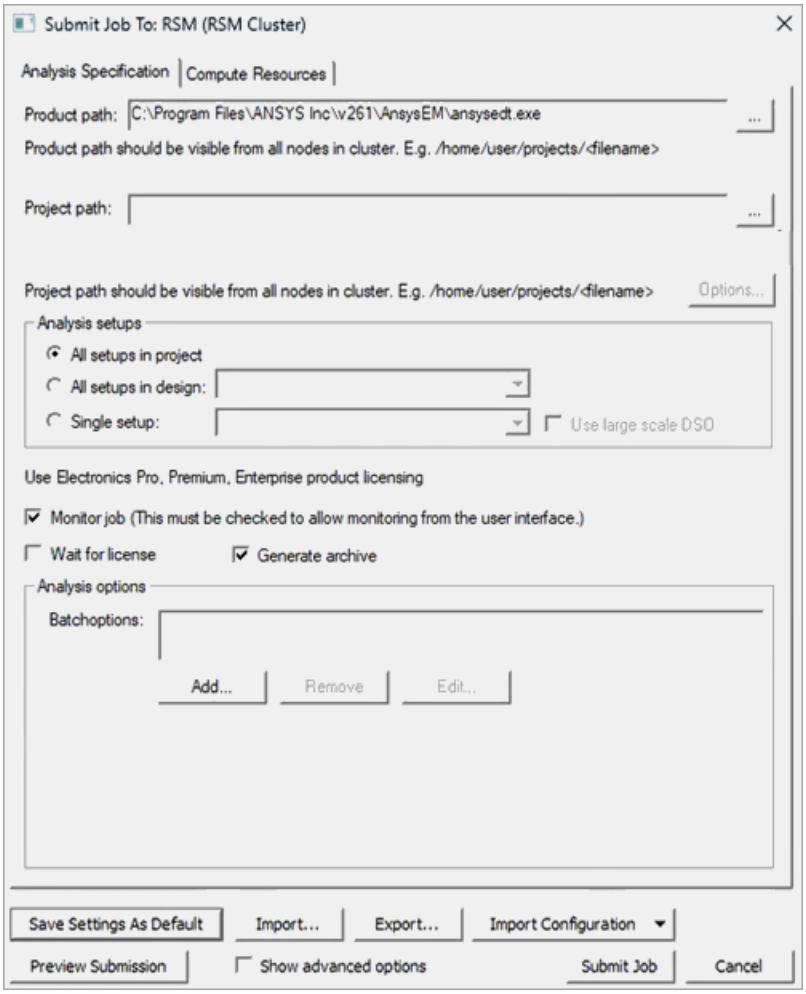
   In the RSM environment, you must specify a machine list (See: HPC and Analysis Options).

   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 **Icepak** > **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 2026 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.

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.

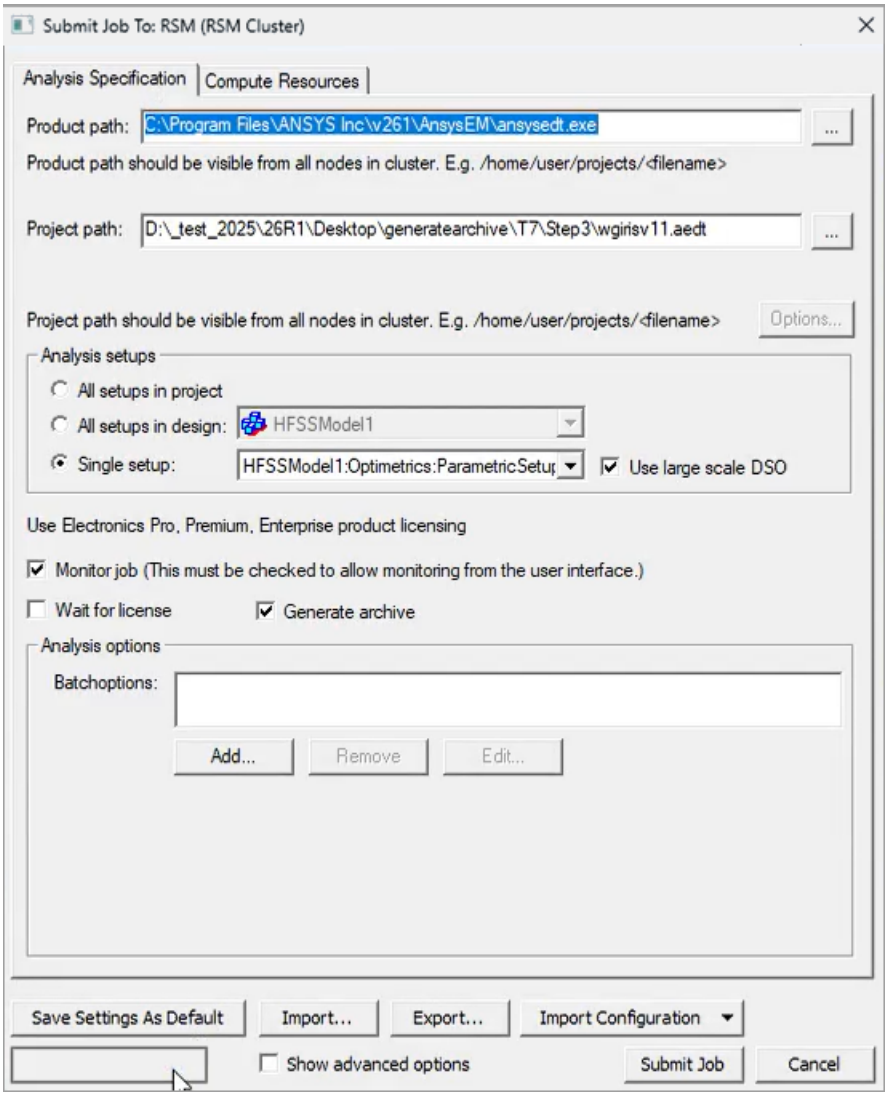
      For Large Scale DSO, select a **Single setup** from the drop-down menu and ensure the **Use large scale DSO** check box is selected.

    - 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Check **Generate archive** to create an archive for the submitted job when it is complete. The project file, batchinfo directory, any external files, and the results directory are combined into the archive. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc.*.. will be written instead. This function will not overwrite an existing archive.
  Note that if an archive file is selected, this option will be renamed **Repackage results into archive**.

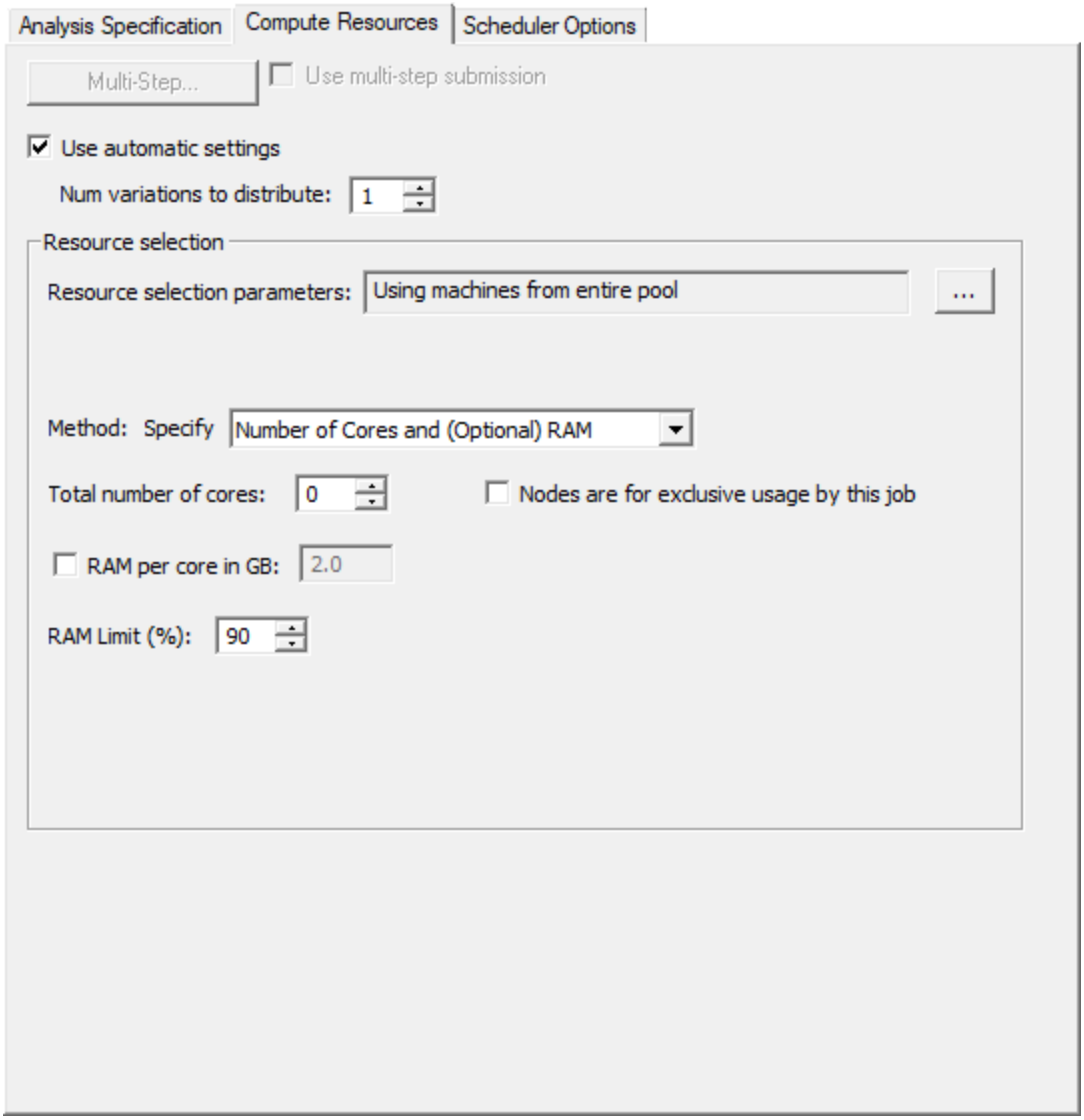
An example configuration is shown below:



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**.

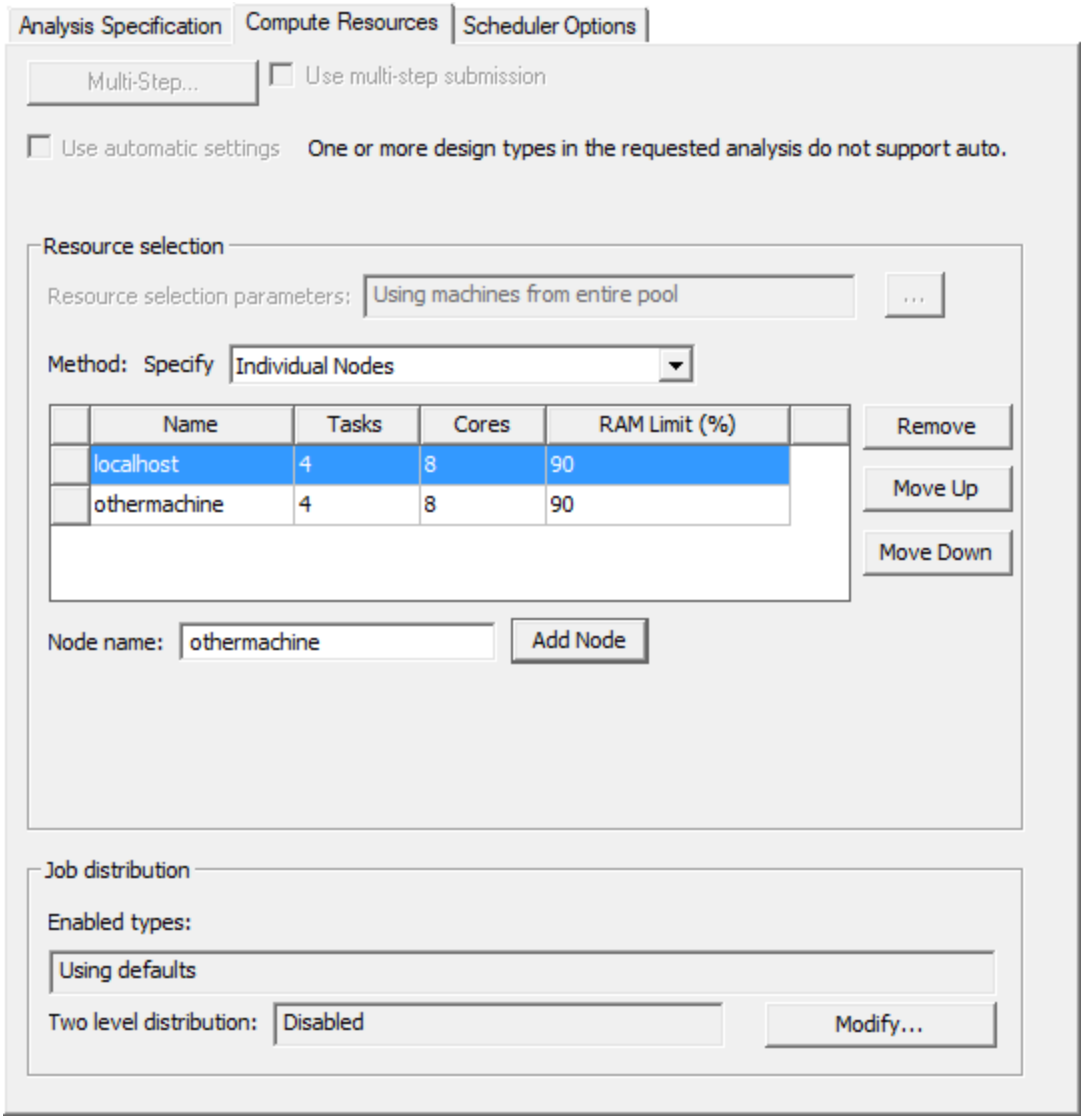
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



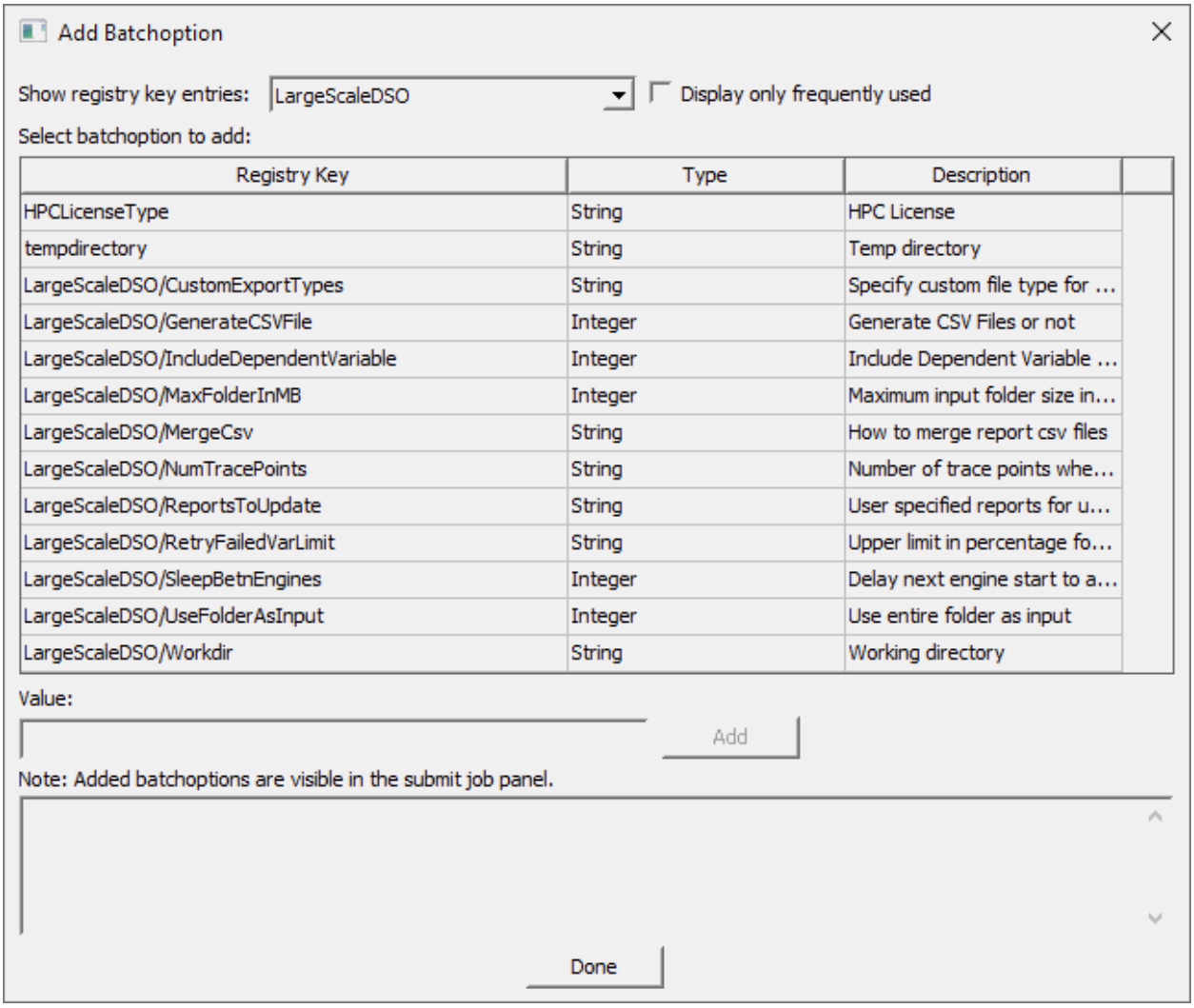
For RSM Large Scale DSO jobs submitted from the Job Submission panel, localhost must be the first node in the resource selection panel. Otherwise, the job will fail.

In the **Job distribution** area, you can enable or disable **Two-level distribution** by clicking **Modify**. See Two-level Distribution Guidelines.

- Add **Batchoptions**, if desired.

  The following shows batch options specific to Large Scale DSO.

Ansys Electromagnetics Suite 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

- `-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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `LargeScaleDSO/FailedVarRetryCount`: Retries failed variations.
- `FailedVarRetryCount`: Instructs each task to re-simulate its failed variations when its assigned variations are finished. Input is a positive integer. Default is 0; Value-Zero(0) will disable the re-simulation for failed variations. Value ranges from 0 or positive integer.
- `LargeScaleDSO/VarRedistribution`: Redistribution batch option; Value 0 disables redistribution (default), and Value 1 enables redistribution.
- `LargeScaleDSO/RedistributionLimit`: Specifies the minimum estimated remaining time (in minutes) for variations to redistribute to another task. Value must be a positive integer; the default is 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"
  ```

  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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-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.

- `-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.

- `-mergecsv: [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)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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'='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 that this option only applies if an archive file is provided as input.

- `-generatearchive`

  After the simulation is complete, create an archive containing the project and results files. Note that this option only applies if project file is provided as input.

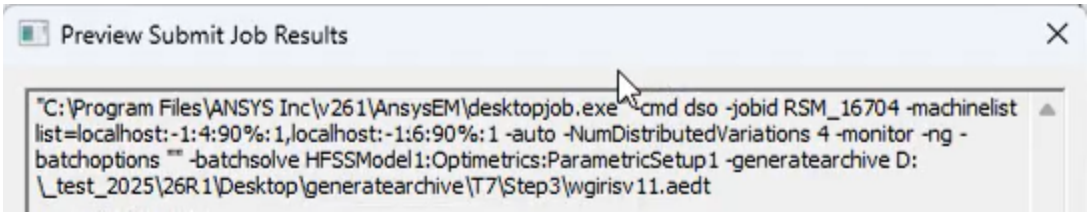
- **-batchsolve**

  Solves the specified parametric setup.

  Syntax for the setup:

  `<design-name>:Optimetrics:<parametric-setup>`

Below is an example of a desktop command to run a Large Scale DSO computation:



Ansys Electromagnetics Suite 2026 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 2026 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.

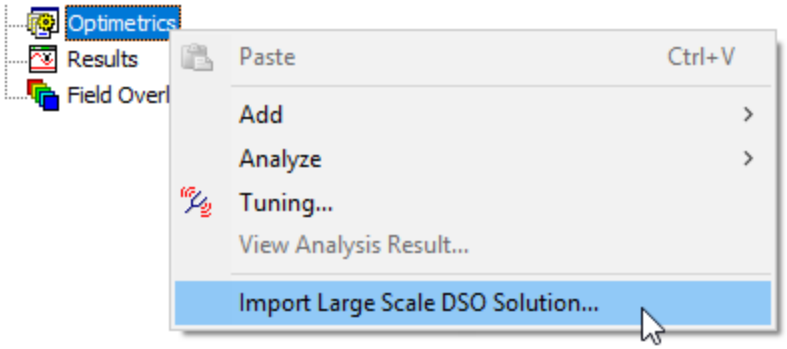
Postprocessing Large Scale DSO Dataset Solutions

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:

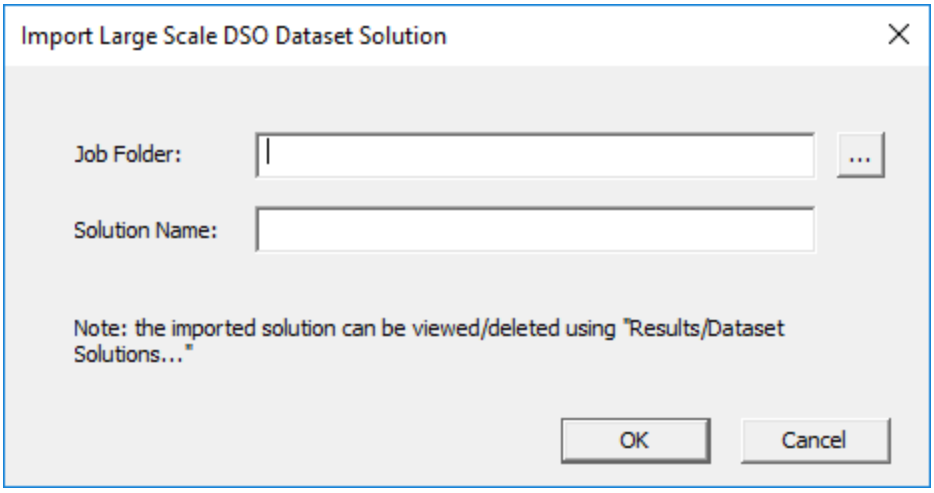
1. In the Project Manager, right-click **Optimetrics** and select **Import Large Scale DSO Solution**.



> **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 2026 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.

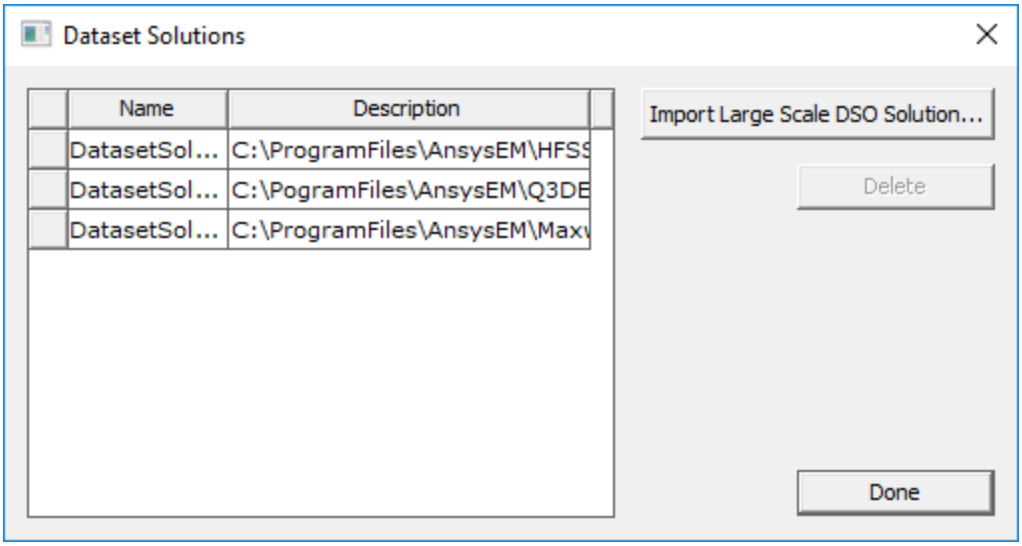
Viewing Imported Solutions

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 2026 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 2026 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 2026 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 2026 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 2026 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.

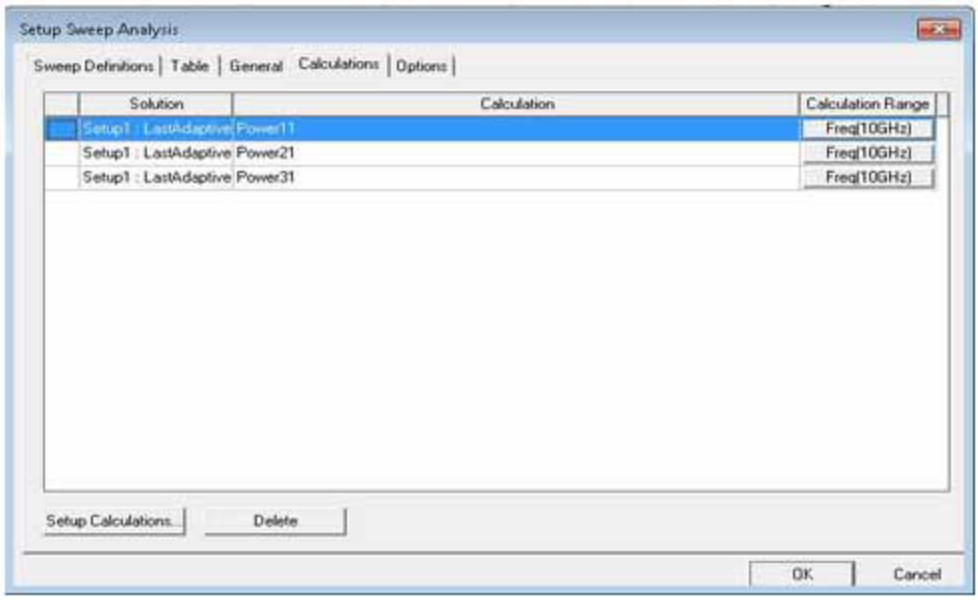
   For this example, use the existing 'ParametricSetup1' as the parametric setup to solve.

4. Follow the steps below to prepare the outputs.

   Outputs from Large Scale DSO come from predefined rectangular plots that are created *before* the Analysis command is issued. Non-rectangular plots of the design (digital plots, for example) are not extracted. The result data of the rectangular plots should be of datatype "double". Other result types (e.g. complex numbers) are not supported.

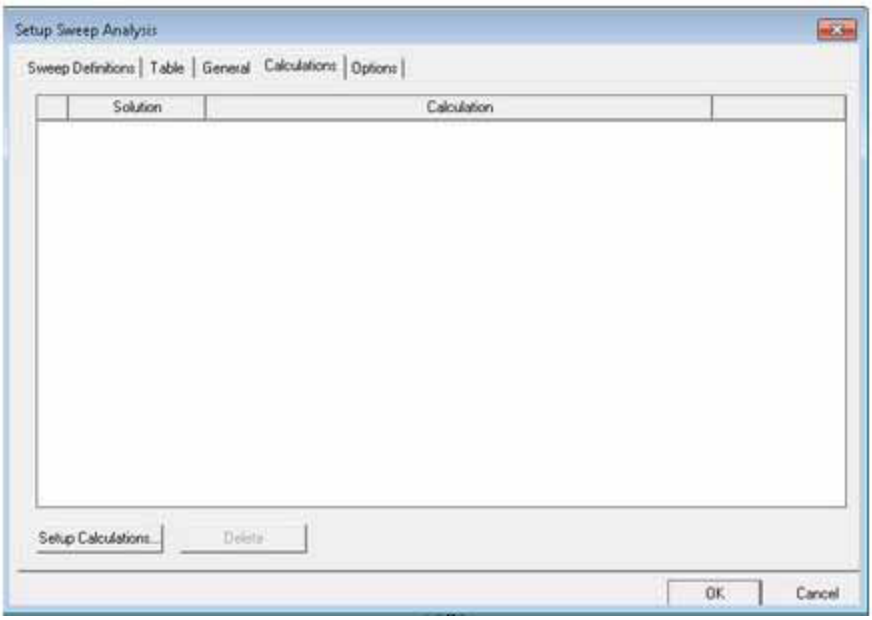
   a. Because these DSO outputs come solely from Rectangular Plots, delete all other postprocessing setups, and then turn off Save Fields And Mesh as shown below.

      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 2026 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:

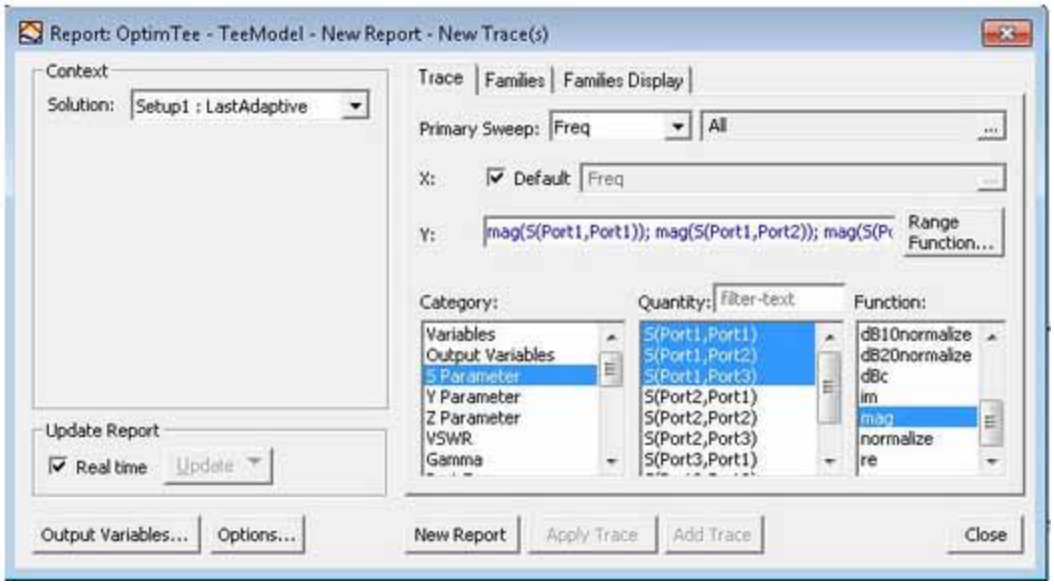


5. Use the Reporter to define outputs.

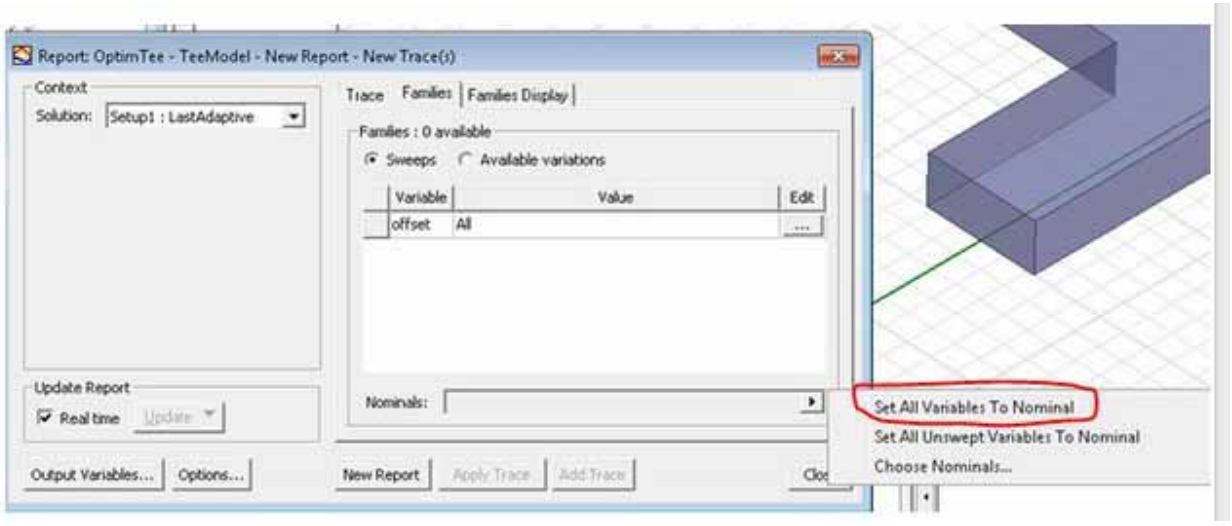
   For this example, you add six traces that correspond to six csv outputs of the Large Scale DSO job: mag(S11), mag(S12), mag(S13), Power11, Power12, and Power13.

Ansys Electromagnetics Suite 2026 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.

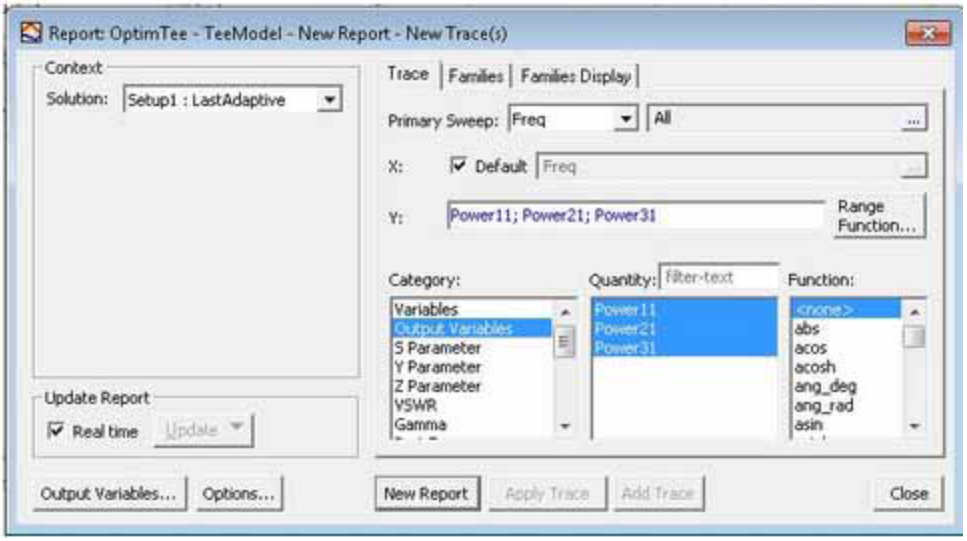
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

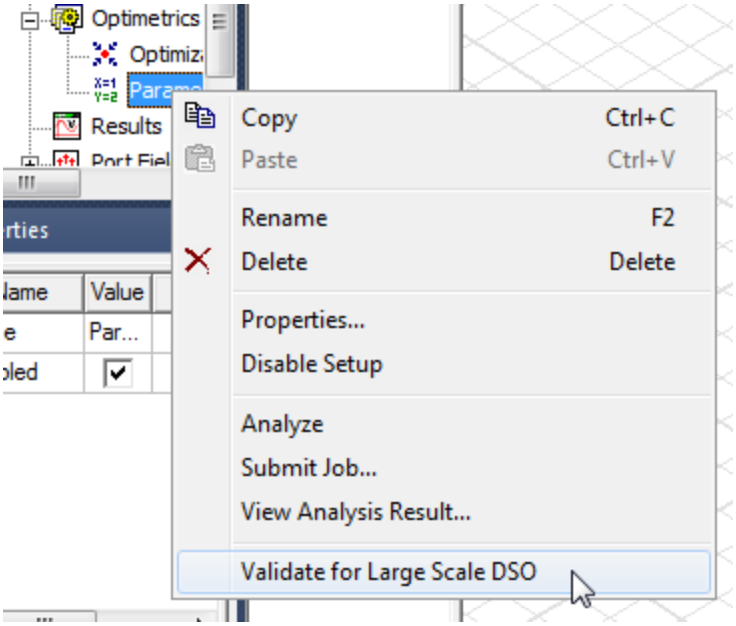


f. Click on the **Families** tab and ensure that all variables are set to **Nominal**.

g. Click the **New Report** button.

Ansys Electromagnetics Suite 2026 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 2026 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\v261\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\v261\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 " 'Icepak/NumCoresPerDistributedTask'=11`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
'Icepak/'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 " 'Icepak/'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 " 'Icepak/'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

Next

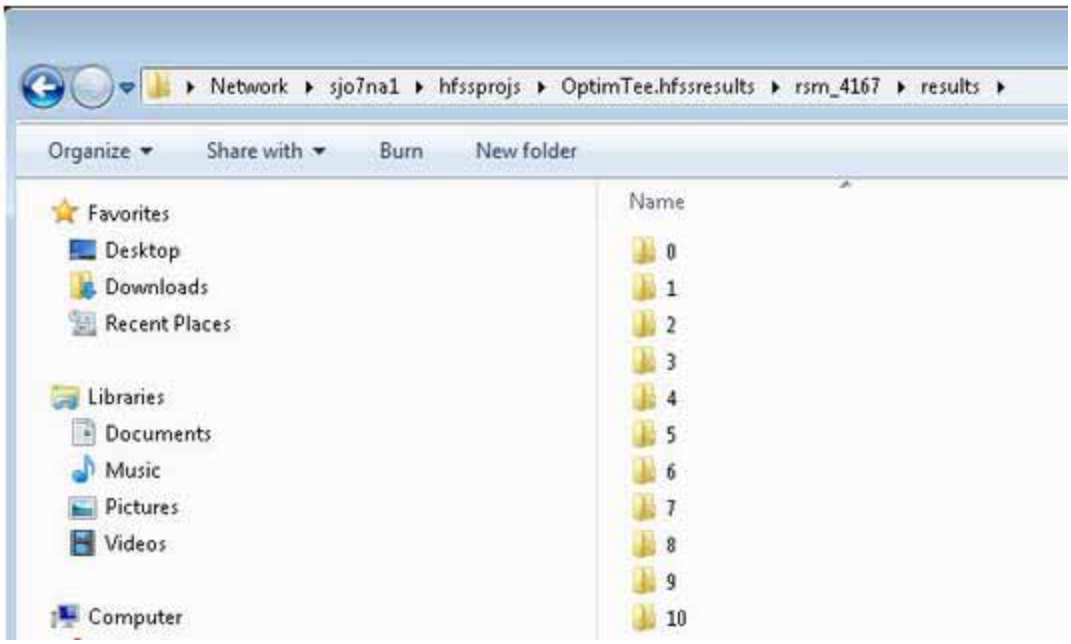
Large Scale DSO Example: Post Process the Results

Large Scale DSO Example: Post Process the Results

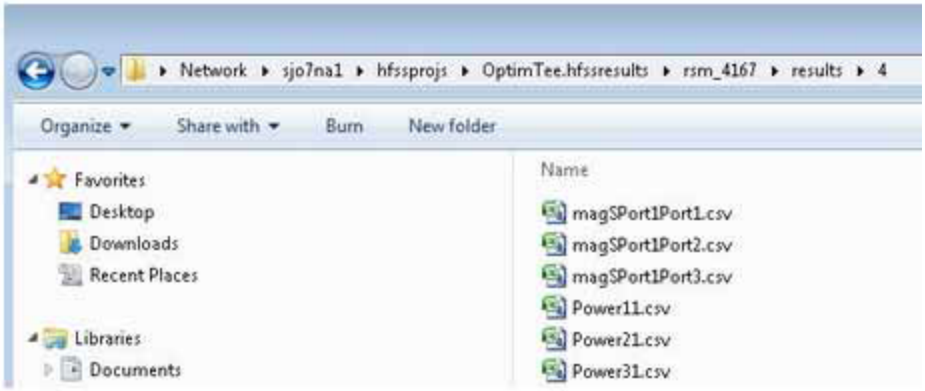
Once the job is done, output is available in the ~\OptimTee.aedt\jobid\results folder. Each variation creates a subfolder, which in turn has one csv file per trace of each report. See the detailed information regarding job monitoring and the location of the analysis logs.

The following figure shows the results for 10 variations as located in 10 folders.

Ansys Electromagnetics Suite 2026 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.

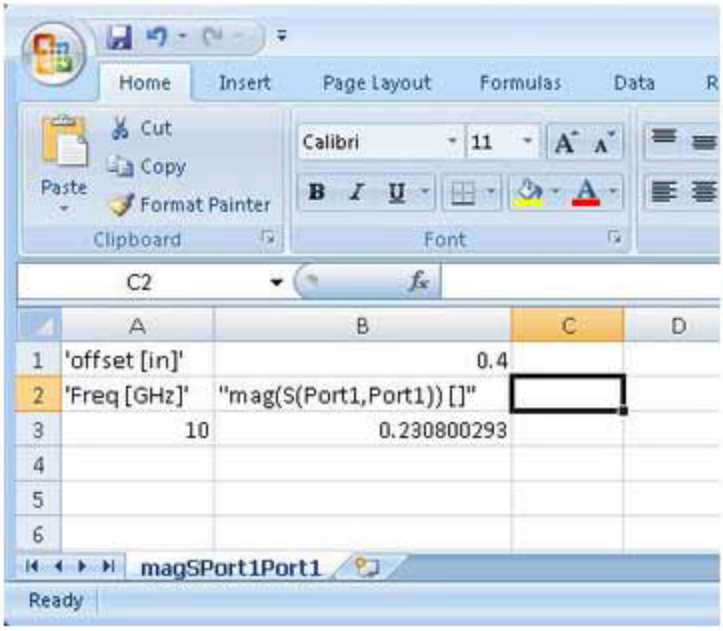


You have three options for postprocessing csv files.

- Import Large Scale DSO Dataset Solution
- Use Microsoft Excel or any other application that has csv post processing functionality.

Ansys Electromagnetics Suite 2026 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 2026 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 2026 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 2026 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 2026 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\2023\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\2023\LS_
DSO\OptimTee.aedt
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

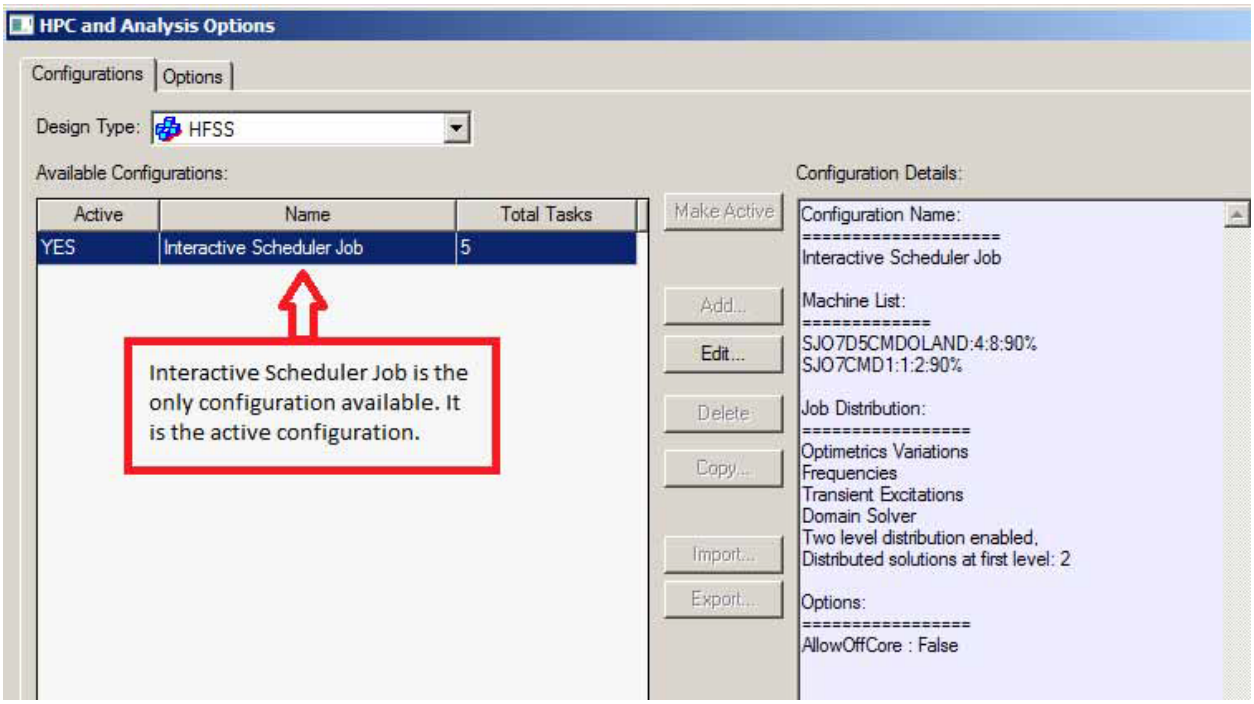
DSO Configuration for Interactive Scheduler Jobs

> **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** window. 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** window.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



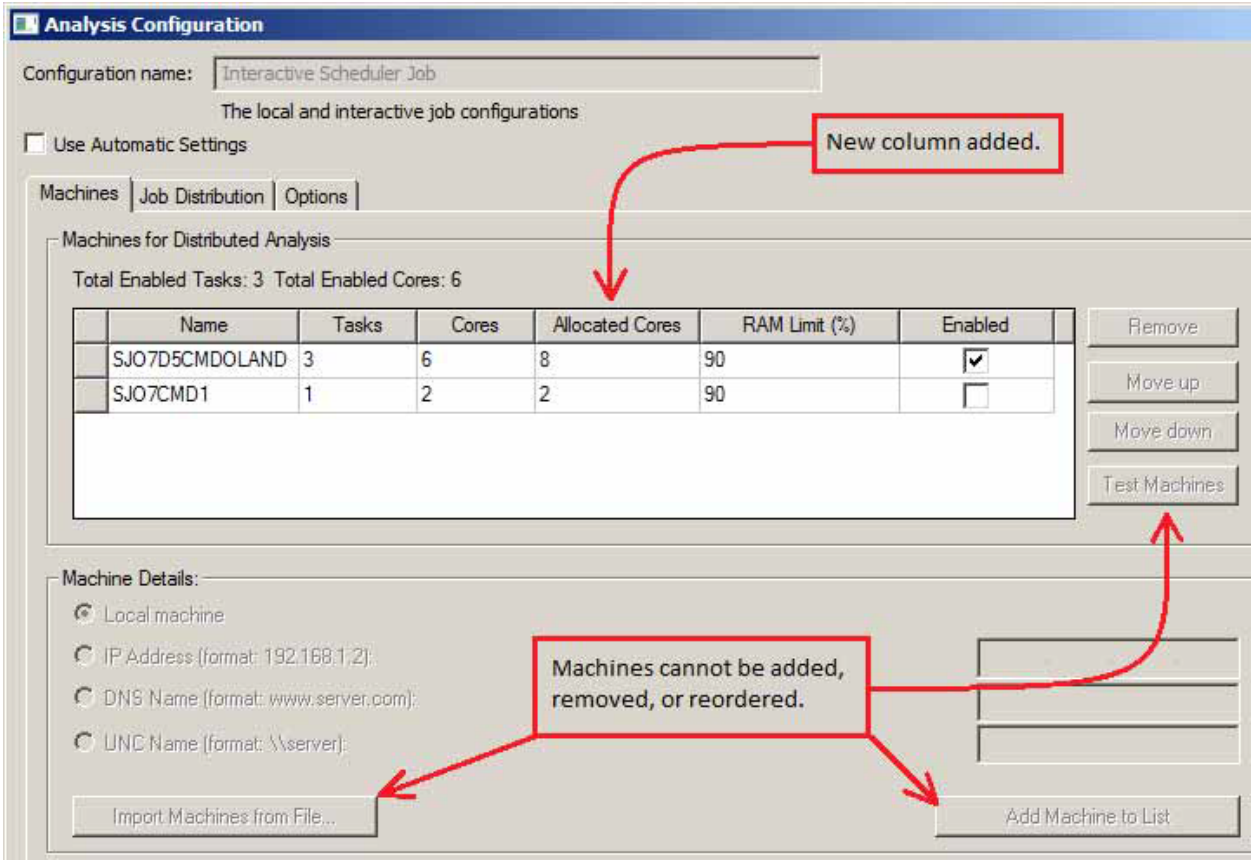
One major difference between interactive scheduler mode and batch mode or normal interactive mode is:

- 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 2026 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.

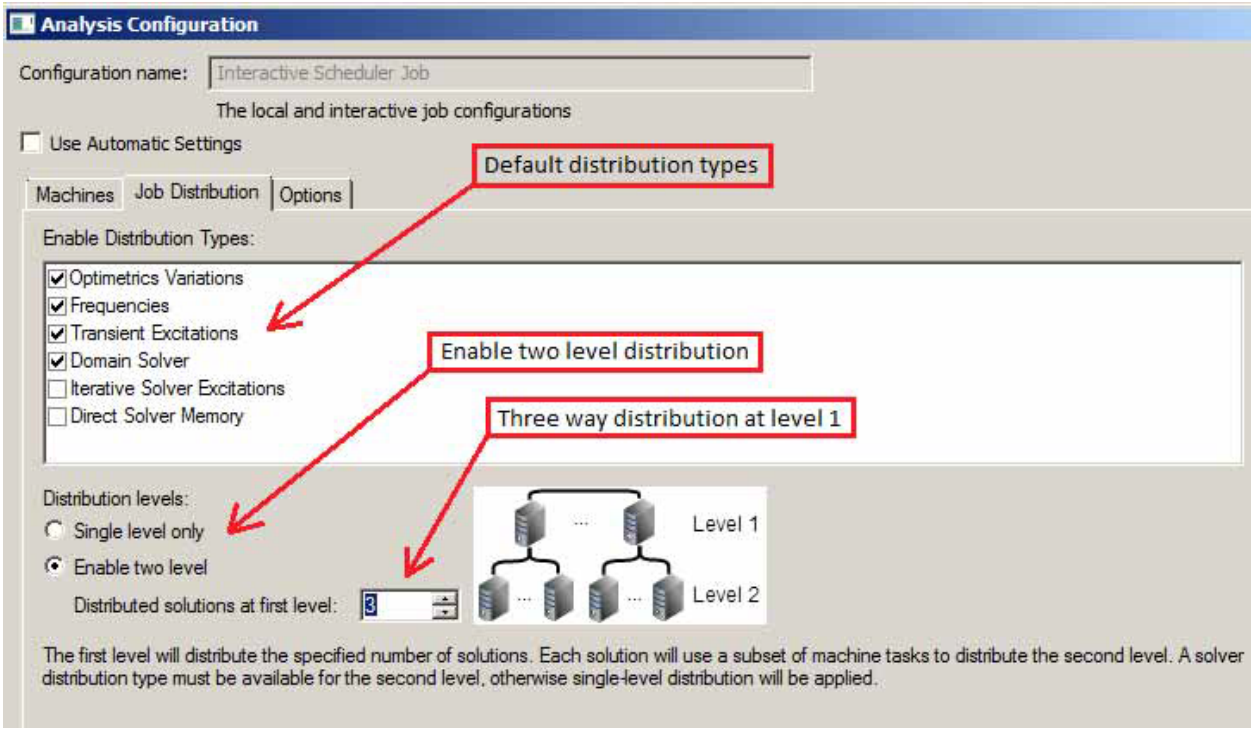
Example

Ansys Electromagnetics Suite 2026 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.

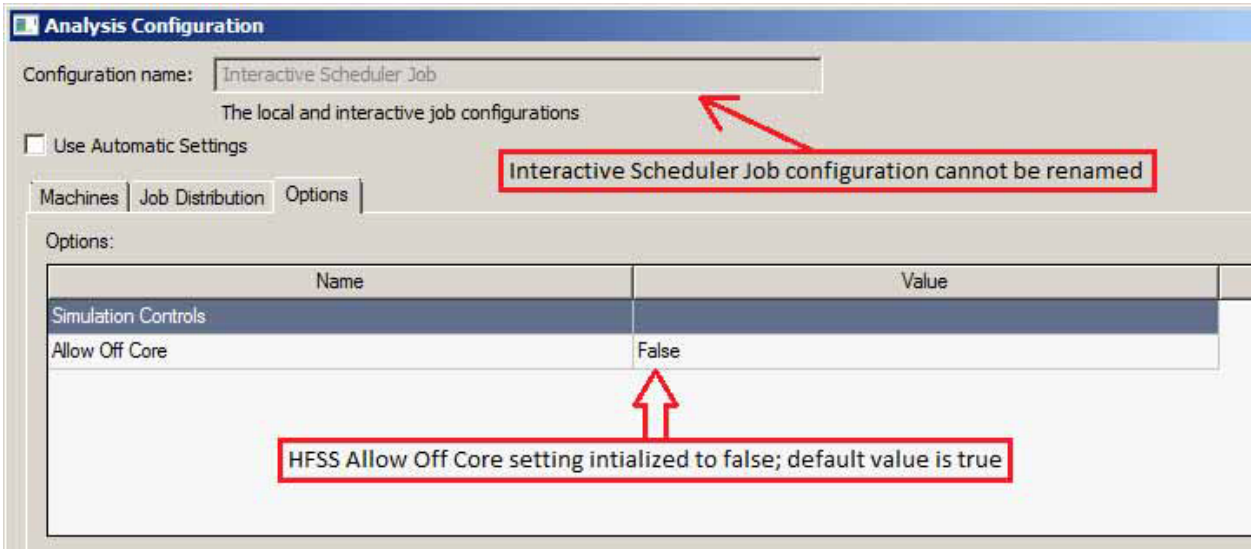
Example

Command line:

```
ansysedt -batchoptions 'HFSS/AllowOffCore'=0
```


Ansys Electromagnetics Suite 2026 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 options 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 2026 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 Burst Computing
- 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 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`
Machine/Host/Node	Consists of one or more processors, memory, disk, etc.
Multiprocessing	A single engine uses multiple cores on the same machine.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Term	Definition
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.

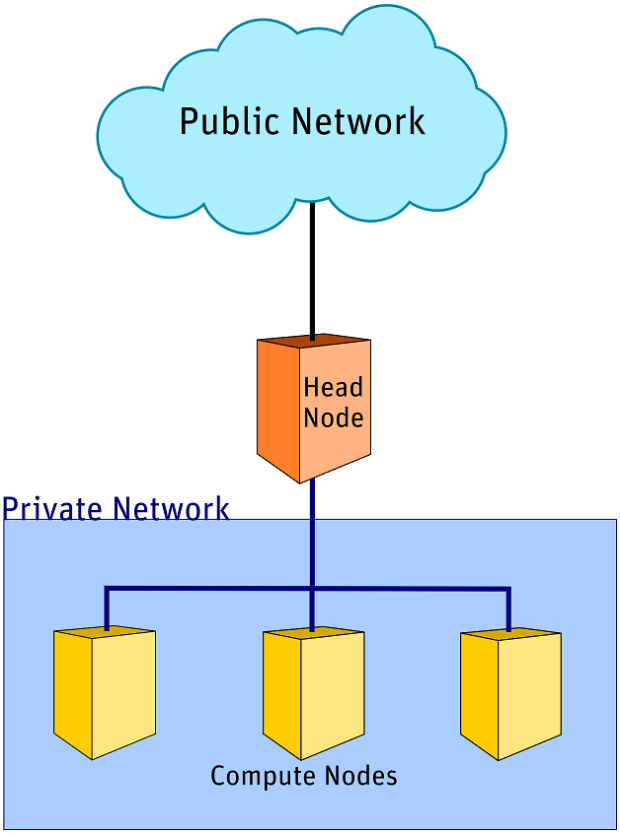
What a Scheduler Does

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 2026 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 2026 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\v261\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/v261/AnsysEM on each execution node of the cluster.

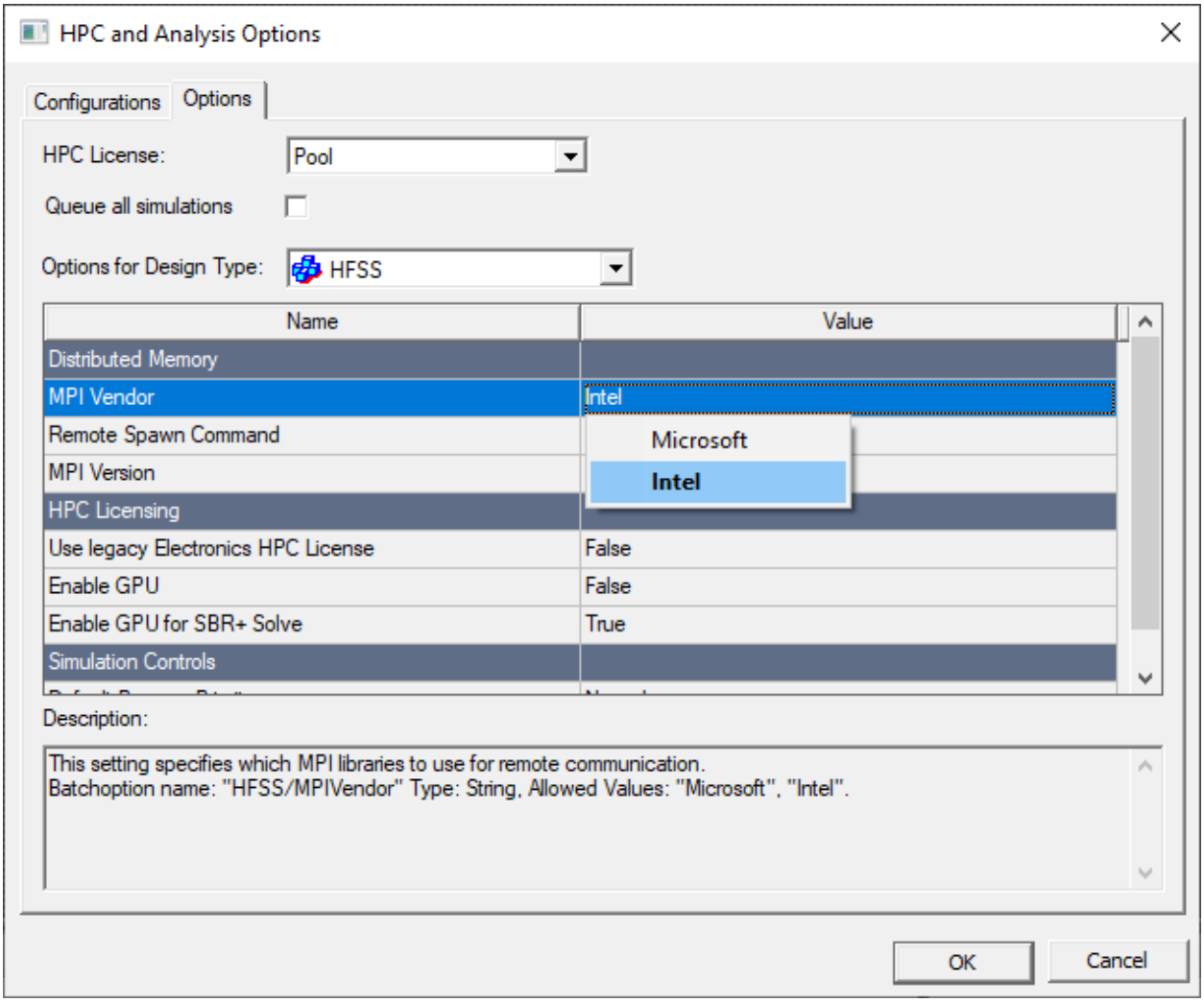
MPI Vendor and MPI Version

For HFSS, HFSS 3D Layout, Q3D Extractor, Maxwell, and Icepak design types:

These settings may be used to specify the MPI Vendor and Version for distribution via MPI.

Go to the **Options** tab of the **HPC and Analysis Options** dialog box, and select the design type. Then you can select the MPI Vendor and MPI version.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



MPI Vendor

The default MPI Vendor is "Intel". On Linux, "Intel" is the only supported MPI vendor. On Windows, both "Intel" and "Microsoft" are supported MPI vendor options. For jobs distributed to multiple hosts, the "Microsoft" MPI Vendor is only supported when running as a Windows HPC job.

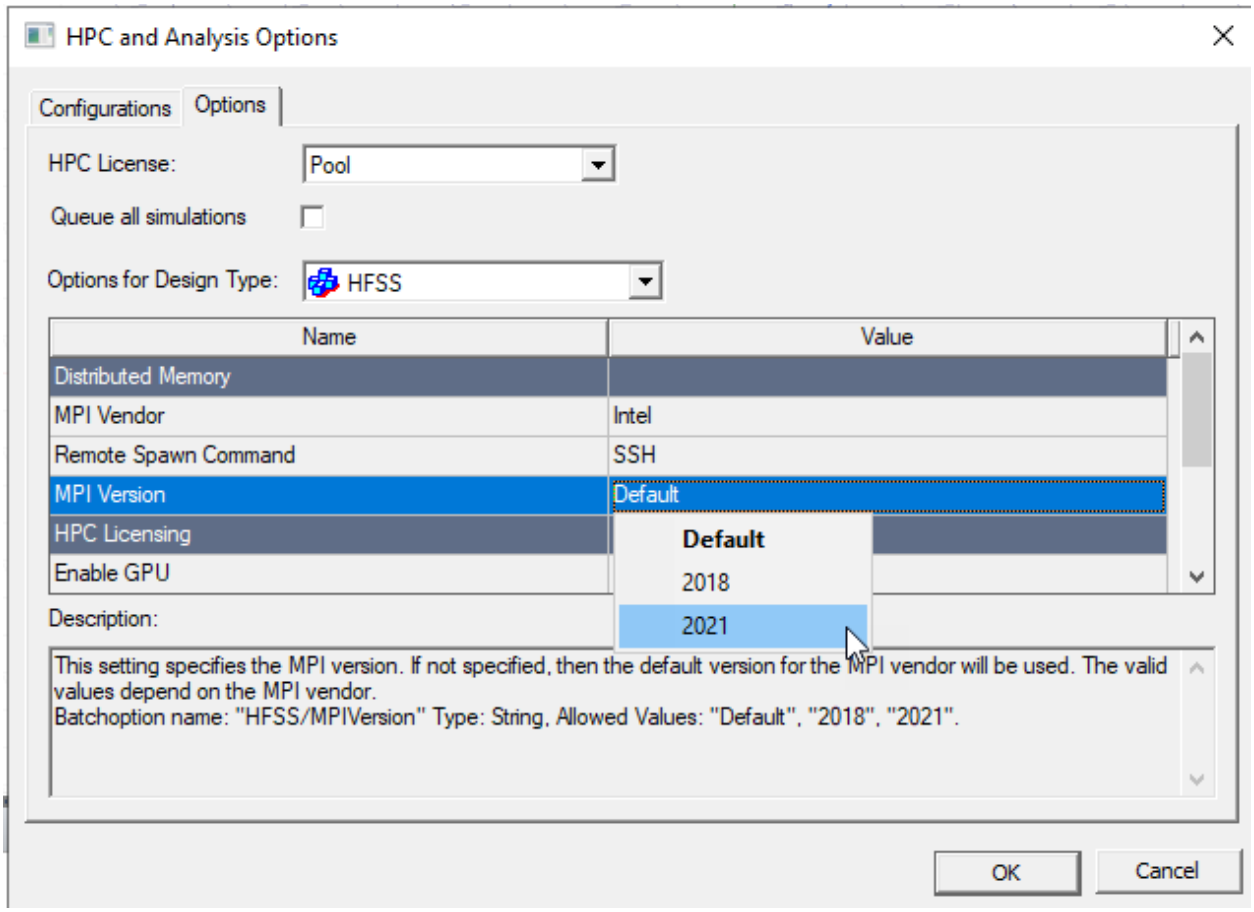
MPI Version

If only a single MPI Version is supported for the selected MPI Vendor, then the MPI Version option is ignored. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

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 2026 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 2026 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 2026 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 2026 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.

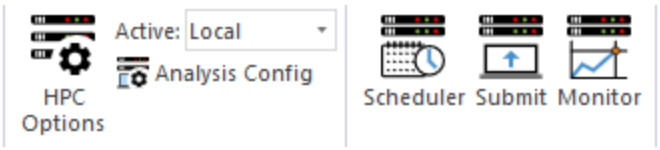
Job Submission User Profile on Cluster Compute Nodes

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

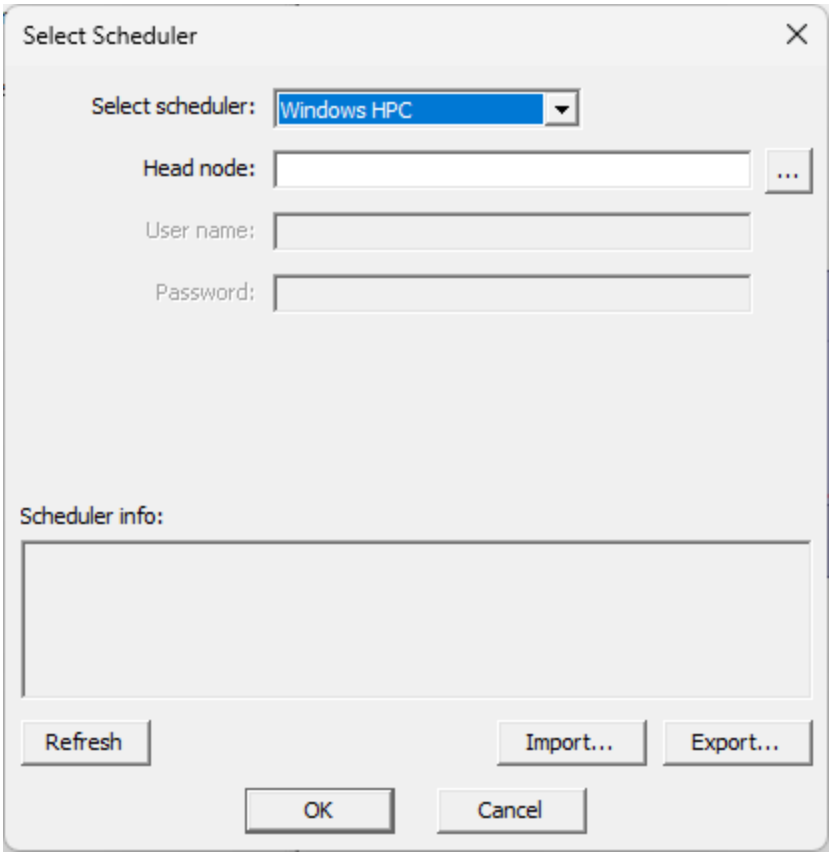
In order to submit jobs using **Windows HPC**:

- 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 2026 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 **Icepak** > **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.

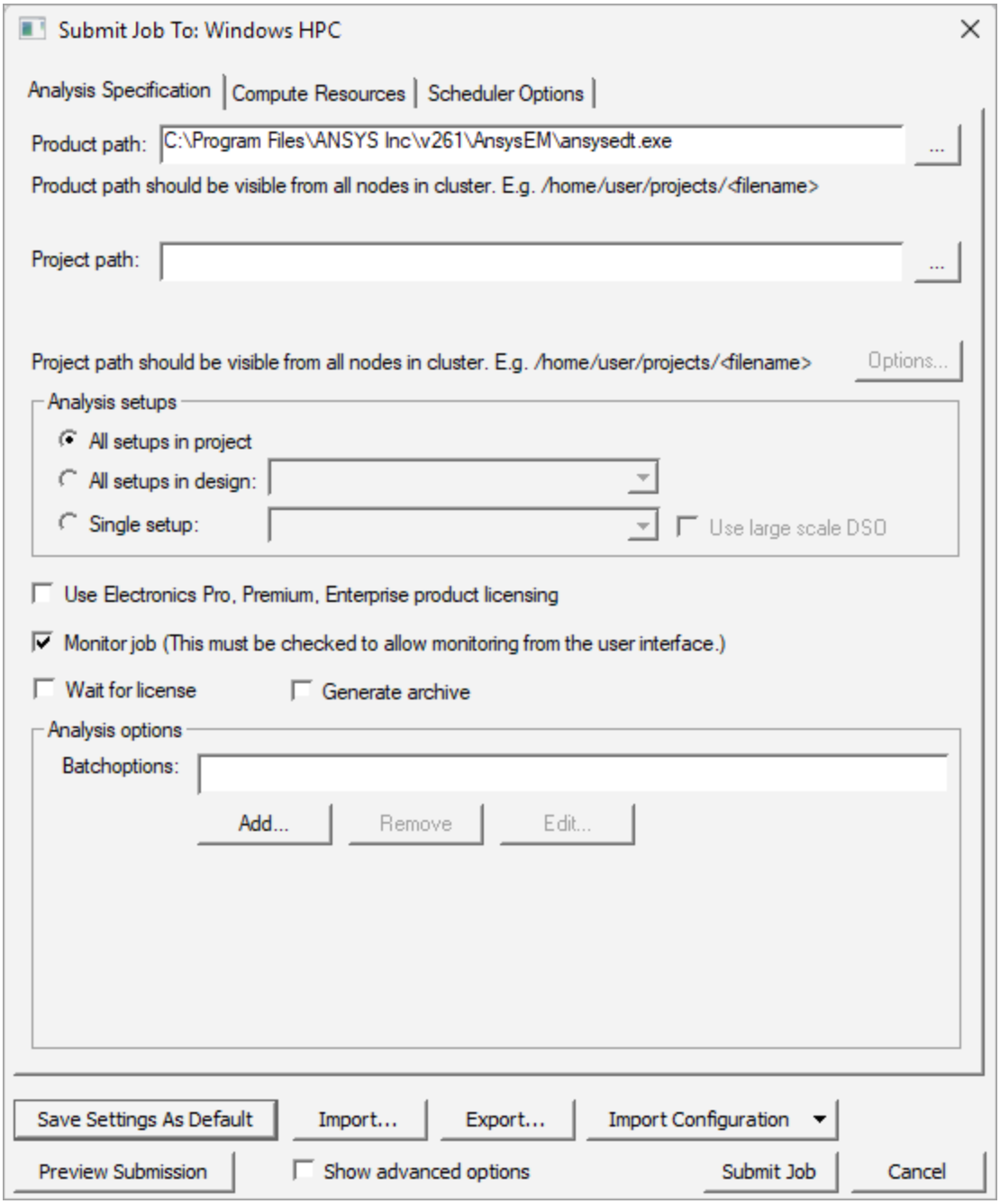
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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 one of the following:

- **All setups in the project**
- **All setups in a specified design** (you select the design from the drop-down menu)

Ansys Electromagnetics Suite 2026 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

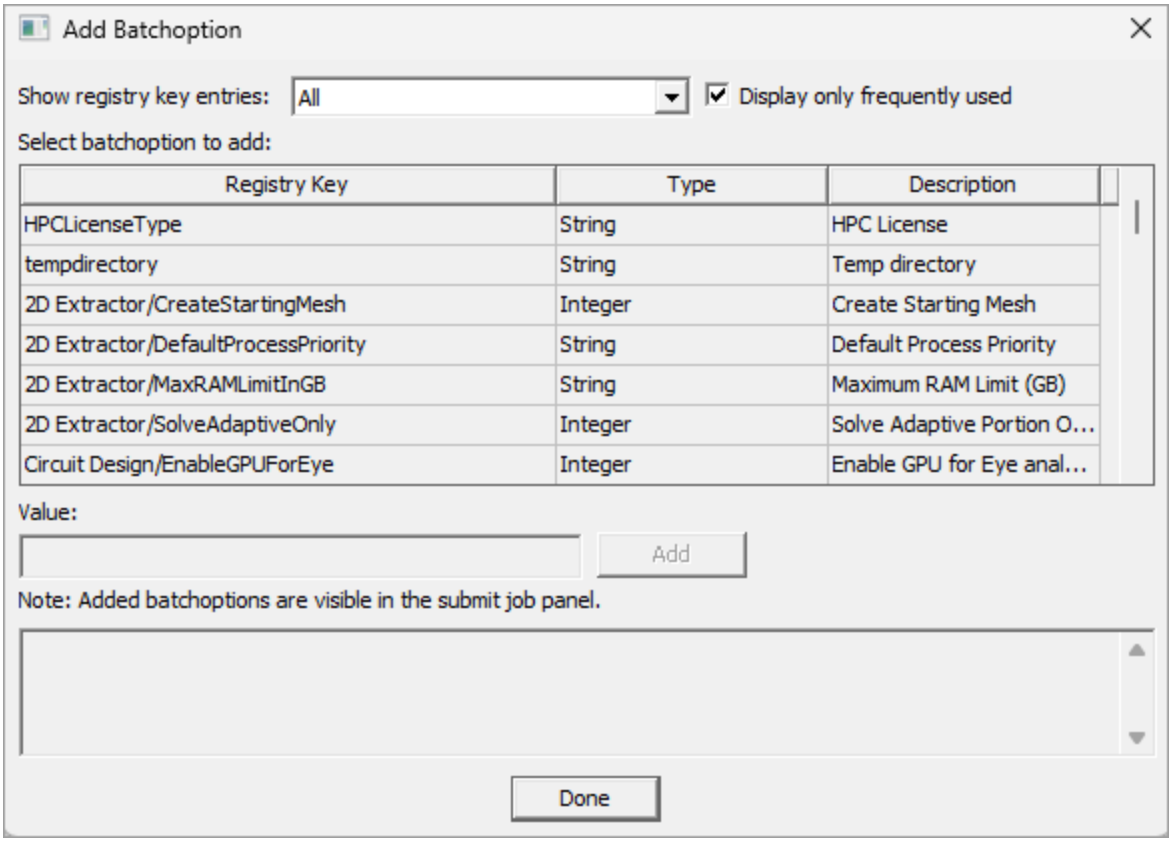
- **Monitor job** – you must enable this option to monitor the job from the user interface. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** function or by checking the dialog box that opens when you submit the job.
- **Wait for license** – specify whether to wait until a license if available before starting a simulation.
- **Generate Archive** – Generates an archive for the submitted job once it is complete. The project file, batchinfo directory, any external files, and the results directory are combined into the archive. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc.*.. will be written instead. This function will not overwrite an existing archive.

  Note that if an archive file is selected, this option will be renamed **Repackage results into archive**.

- **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.

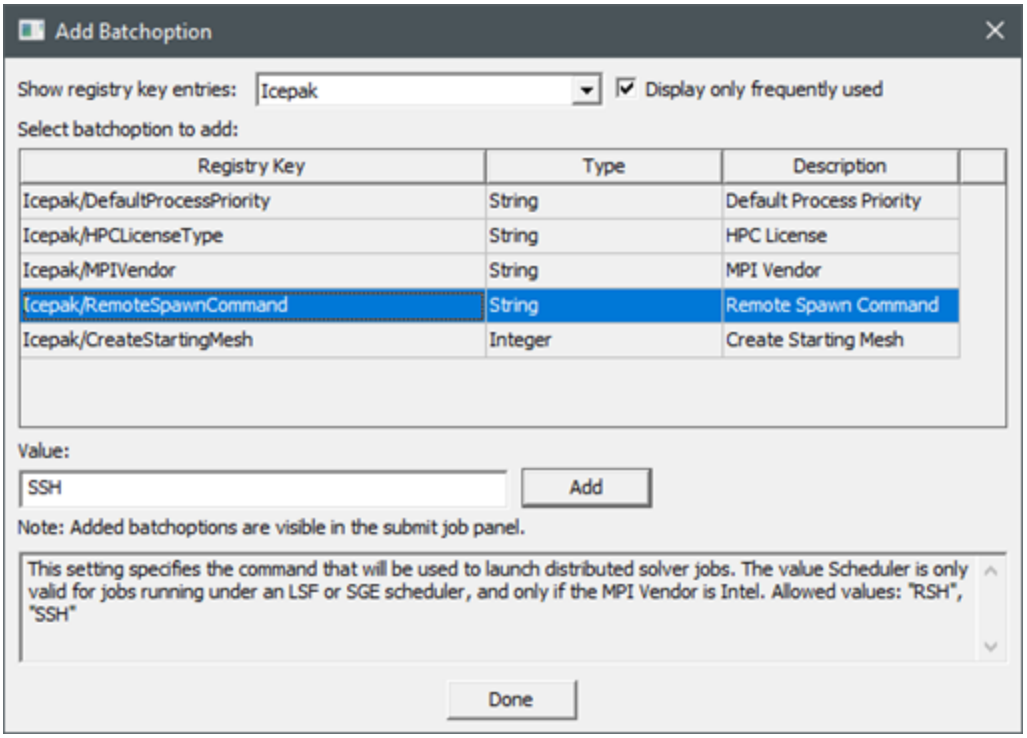
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



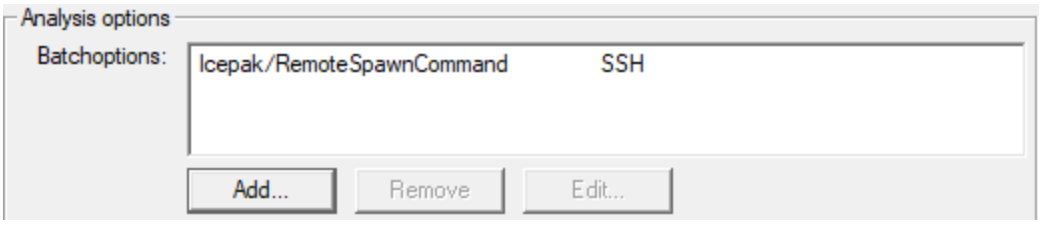
This dialog box provides access to all -batchoption commands. The drop-down menu lets you select specific categories, and you can choose to display only frequently used commands. You can edit and remove any batch options you specify.

Select a Registry Key in order to show the current Value for the type. The lower field explains the meaning of the Type Value.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

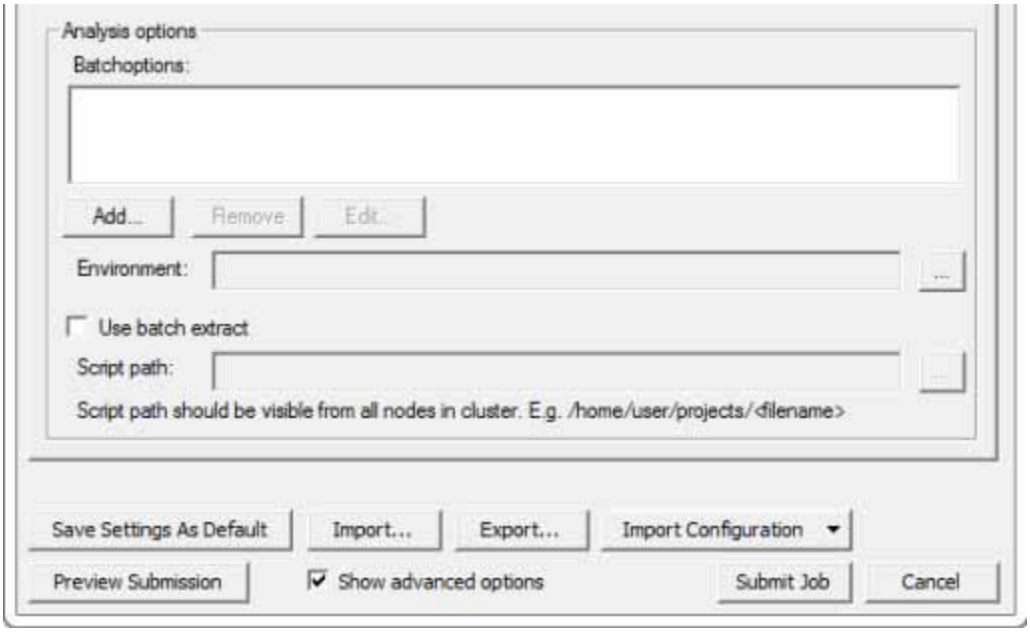


Any batchoptions for which you select **Add** will be visible in the **Submit Job** dialog box.

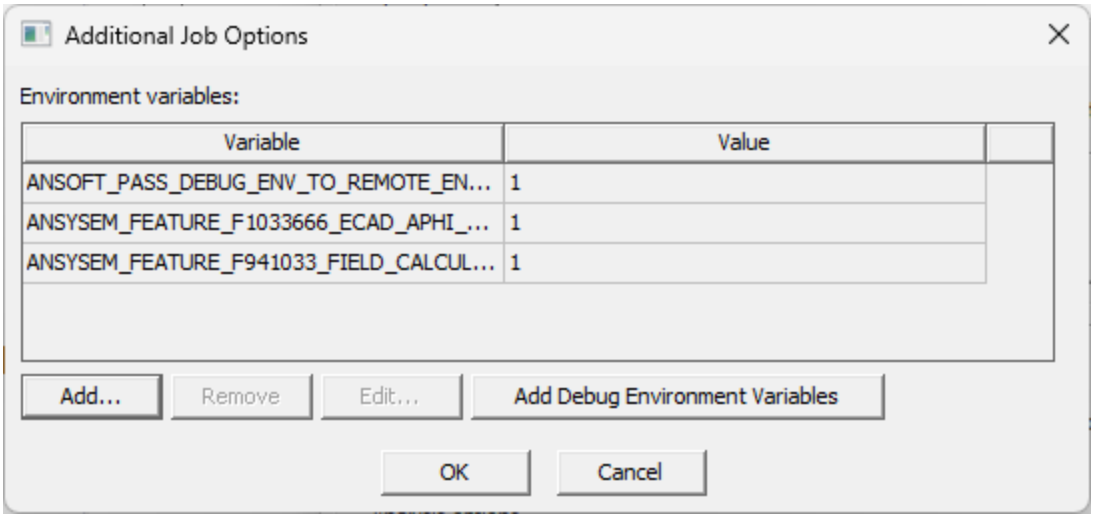


If you have the **Show advanced options** box checked in the **Submit Job** dialog box, the Environment field and the Use batch extract fields display.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

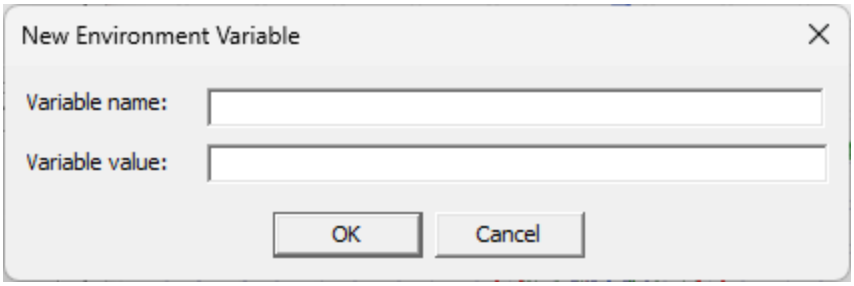


The **Environment** field lets you specify any environment variables. Click the ellipsis button [...] to display the **Additional Job Options** dialog box.



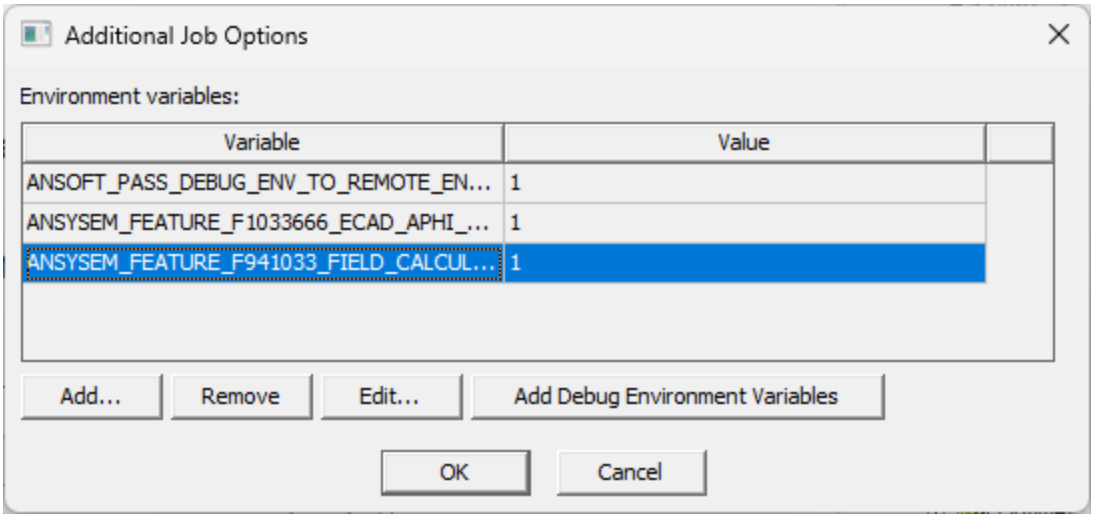
Click the **Add...** button to open the **New Environment Variable** dialog box.

Ansys Electromagnetics Suite 2026 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.

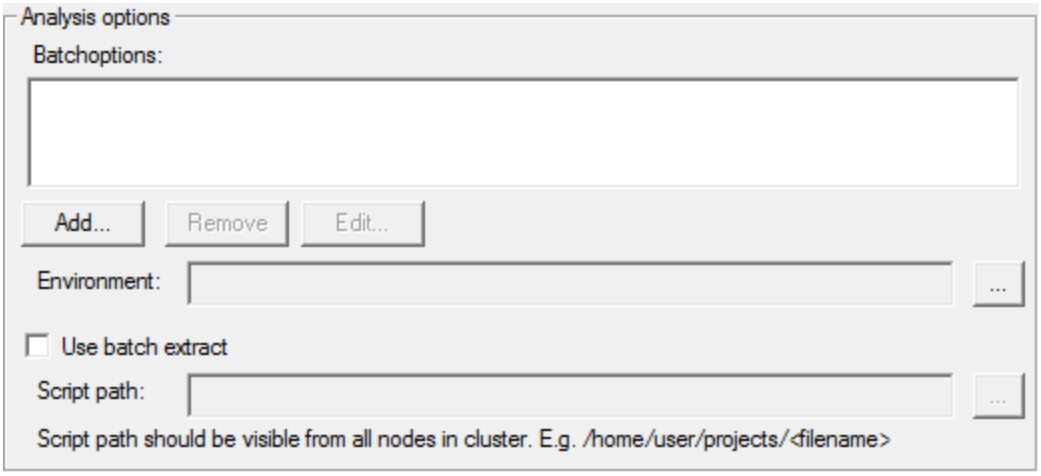
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 (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.

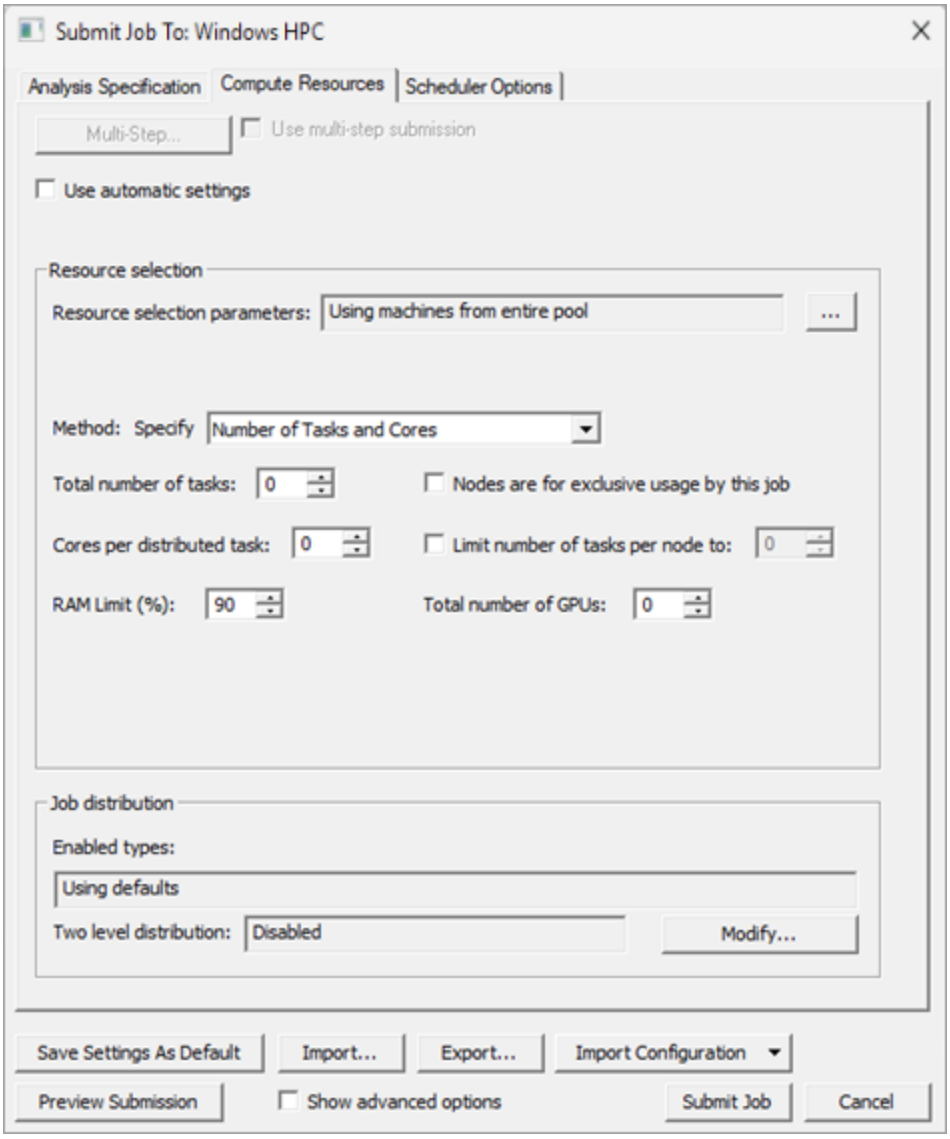


See Running Ansys Electronics Desktop from a Command line for an explanation of the solve information available through batch extract.

The **Preview Submission** button opens a window that shows the text commands that will be sent to the scheduler.

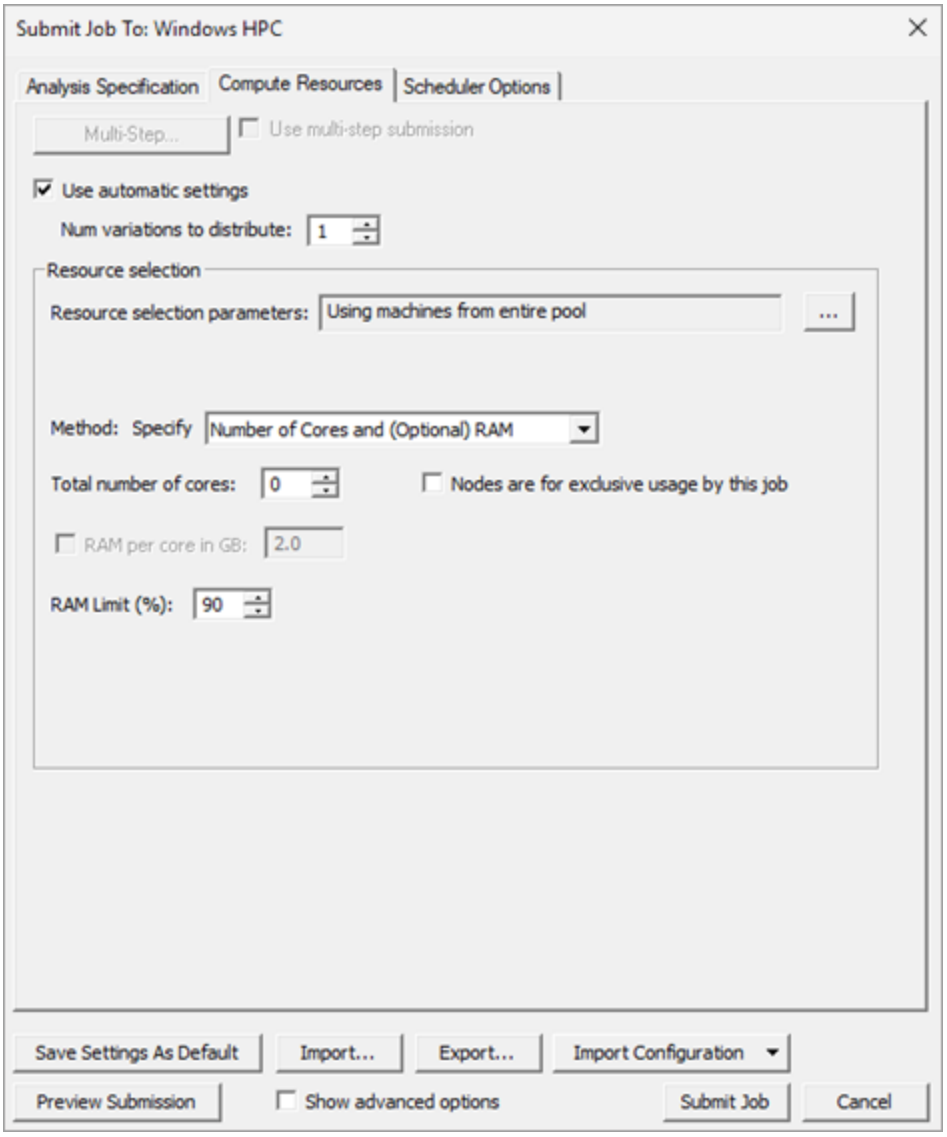
The following figure shows the **Compute Resources** tab of the **Submit Job To** dialog box.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

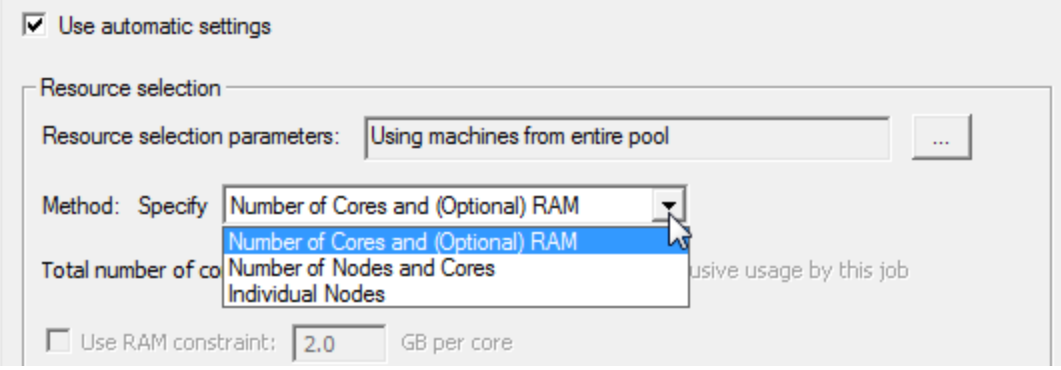


With **Use automatic settings** selected, the **Job distribution** field is removed. When using automatic settings, you can specify **Resource selection** parameters. The ellipsis button [...] opens the Compute Resource Selection Parameters dialog box. If you do not specify any parameters, the default 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**.

> **Note:**
>
> Automatic distribution settings are not available for Icepak designs.

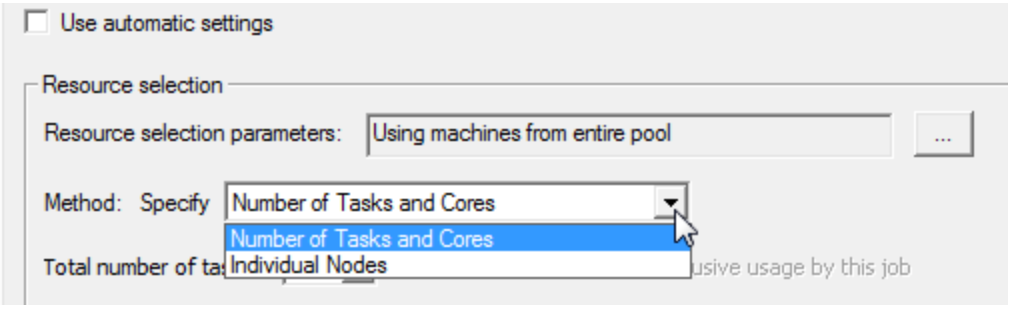
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

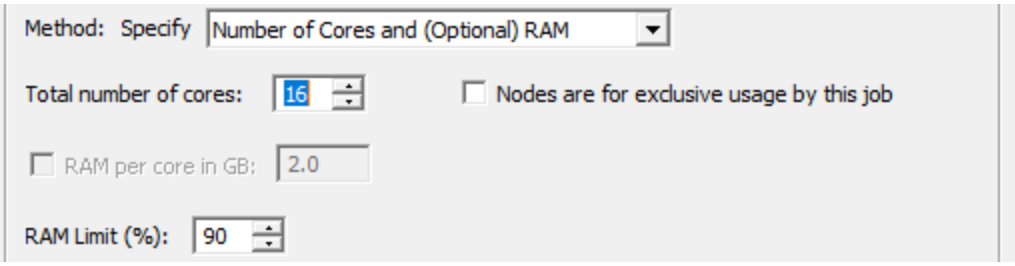
If you select **Use automatic settings**, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains.

If you uncheck or cannot access **Use automatic settings**, these two Methods are listed:



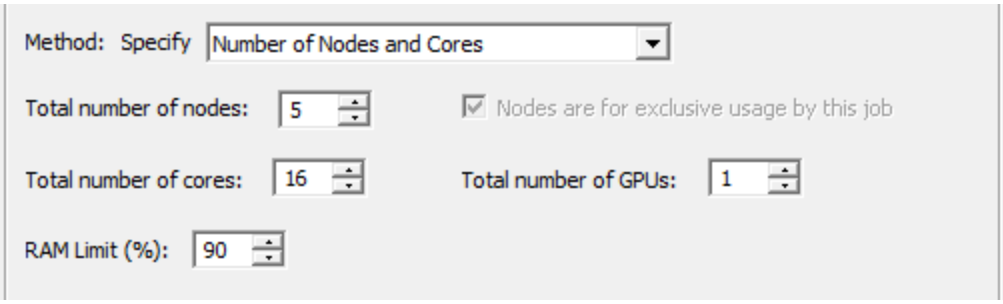
For Windows, there is a 64-core limit for a single machine. Each Method selection changes the available options listed:

- Specify Number of Cores and (Optional) RAM

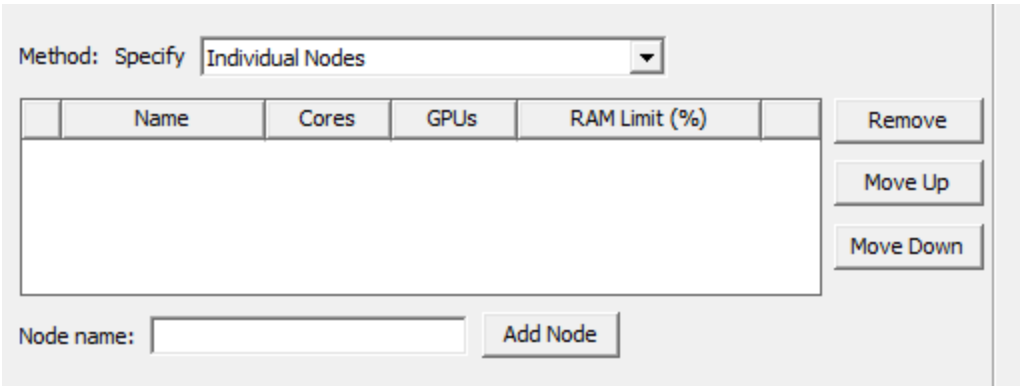


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Nodes and Cores

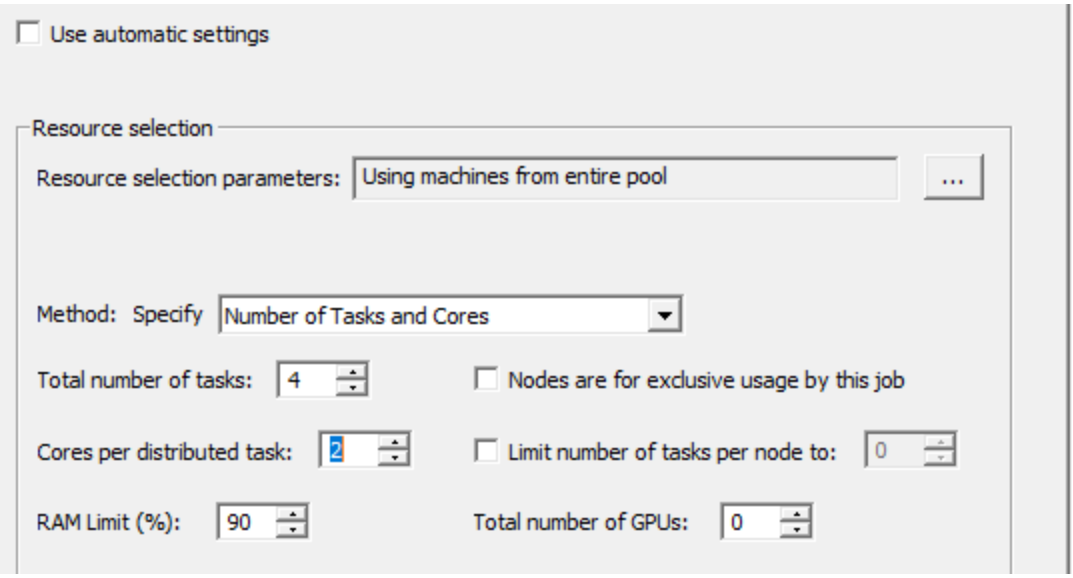


- Individual Nodes



- 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):

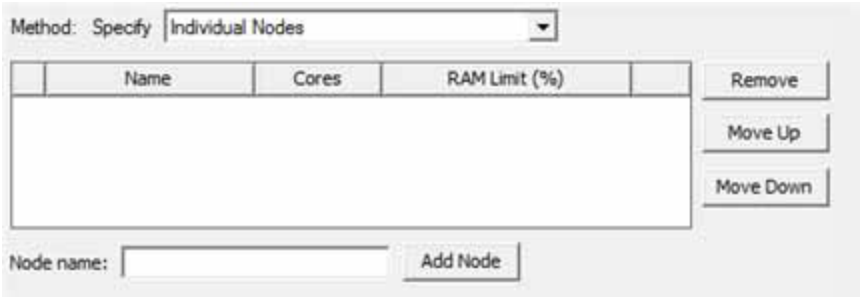
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual Node List

For Windows HPC jobs, you may either specify a node list, or specify job parallelization parameters, but not both.

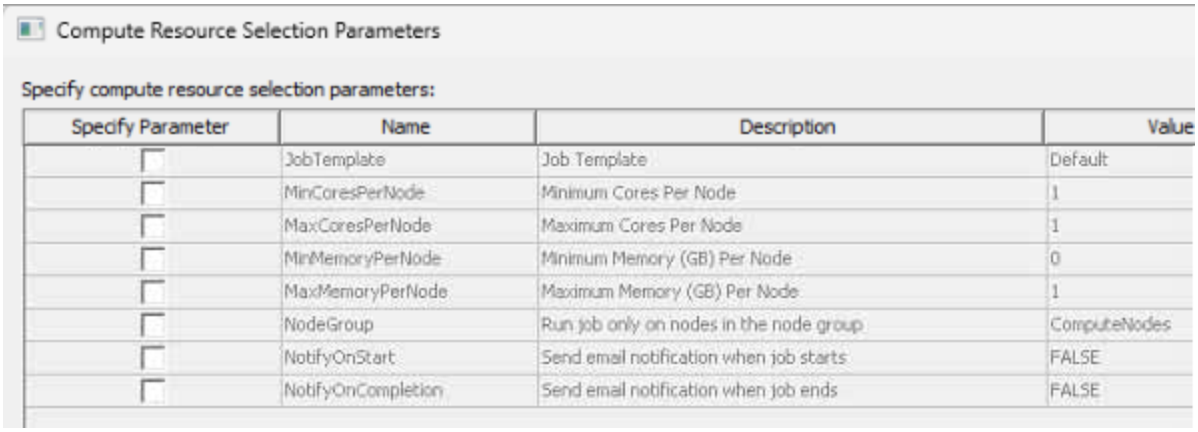
If you select the Individual Nodes Method, you may specify a node list, and the Job parallelization controls are disabled. In this case, the node list should only include cluster nodes that are valid for the job. For each node, you enter the node name and add the node. In the table, you can specify the 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 Settings

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.


Compute Resource Selection Parameters

Specify compute resource selection parameters:

Specify Parameter	Name	Description	Value
☐	JobTemplate	Job Template	Default
☐	MinCoresPerNode	Minimum Cores Per Node	1
☐	MaxCoresPerNode	Maximum Cores Per Node	1
☐	MinMemoryPerNode	Minimum Memory (GB) Per Node	0
☐	MaxMemoryPerNode	Maximum Memory (GB) Per Node	1
☐	NodeGroup	Run job only on nodes in the node group	ComputeNodes
☐	NotifyOnStart	Send email notification when job starts	FALSE
☐	NotifyOnCompletion	Send email notification when job ends	FALSE

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:

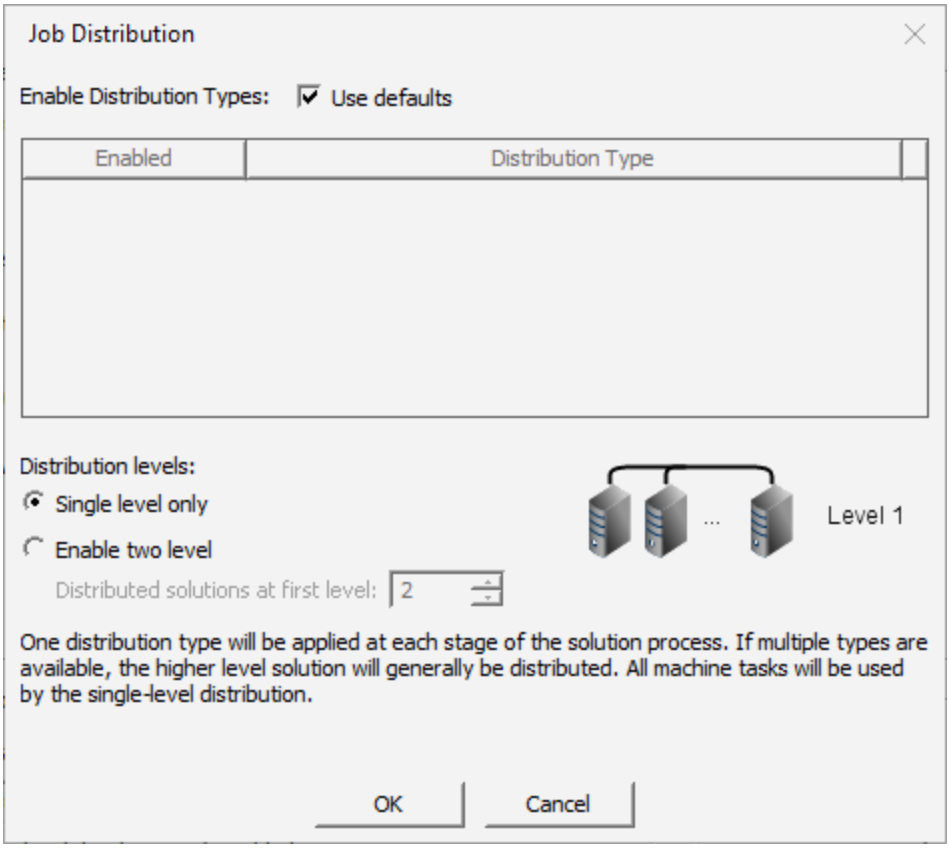
- Total number of tasks: The number of nodes requested for the job is the total number of tasks divided by limit on the number of tasks per node, rounded up if it is not an integer.
- Cores per distributed task. This determines the amount of multiprocessing per task.
- Whether nodes are for exclusive 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".

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For more information about job parallelization in Icepak, see Specifying Tasks and Cores for Job Schedulers.

Job Distribution

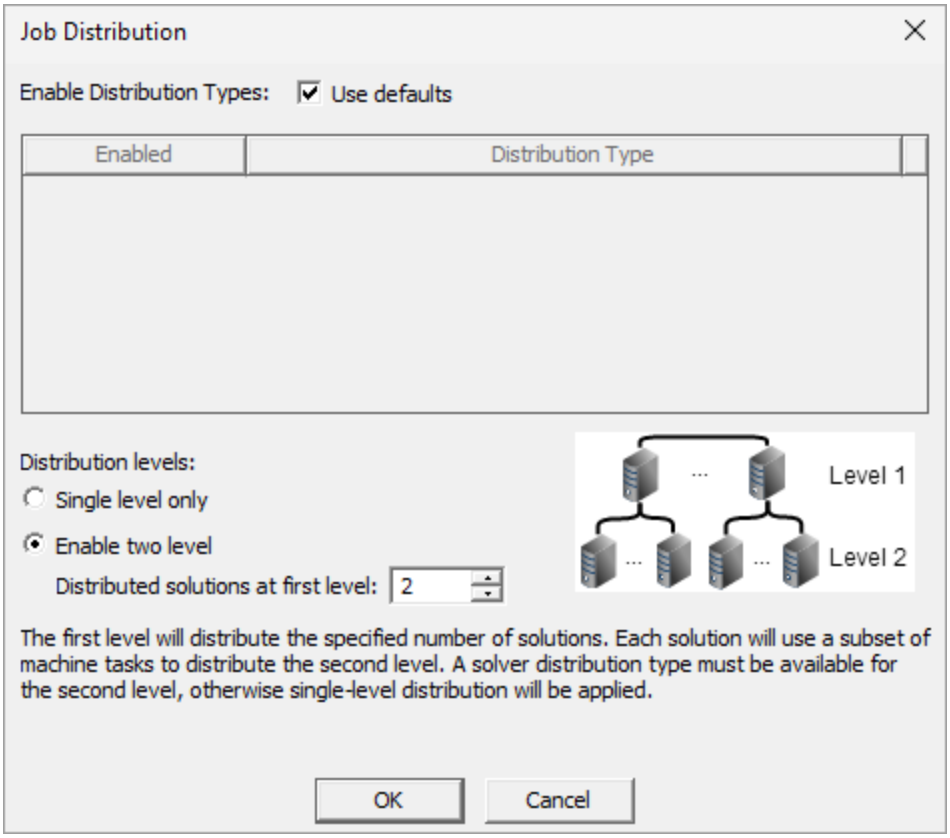
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the **Job Distribution** dialog box and select the **Enable two level** option if applicable and desired.



- Second level distribution operates within DSO. If available and enabled you can specify

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the number distributed solutions for level 1.

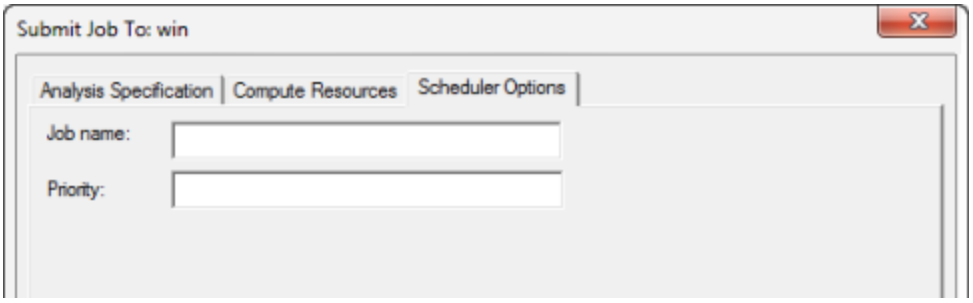


In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, 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 2026 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, **Analysis Specification** 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 2026 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 2026 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

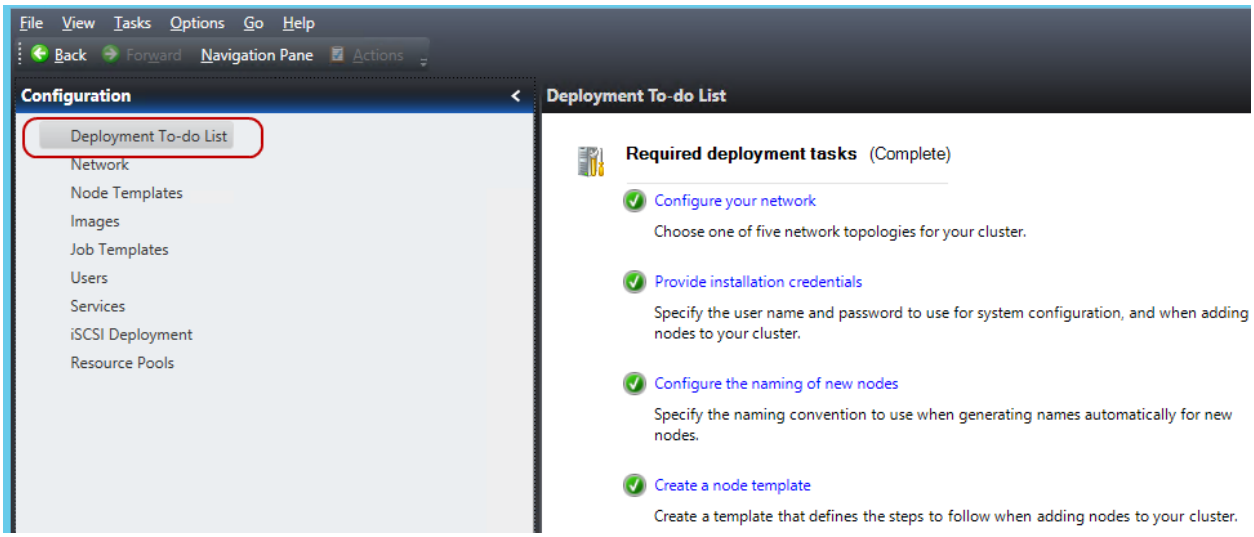
If the AffinityType is set to “No Jobs,” then we can allow non-exclusive jobs in many cases, but non-exclusive jobs will be disallowed if you specify an explicit host list (because all resources on all specified hosts are used), if you specify nodes and cores (to ensure that the job has exclusive access 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

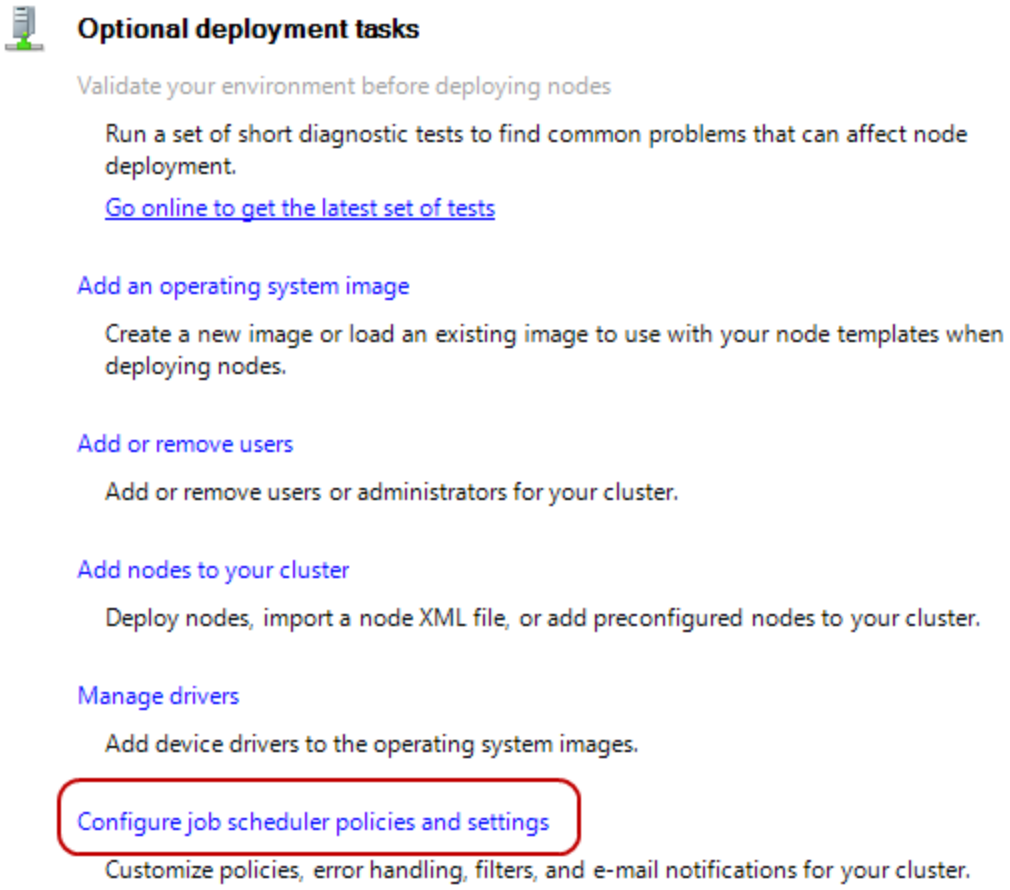
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



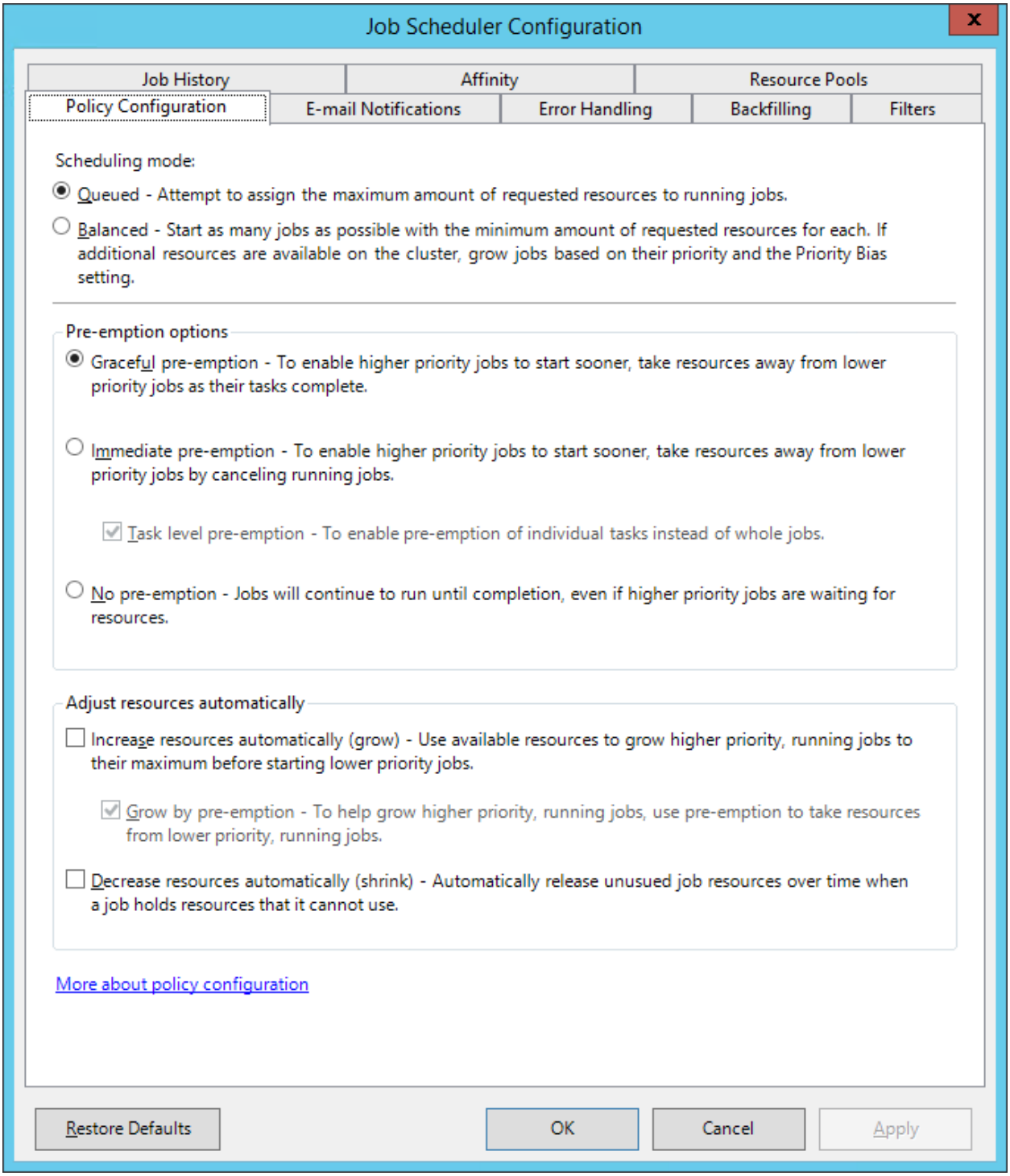
Next, the "Configure job scheduler policies and settings" should be selected. This is under the heading "Optional deployment tasks."

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



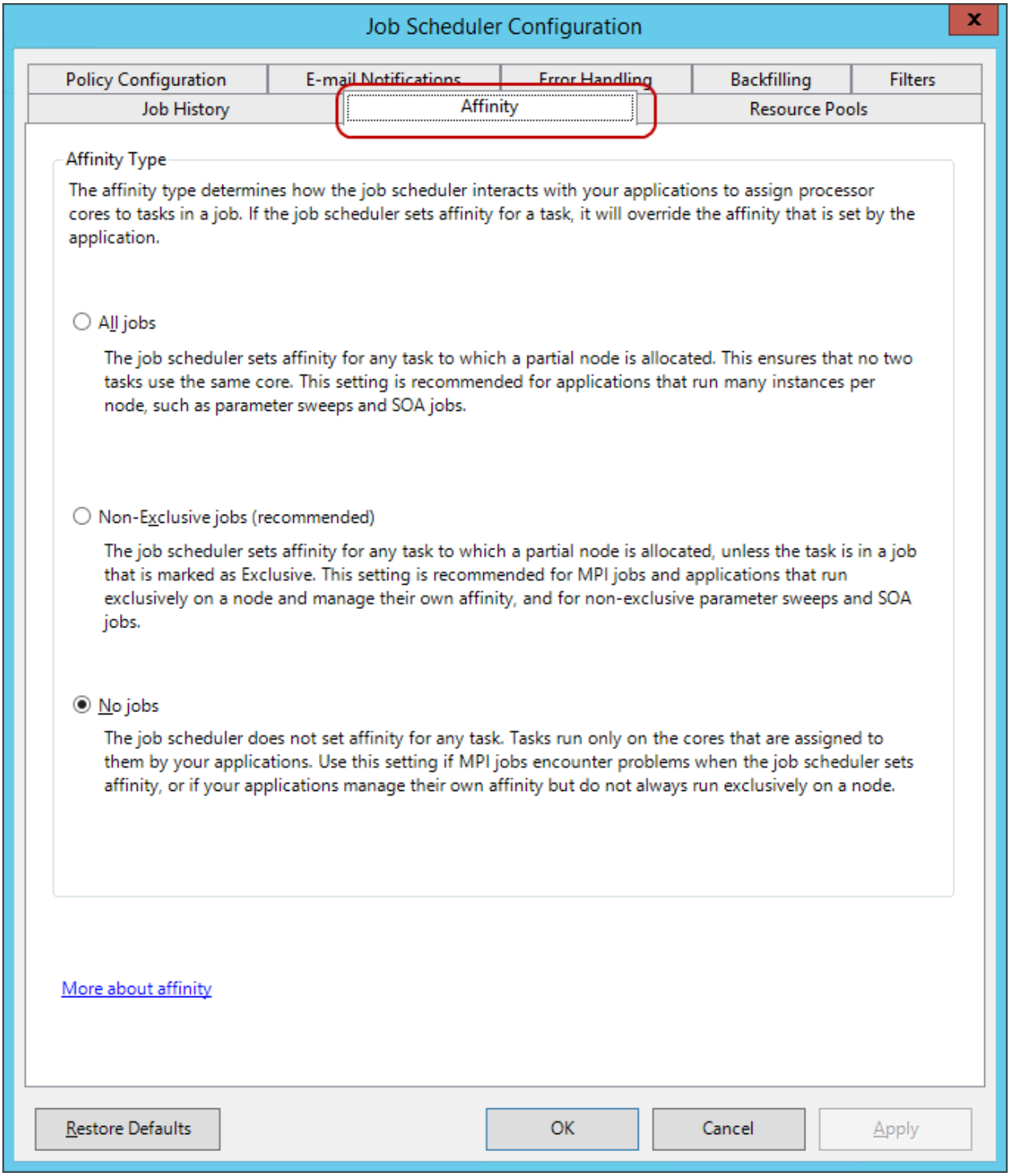
The *Job Scheduler Configuration* dialog box pops up, as shown below.

Ansys Electromagnetics Suite 2026 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 2026 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 2026 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

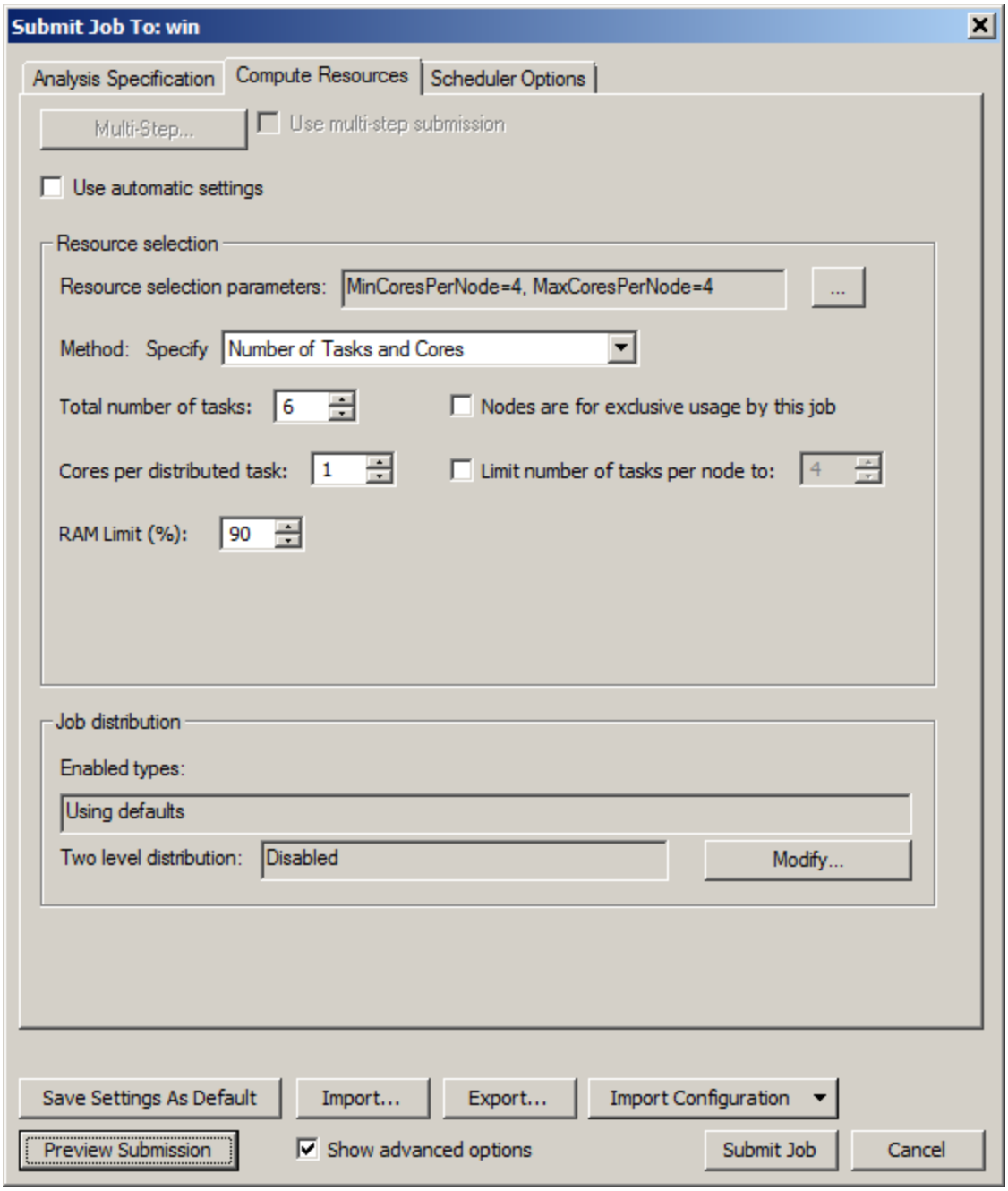
Most of the following screen shots are for the same project file submitted to a Windows HPC cluster with the AffinityType set to "No Jobs". The scheduler resource selection parameters are the same for all of these screen shots:

- 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.

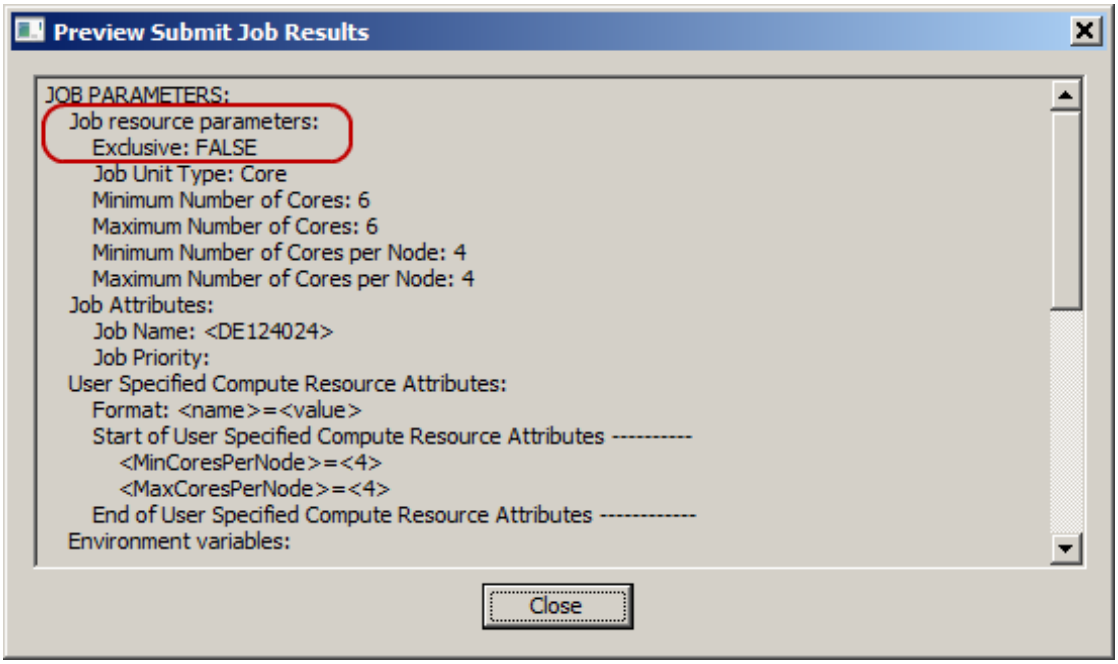
The following screen shot shows this configuration with a Submit Job with Task and Cores specified and nodes as non-exclusive.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

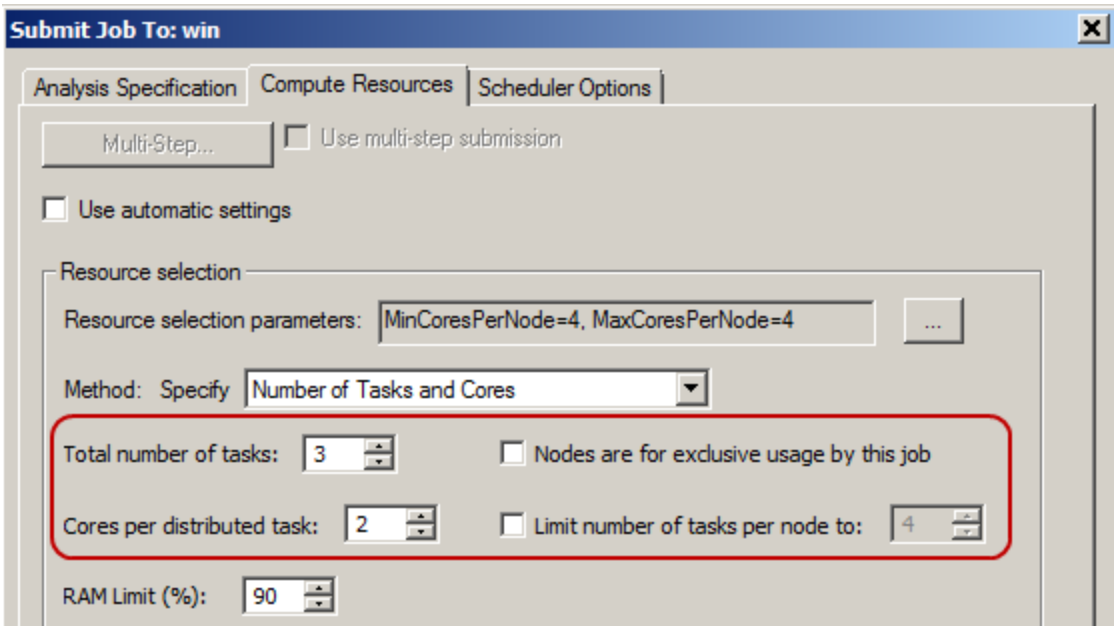


Jobs may only be submitted using this resource selection method with the exclusive parameter set to false if all three of the following conditions are met:

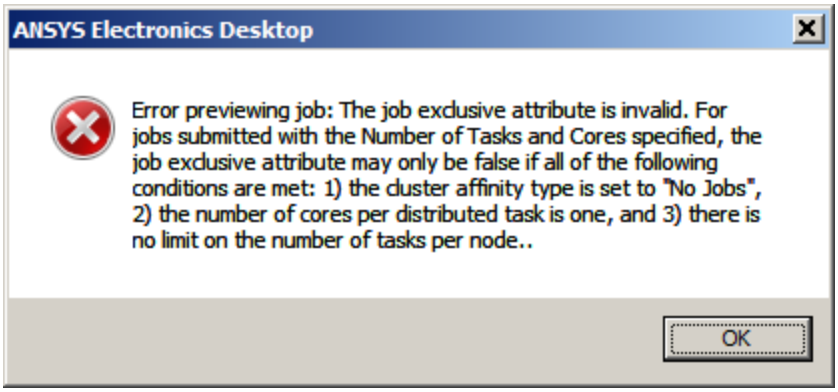
1. The cluster AffinityType is "No Jobs",
2. There is no multiprocessing, and
3. There is no limit on the number of tasks per node.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



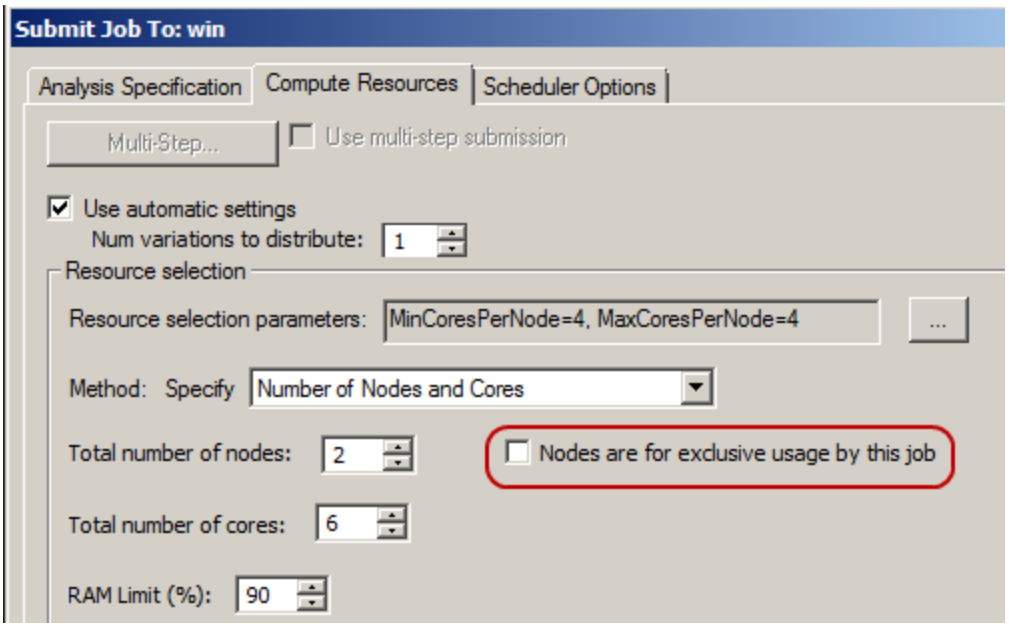
The following figure and screen shot ErrorMsgTasksAndCoresNonExMP.png shows the error message box shown to the user.



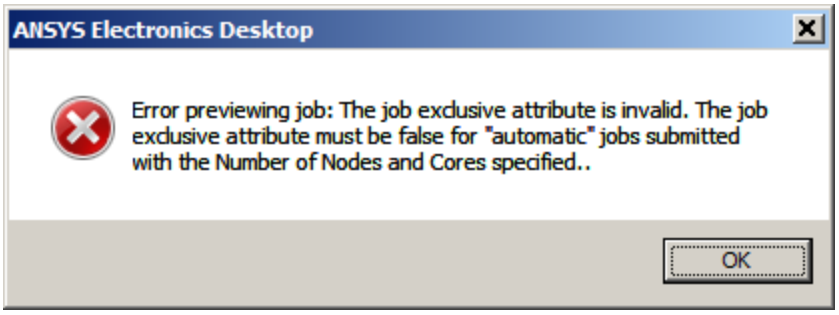
The next example shows submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Nodes and Cores". If you select *Use automatic settings* with *Num variations to distribute* set to 1, Optimetrics variations 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 2026 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:

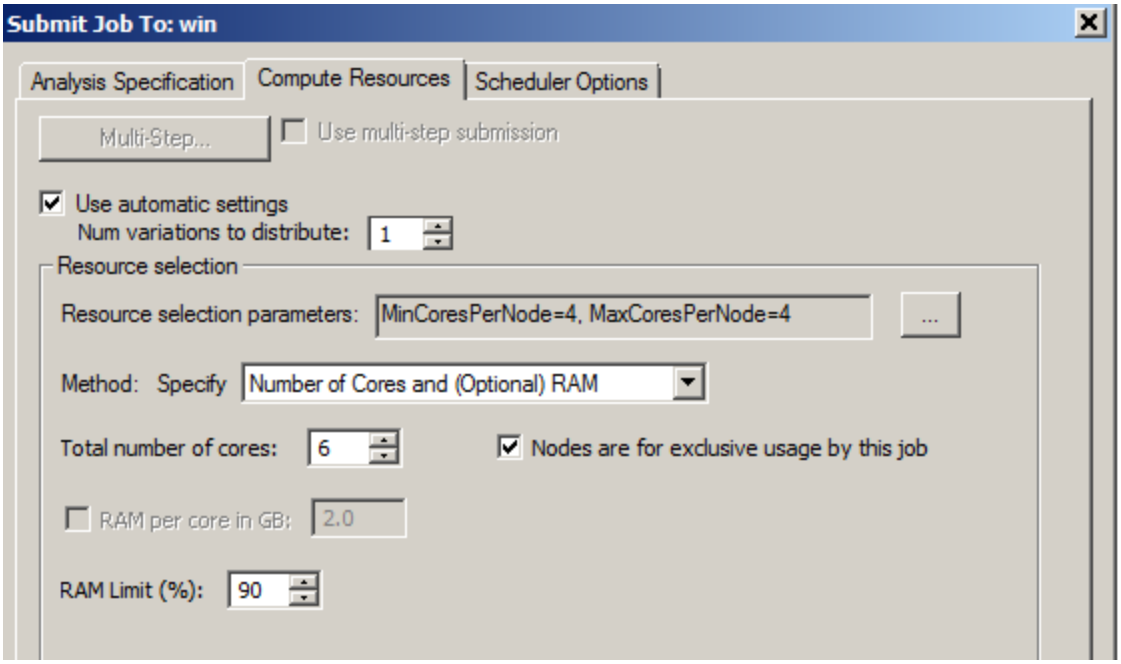


The following figure shows the error message.

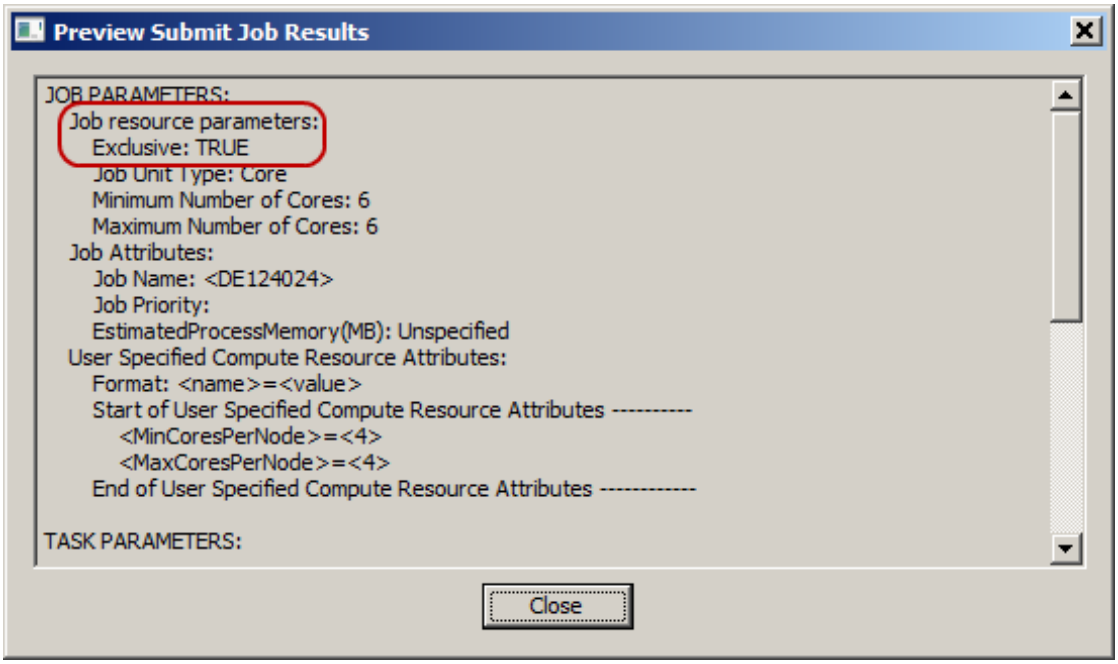


The next example show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Cores and (Optional) RAM". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

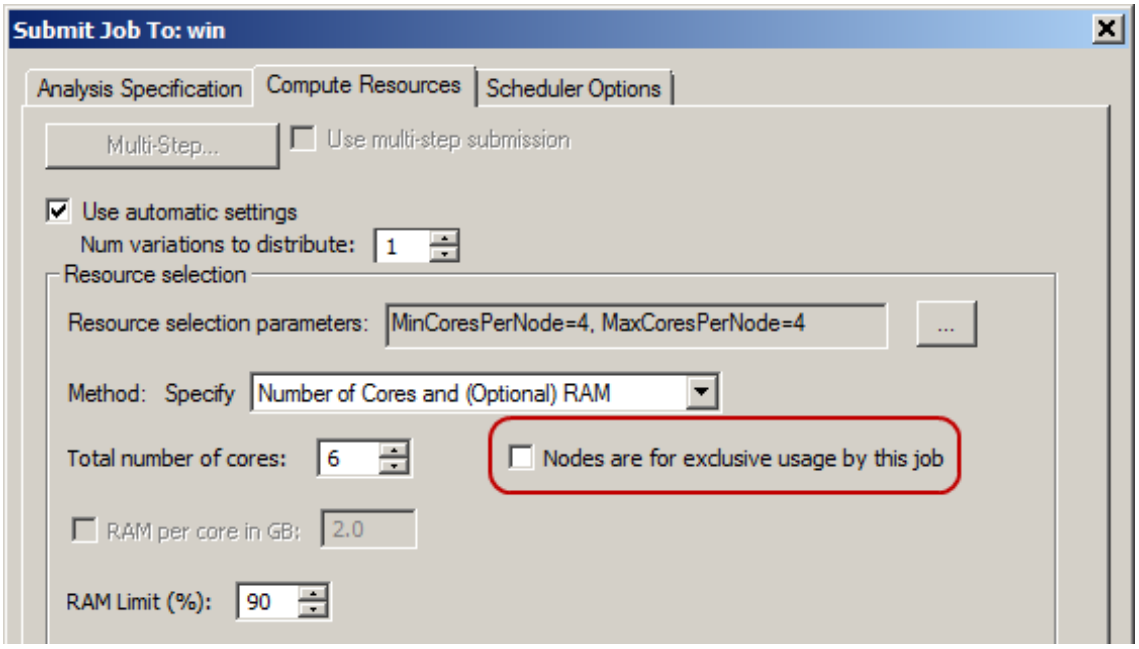


The following figure shows the Job Preview for this case, indicating that the exclusive parameter for the job is true.

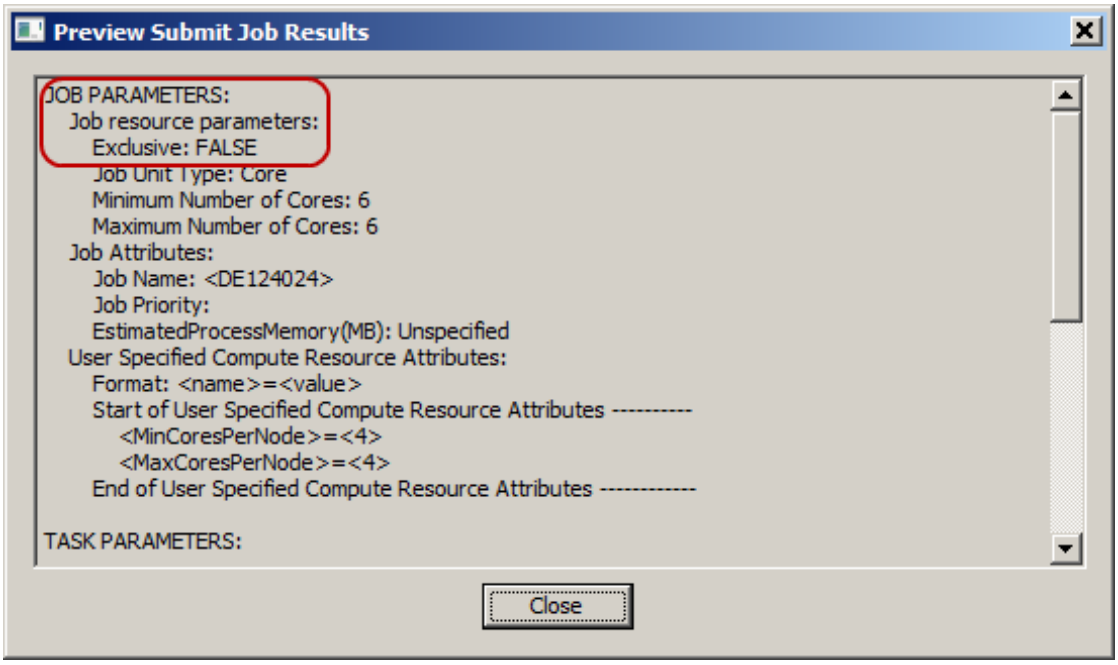


If the exclusive check box is unchecked, then the job is submitted with the exclusive parameter set to false. The following figure shows this configuration.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



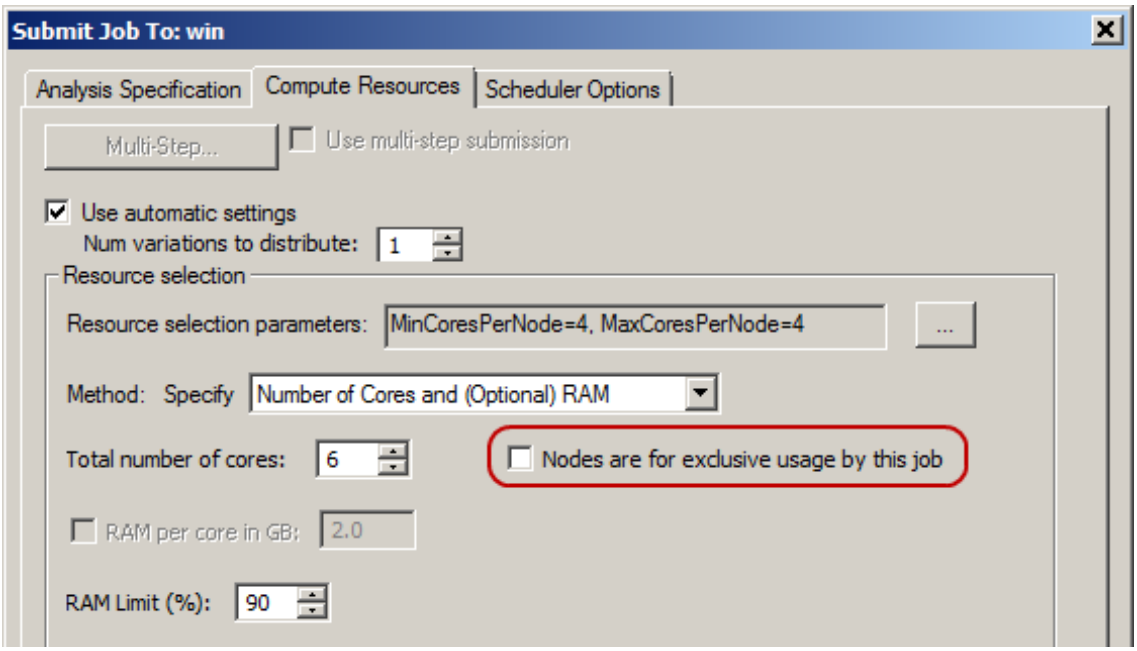
The following figurescreen shot PreviewCoresAndRAMNonEx.png shows the Job Preview for this case, indicating that the exclusive parameter for the job is false.



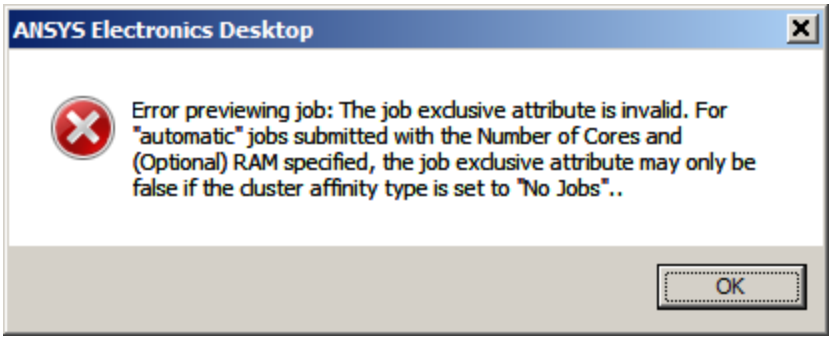
The last example resembles the previous one, except that the Windows HPC cluster AffinityType is set to "Non Exclusive Jobs" in this case. The job is submitted with the "Use

Ansys Electromagnetics Suite 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Desktop GUI. Ensure that scheduler commands are available in the path before you launch Electronics Desktop.

> **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 2026 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the LSF scheduler proxy library, the ConfigString entry in the ansoftrsmservice.cfg configuration file is ignored. It may be empty or omitted entirely.

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 2026 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 2026 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\v261\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\v261\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\v261\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\v261\AnsysEM\ansysedt" -ng -
BatchSolve -Distributed ~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2026 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\v261\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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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/v261/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2026 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/v261/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/v261/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/v261/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 2026 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/v261/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 2026 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 2026 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 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 2026 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 2026 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\v261\AnsysEM", then the scheduler proxy library must be installed in directory "C:\Program Files\ANSYS Inc\v261\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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 Multiprocessing in the Same Job" on the facing page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v261\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```


Ansys Electromagnetics Suite 2026 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\v261\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Analysis and Multiprocessing in the Same Job

Distributed-processing using 4 engines and multiprocessing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v261\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):

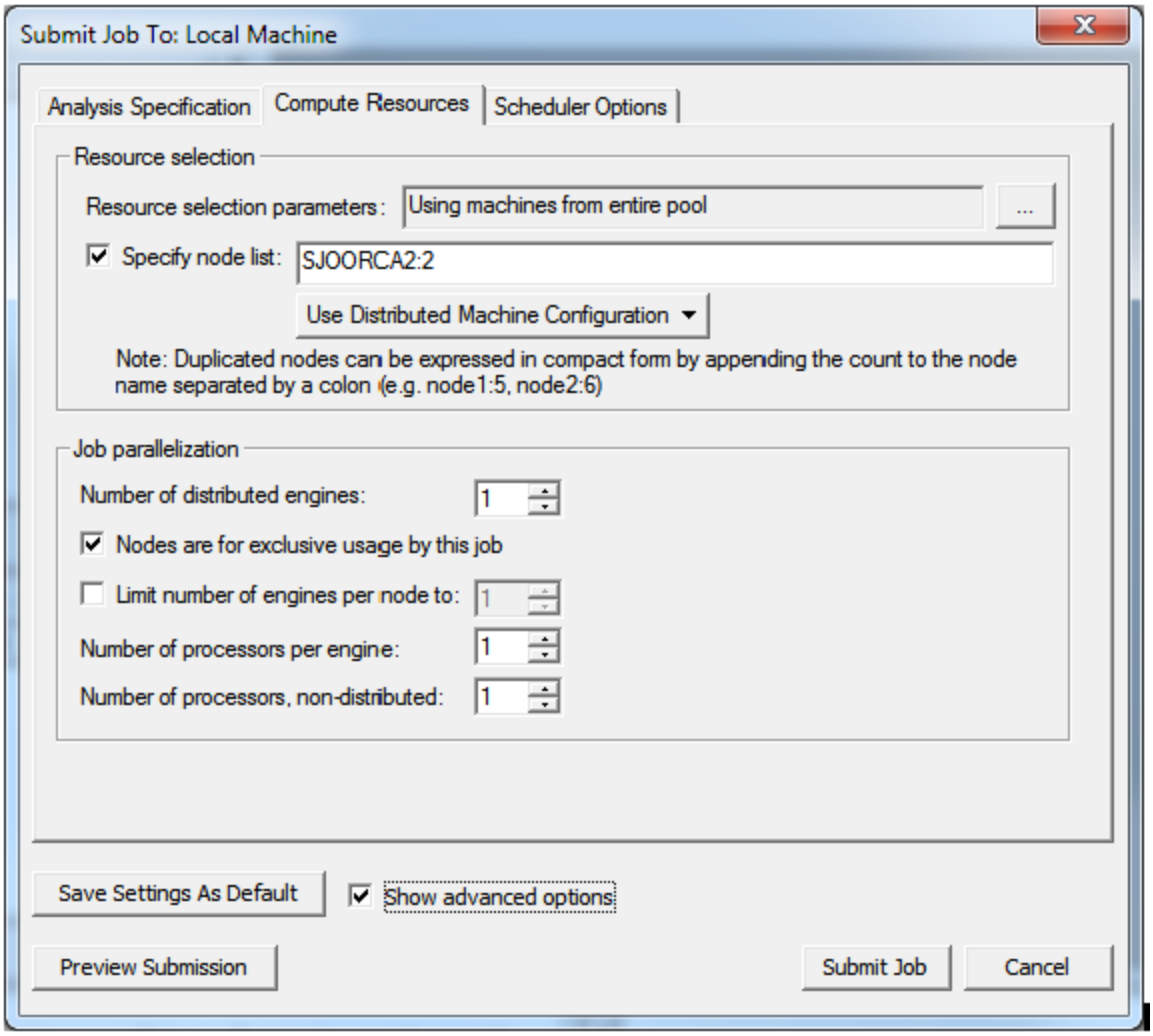
This function returns some basic information about the scheduler with which the scheduler proxy library interacts.

- 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 2026 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 2026 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 2026 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.

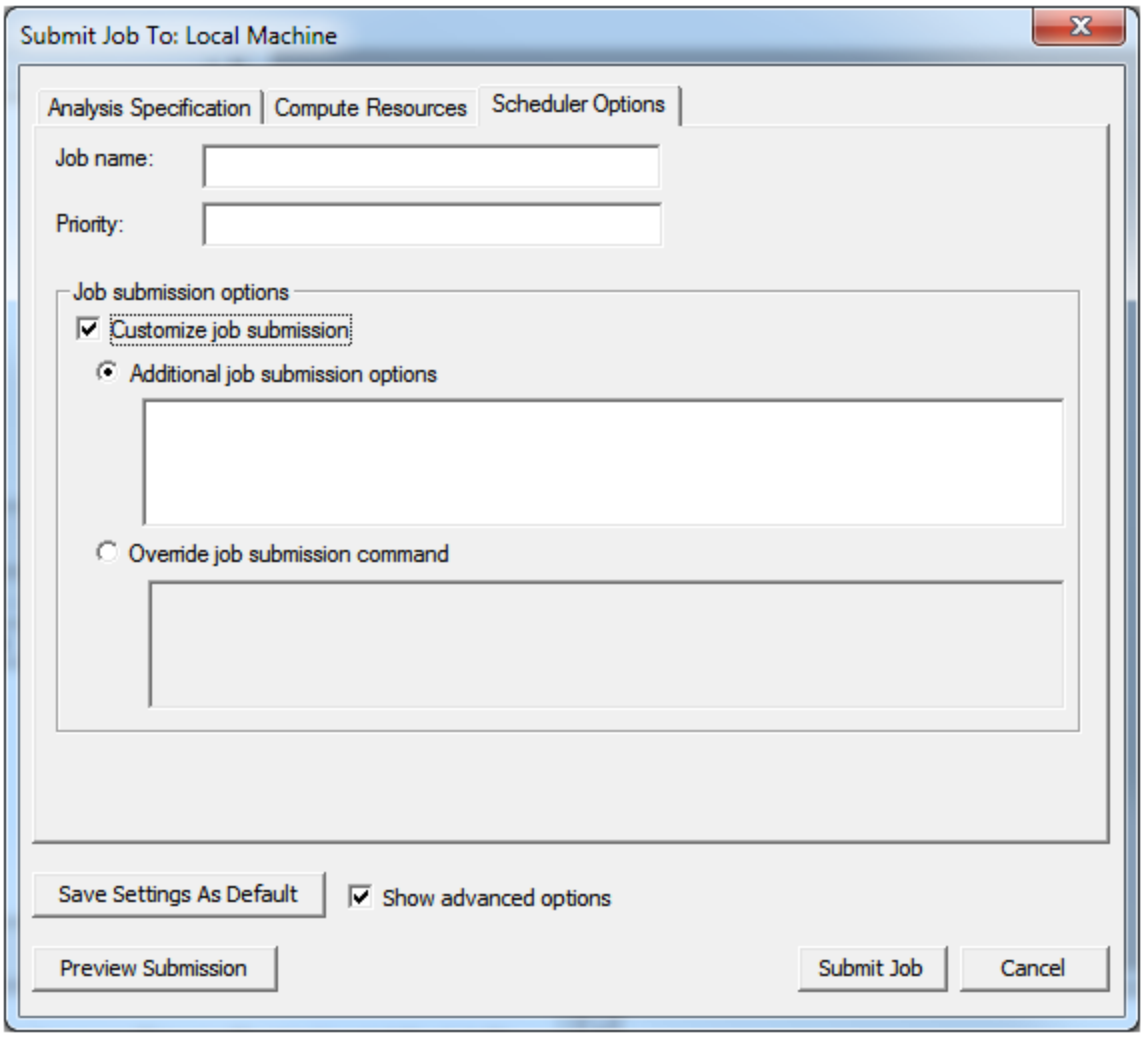
The **computeResources** argument is a reference to an object of type **UniformComputeResourcesStruct**. This **struct** contains zero or more resource attribute settings for the job. Each resource attribute setting consists or a resource name and a resource value. The resource name is the name of one 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 2026 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 2026 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 2026 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 2026 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 2026 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 five 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 six engines would be started, but at most five 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 2026 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 2026 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 2026 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 2026 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\v261\AnsysEM", then the IronPython program must be installed in directory C:\Program Files\ANSYS Inc\v261\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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

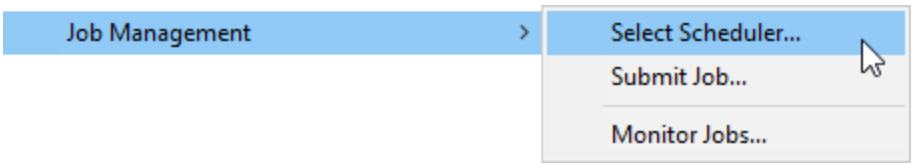
Note:

- 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.

Selecting a Scheduler

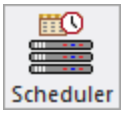
Access the **Select Scheduler** window one of three ways:

- Click **Tools** > **Job Management** > **Select Scheduler...**



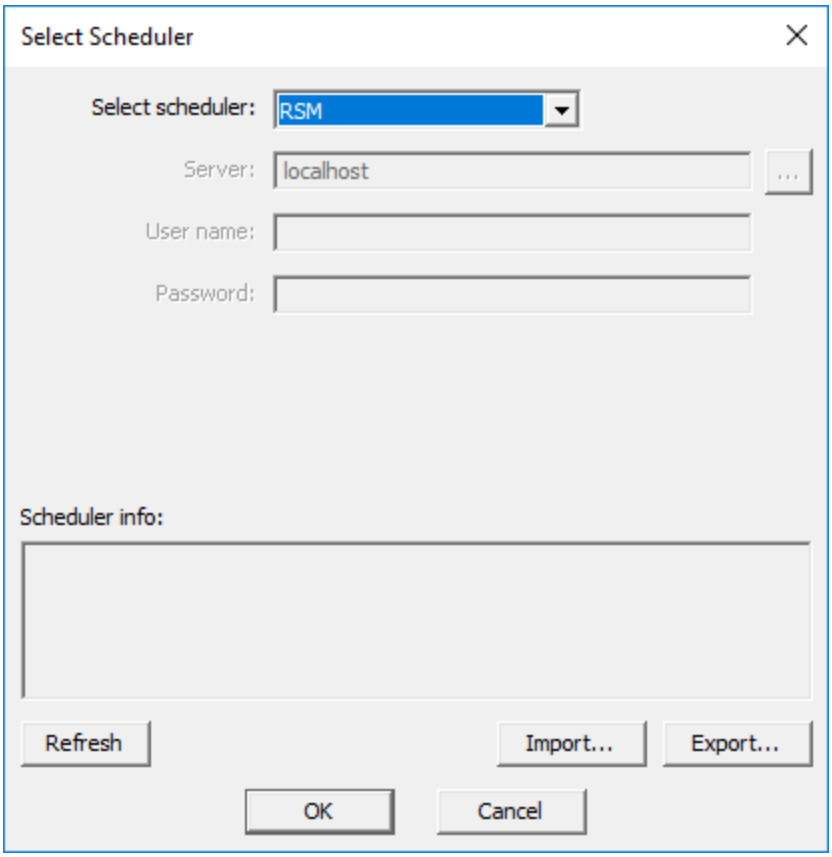
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 2026 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 Burst Computing 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 Burst Computing 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 that 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 postprocessing 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 2026 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.

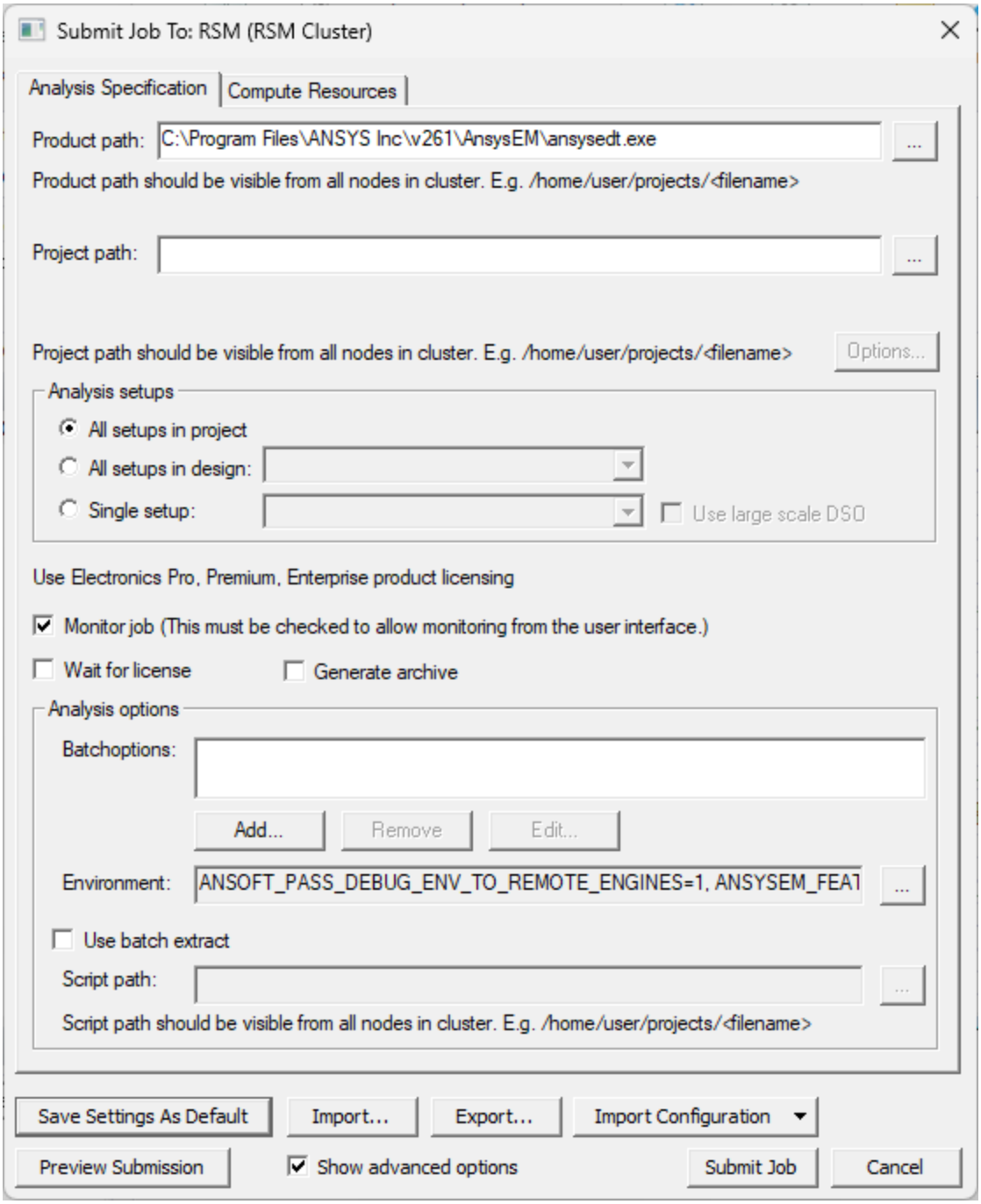
Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to 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 **Icepak** > **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Depending on which scheduler is selected, 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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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*.

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.

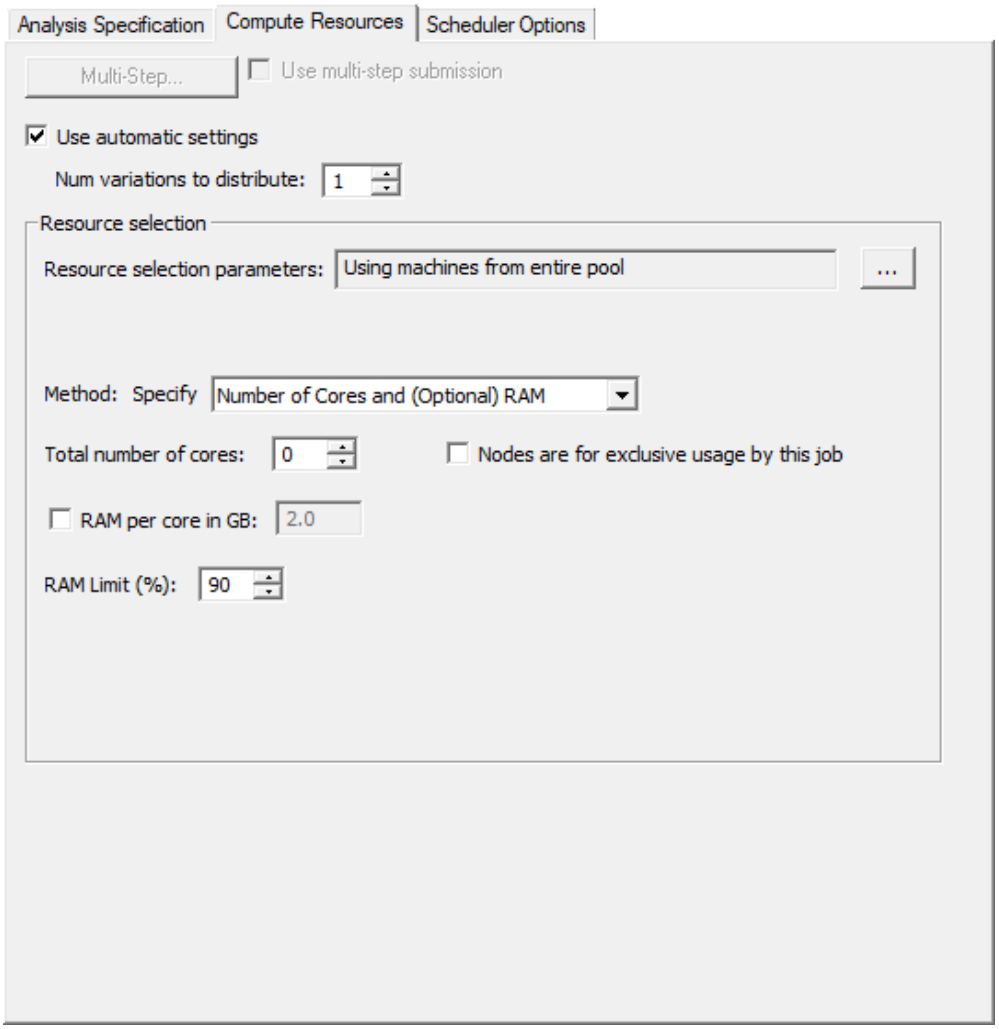
   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See Large Scale DSO for Parametric Analysis.

   > **Note:**
   >
   > Depending on which scheduler is selected (RSM, Windows HPC), the Submit Job window will include a **Generate archive** option, which is used to create an archive for the submitted job when it is complete. The project file, batchinfo directory, any external files, and the results directory are combined into the archive. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc.*.. will be written instead. This function will not overwrite an existing archive.
   >
   > Note that if an archive file is selected, this option will be renamed **Repackage results into archive**.

Ansys Electromagnetics Suite 2026 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.

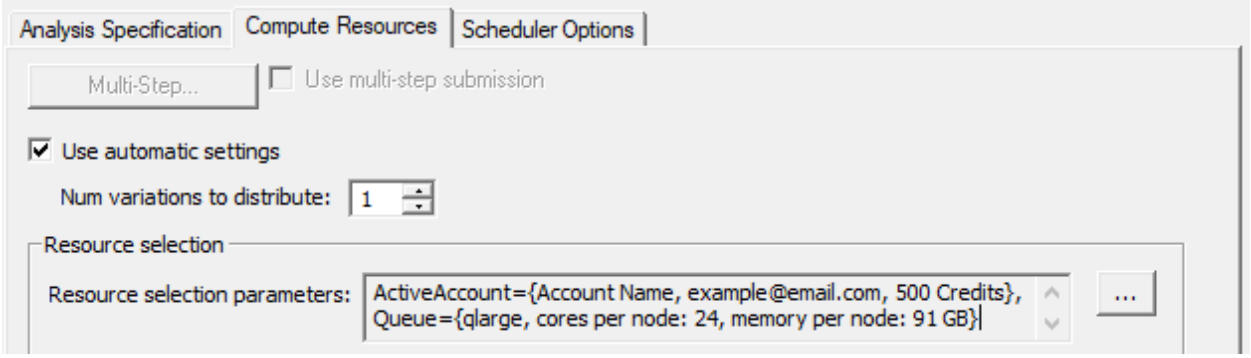
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **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 2026 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 Burst Computing 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:

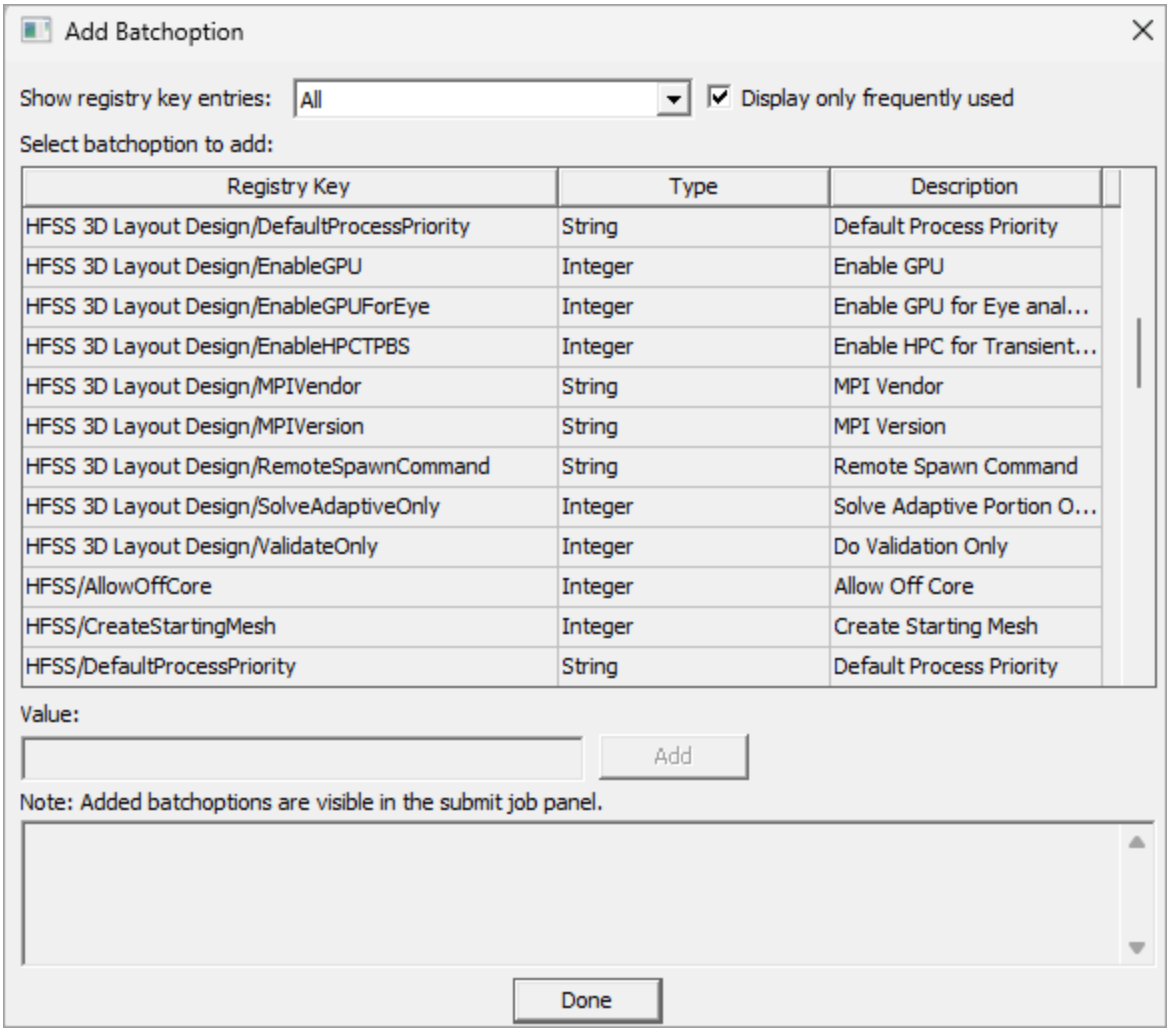
- Batchoptions
- Environment Variables

Ansys Electromagnetics Suite 2026 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.

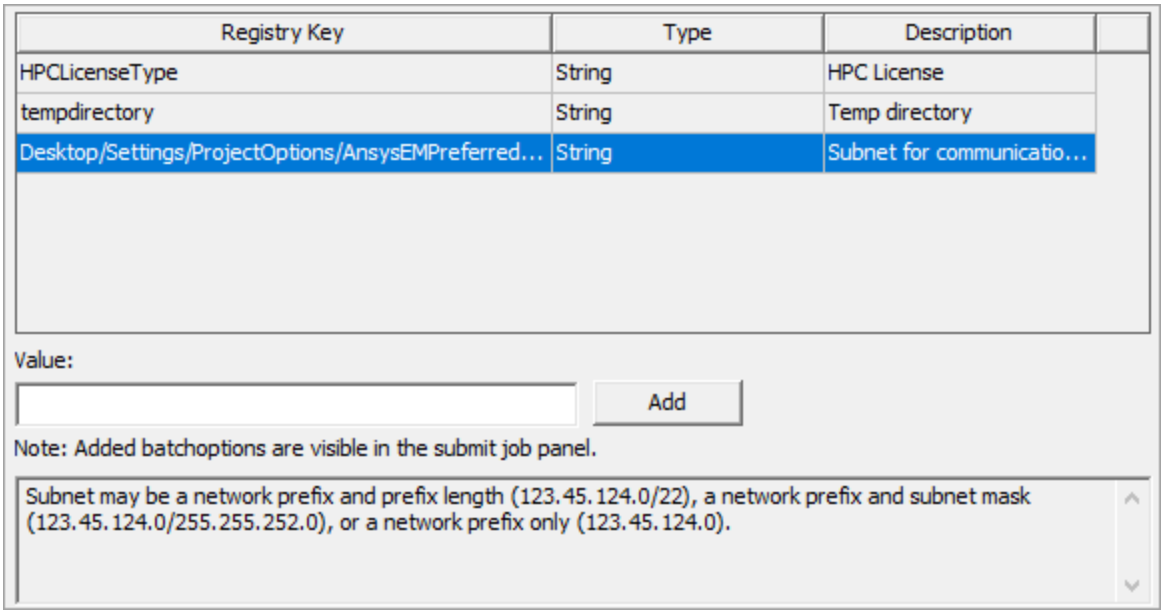


The **Show registry key entries** drop-down menu allows you to select categories of registry keys to display.

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:

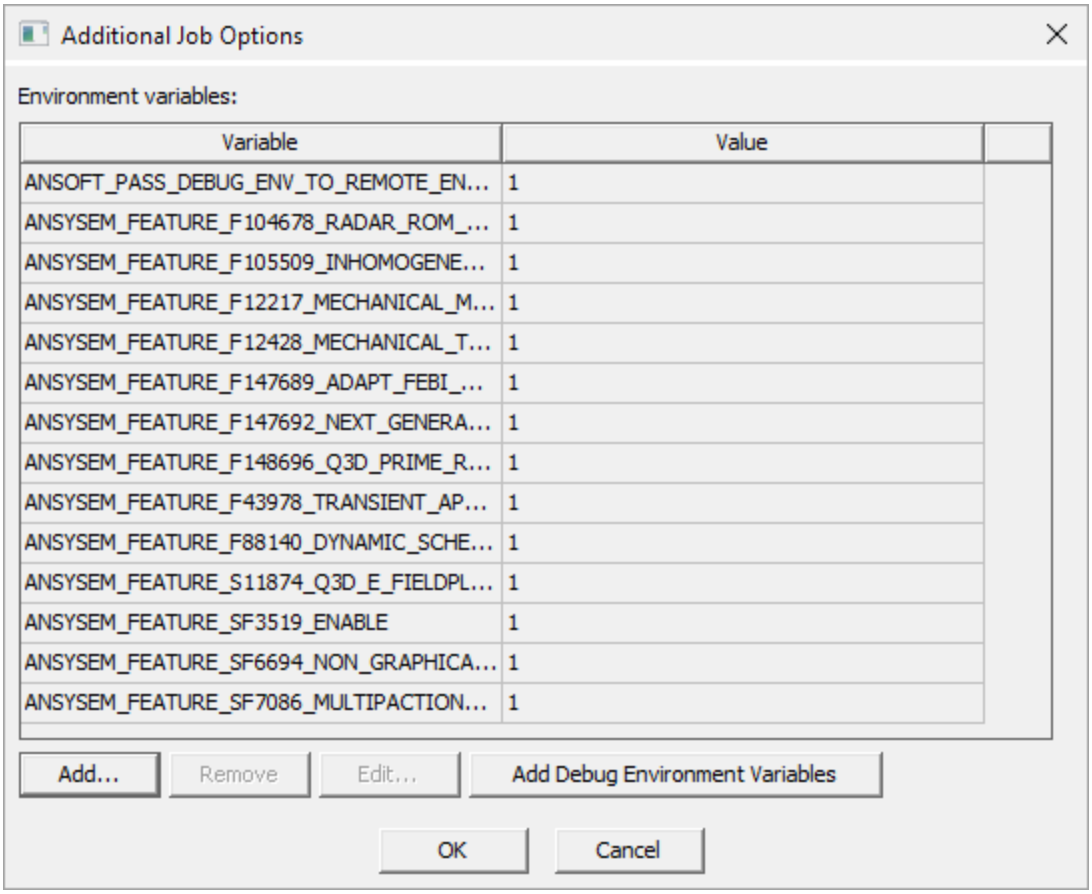
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variables

In the **Submit Job To** window, under **Environment**, click the ellipses (**...**) to open the **Additional Job Options** window.

Ansys Electromagnetics Suite 2026 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 2026 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 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 Multiprocessing 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\v261\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v261\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Analysis and Multiprocessing in the Same Job

Distributed-processing using 4 engines and multiprocessing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v261\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Job Import and Export

The scheduler selection and the job submission window have buttons for **Import...**, **Export...**, and **Import Configuration** at the bottom of the window. 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multi-Step Job Submission

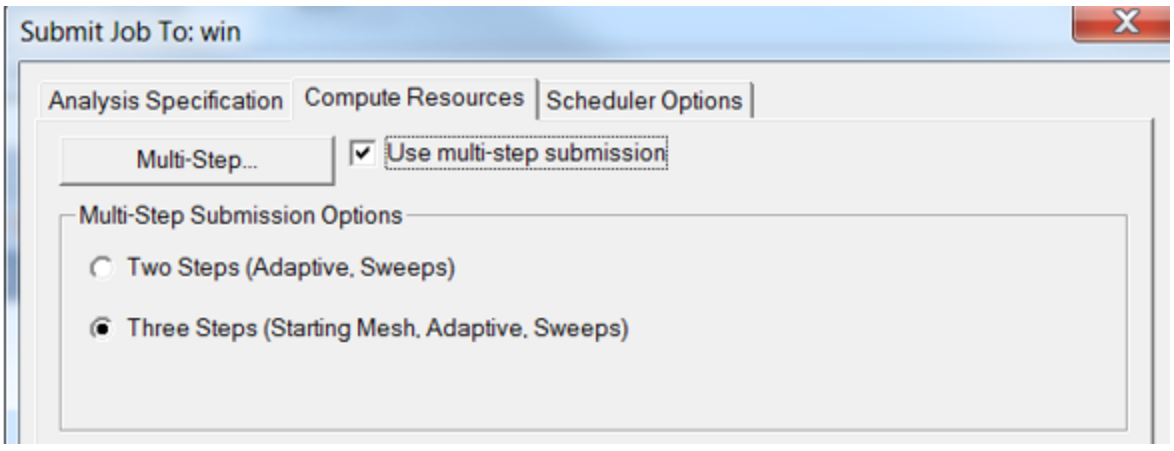
Multi-step job submission allows you to divide the simulation of a project on a cluster as two or more jobs, each of which has unique resource specifications. For example, creating the initial mesh and doing adaptive refinement can use a single machine, while frequency sweeps can easily be distributed 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.

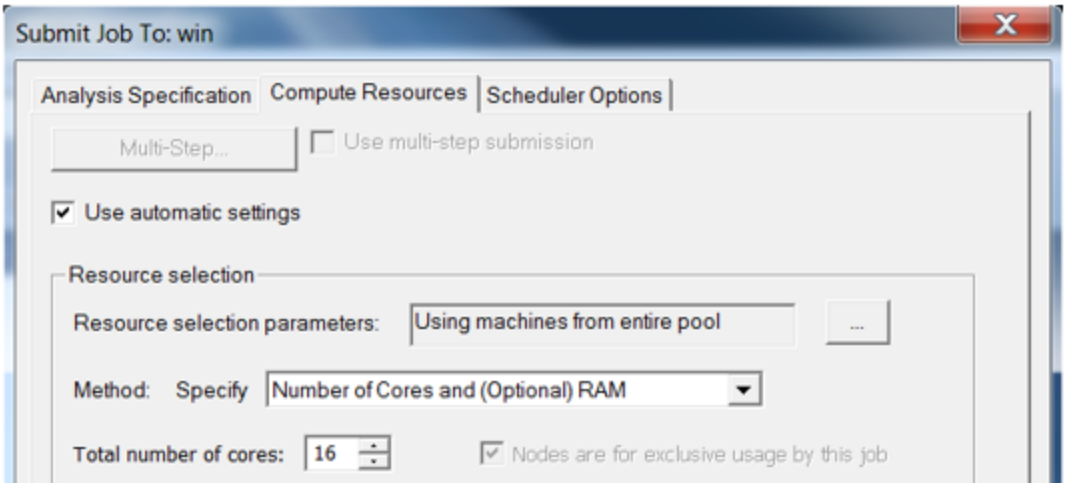
How-to Specify Multi-Step Job Submission

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

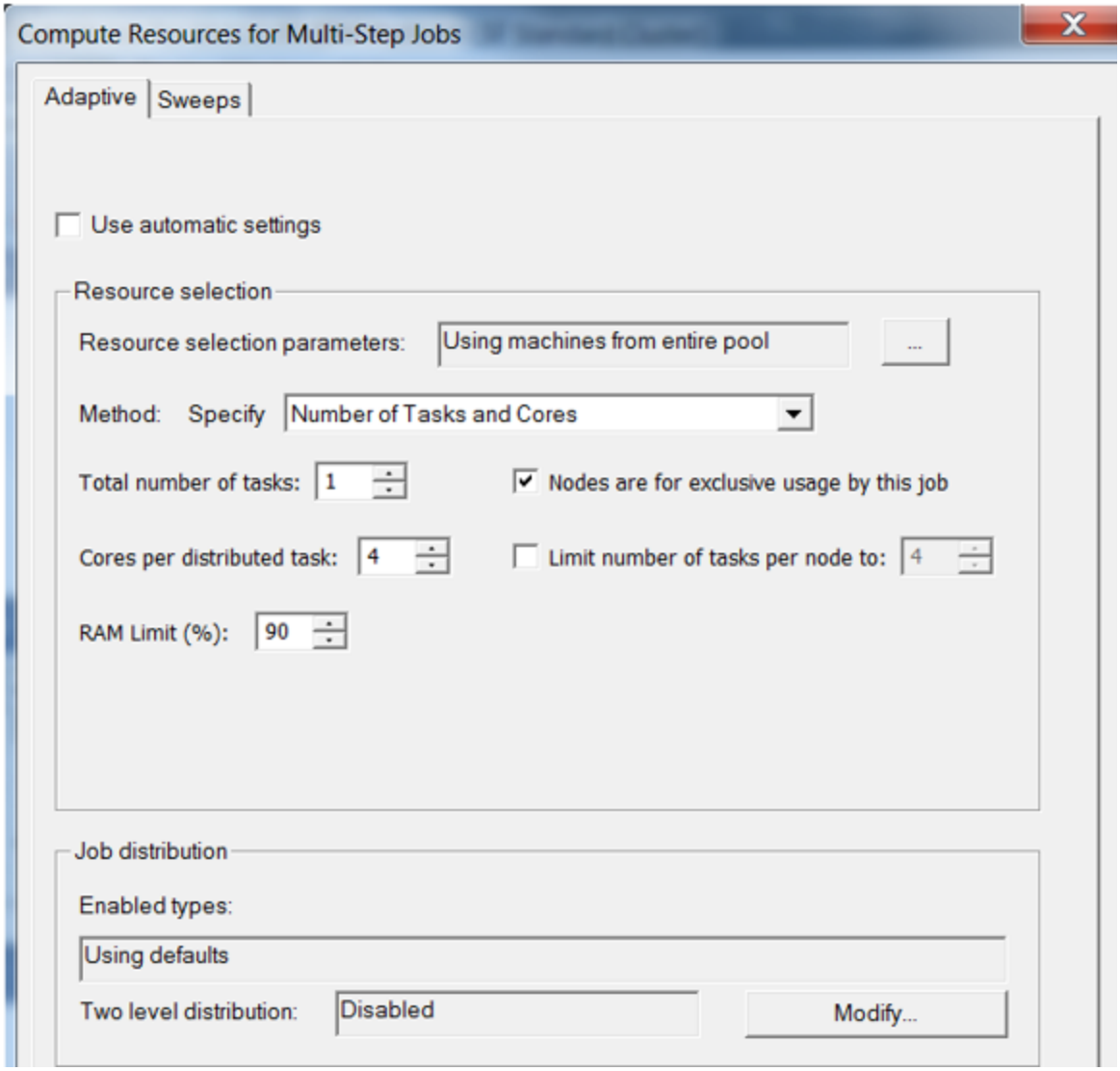


On the **Compute Resources** tab, check the **Use multi-step submission** box and select the appropriate submission option:

- Two Steps for Adaptive and Sweeps
- Three Steps for Mesh, Adaptive, and Sweeps

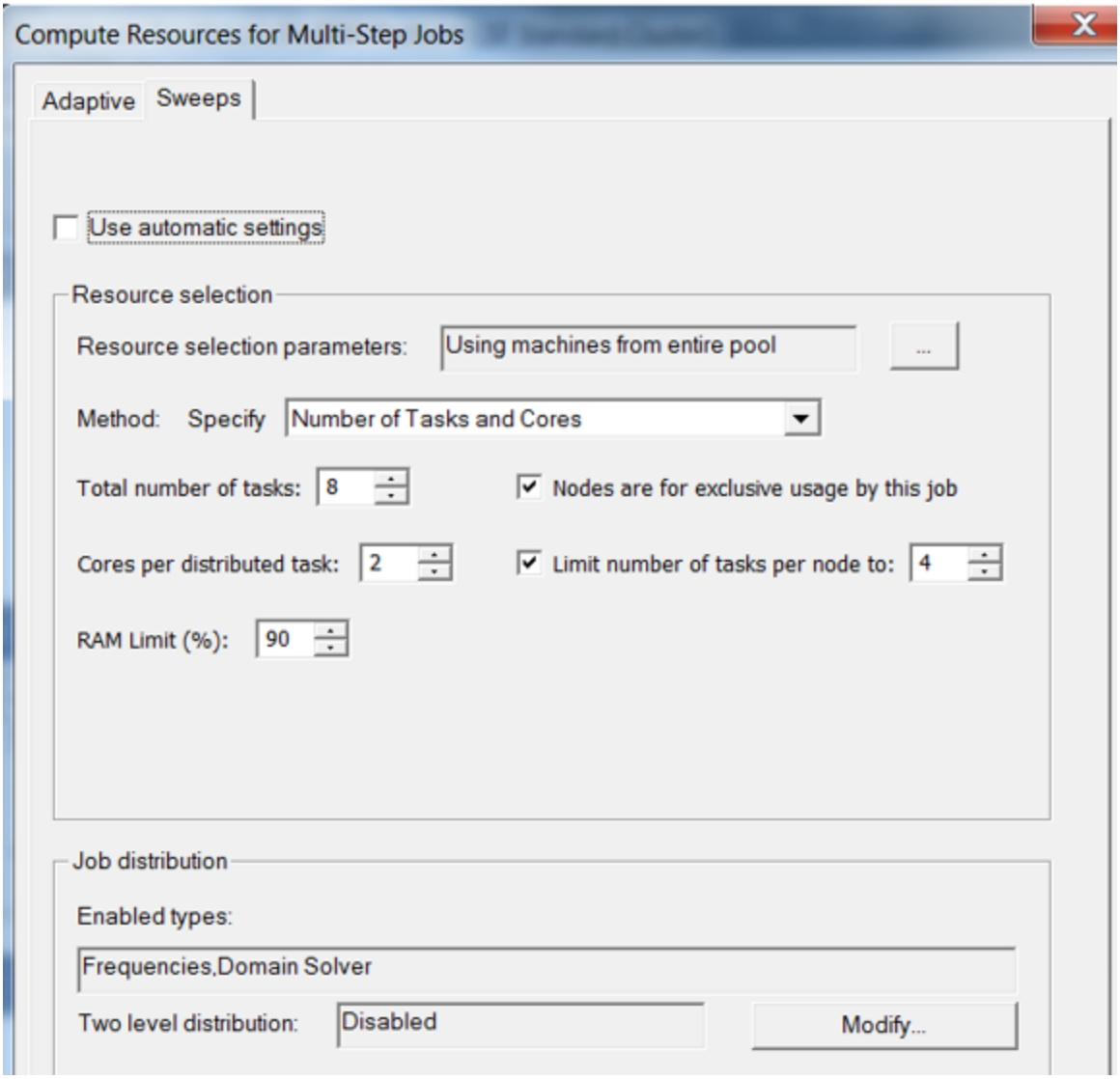
Then click **Multi-Step…**, which will bring up a **Compute Resources for Multi-Step Jobs** dialog box where there is a tab for each step used, that is for potential for Mesh, Adaptive, and Sweeps.

Ansys Electromagnetics Suite 2026 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.

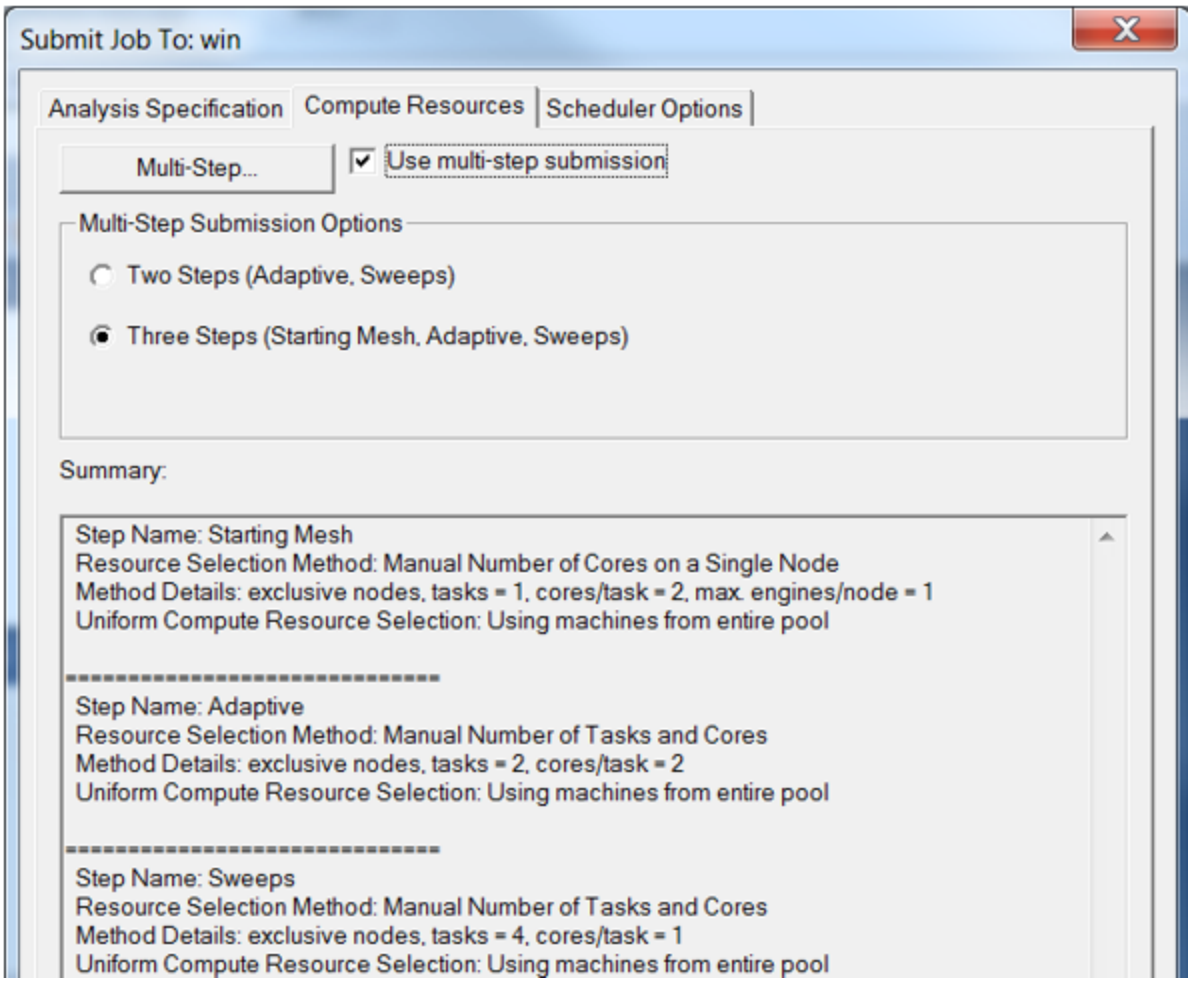
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



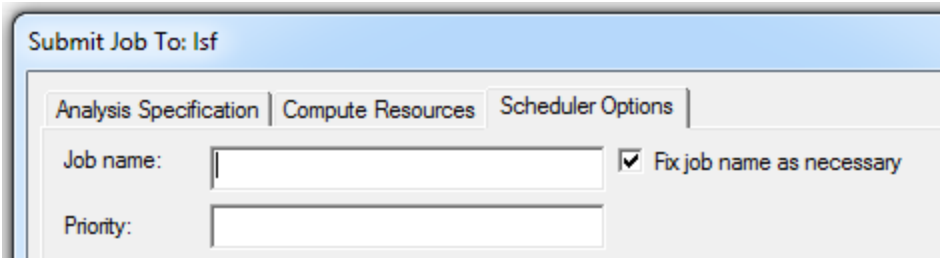
Set the desired compute resources for each step and click **OK**.

The Summary field of the **Compute Resources** tab shows a text summary of resource specifications used for each step.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



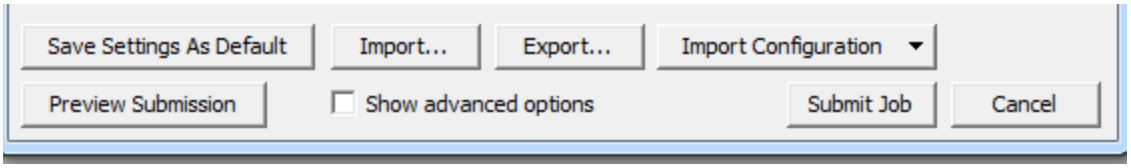
If desired, set the job name on the **Scheduler Options** tab. When the jobs are submitted, each will have a name given by job name (if any) appended with the name of the step for the job. For example, if the job name entered is “MultiStep”, then the individual jobs might be named “MultiStep-Adaptive” and “MultiStep-Sweeps”.



Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Ansys Electronics Desktop does not support Linux to Windows job submission.

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`’

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		SLURM:'{"Proxy":"slurm"}'

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

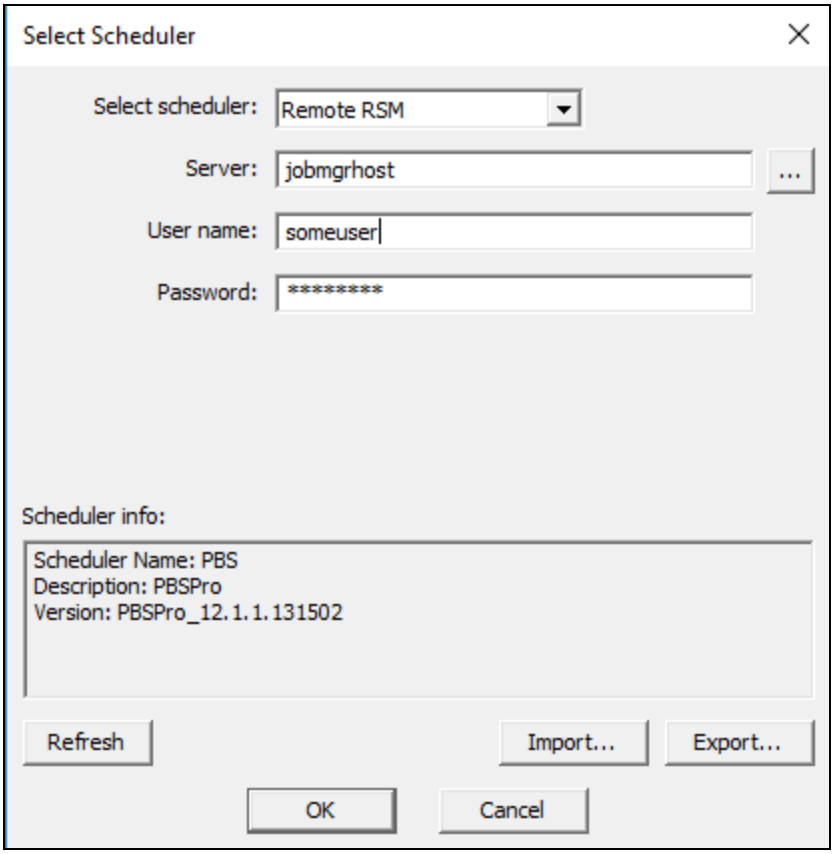
This feature may be used with all Linux schedulers for which job submission from the GUI is supported:

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

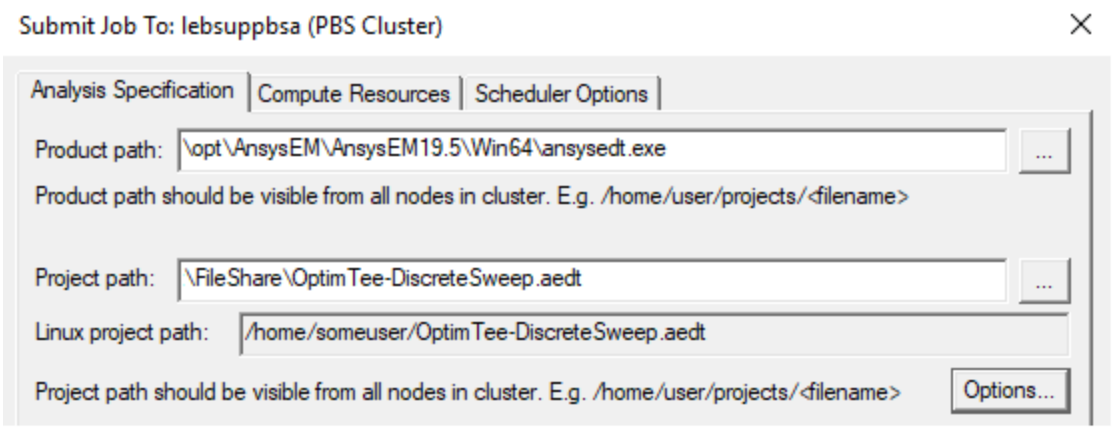


Submit Job

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.

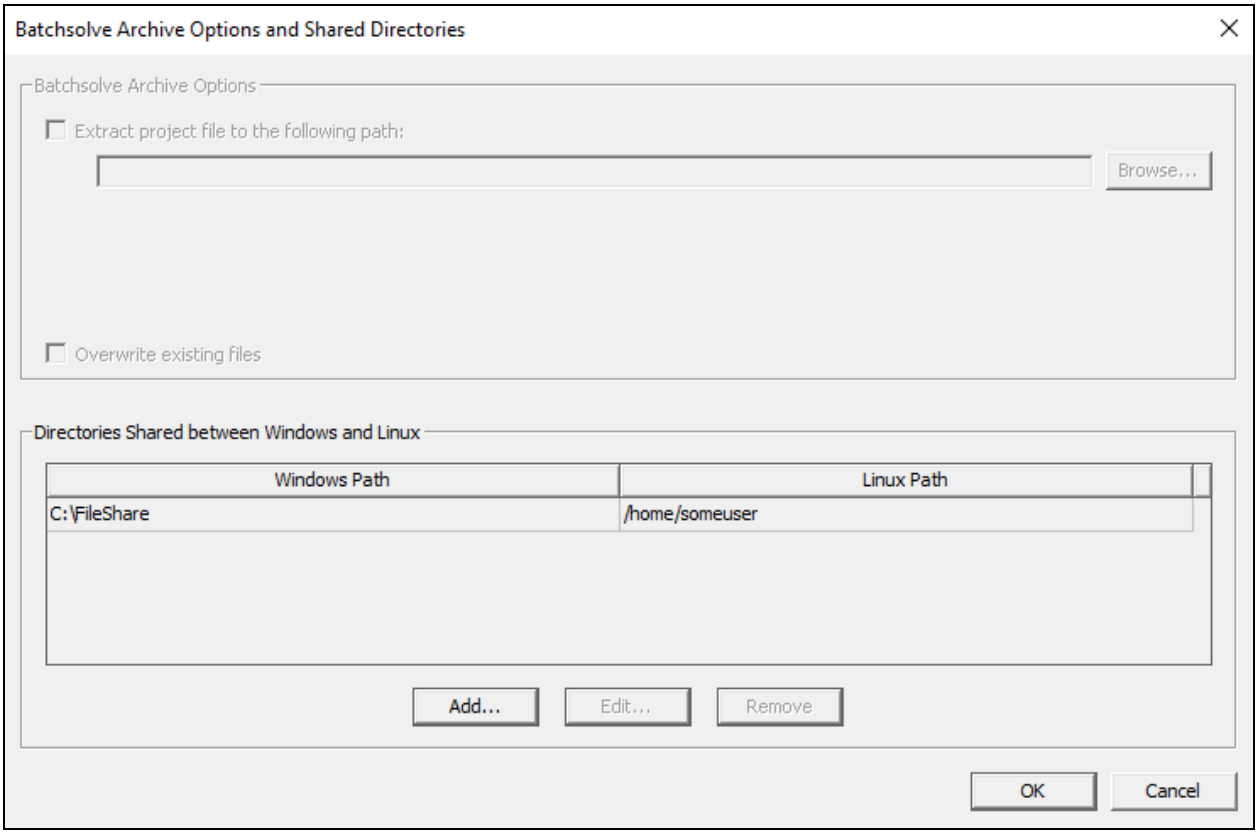
You must enter the Windows path of the project file in the Product path edit control or use the browse button (labeled "...") to select the Windows path of the project file. If the Linux path of the project file can be determined from the specified Windows path and the directories shared between Windows 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



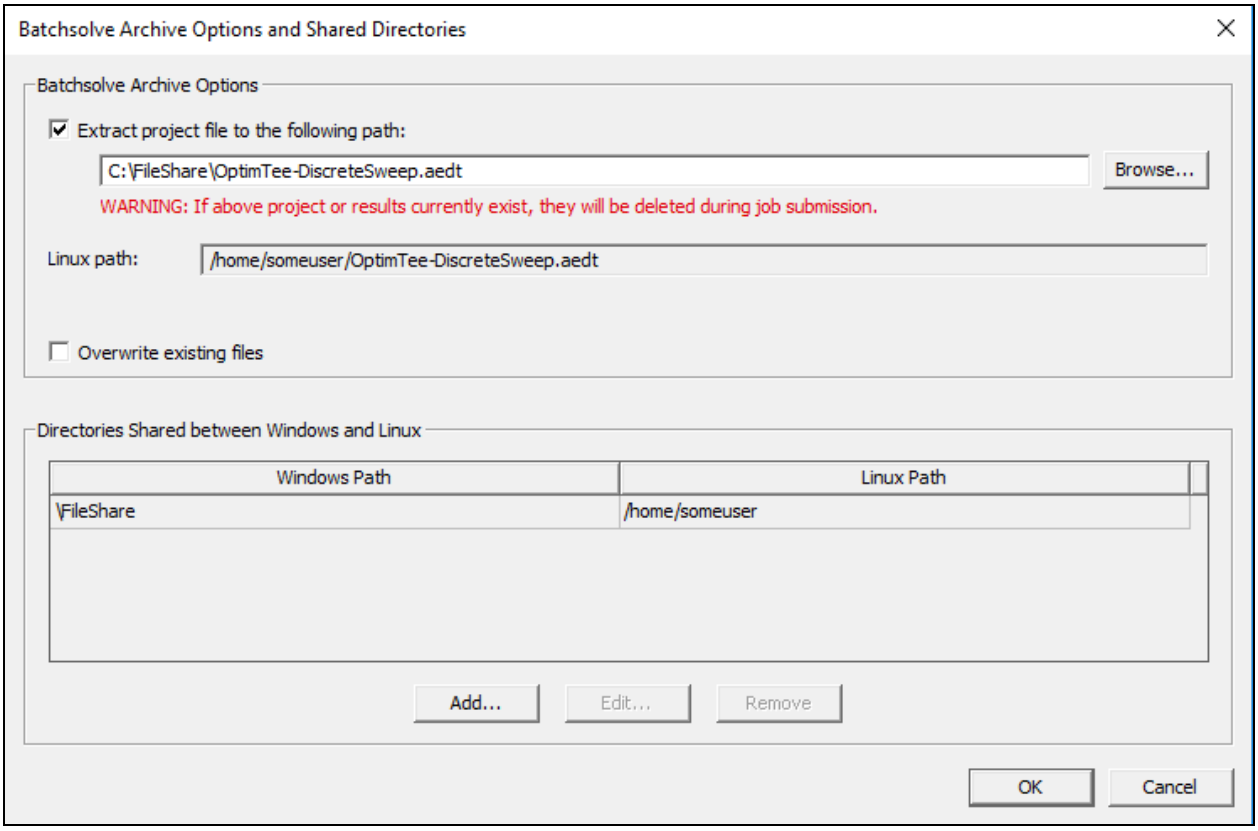
The **Options** button opens a window that you can use to specify archive options for a job. The lower portion of this window allows you to specify one or more directories shared between Windows and Linux. The mapping of directories between Windows and Linux is shown in a grid which displays the Windows 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



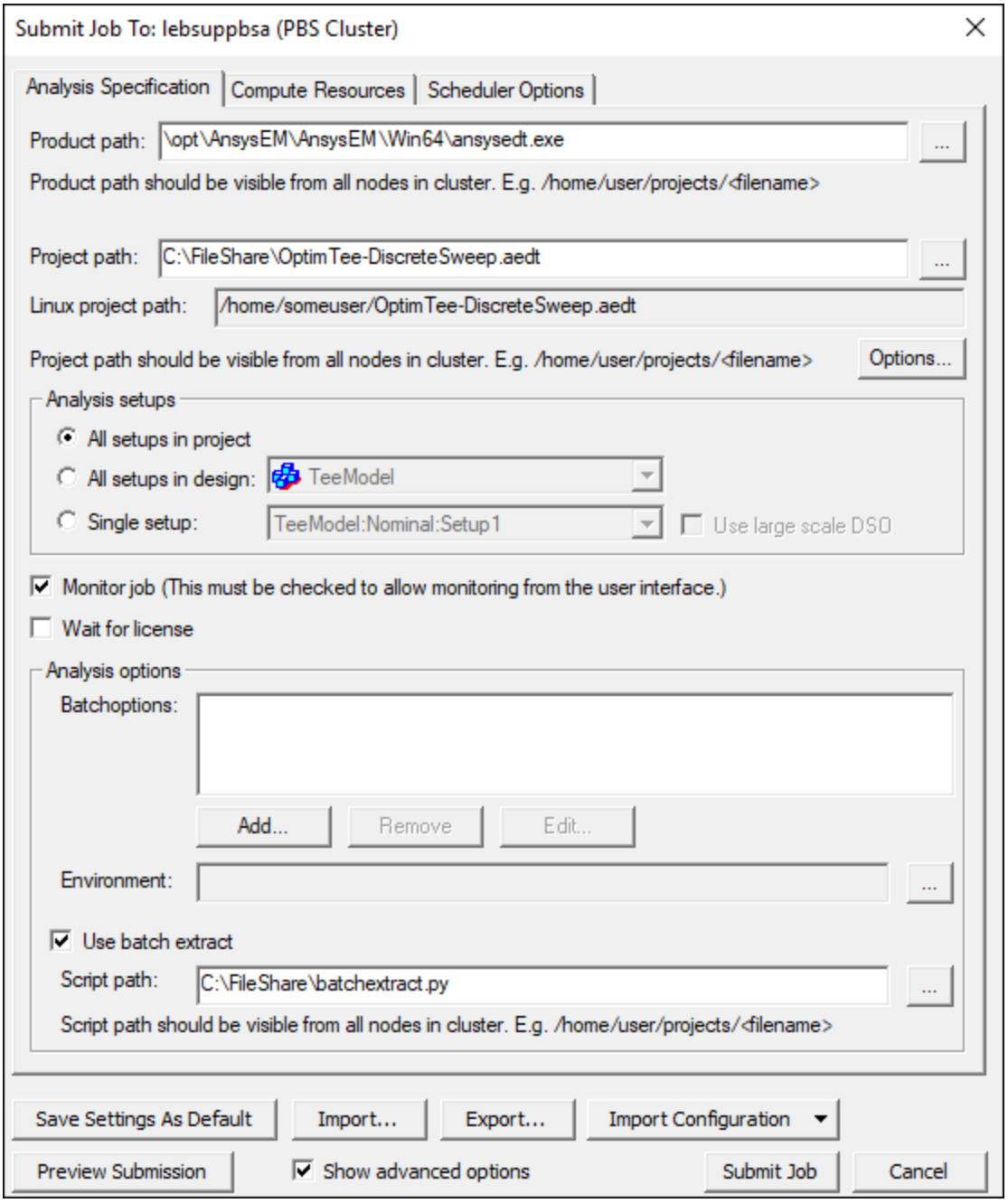
If you specify a project in an archive, the window may be used to specify the Windows pathname of the project to be extracted from the archive. If this is done, the Linux pathname of the target project is determined from the directories shared between Windows and Linux and shown in the upper portion of this window.

Ansys Electromagnetics Suite 2026 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 2026 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 2026 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 Burst Computing is 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 2026 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** > **IcepakScripting 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 Burst Jobs
- Monitoring Large Scale DSO Jobs
- Web Client for Batch Solve Monitoring and Reporting

Monitor Job Window

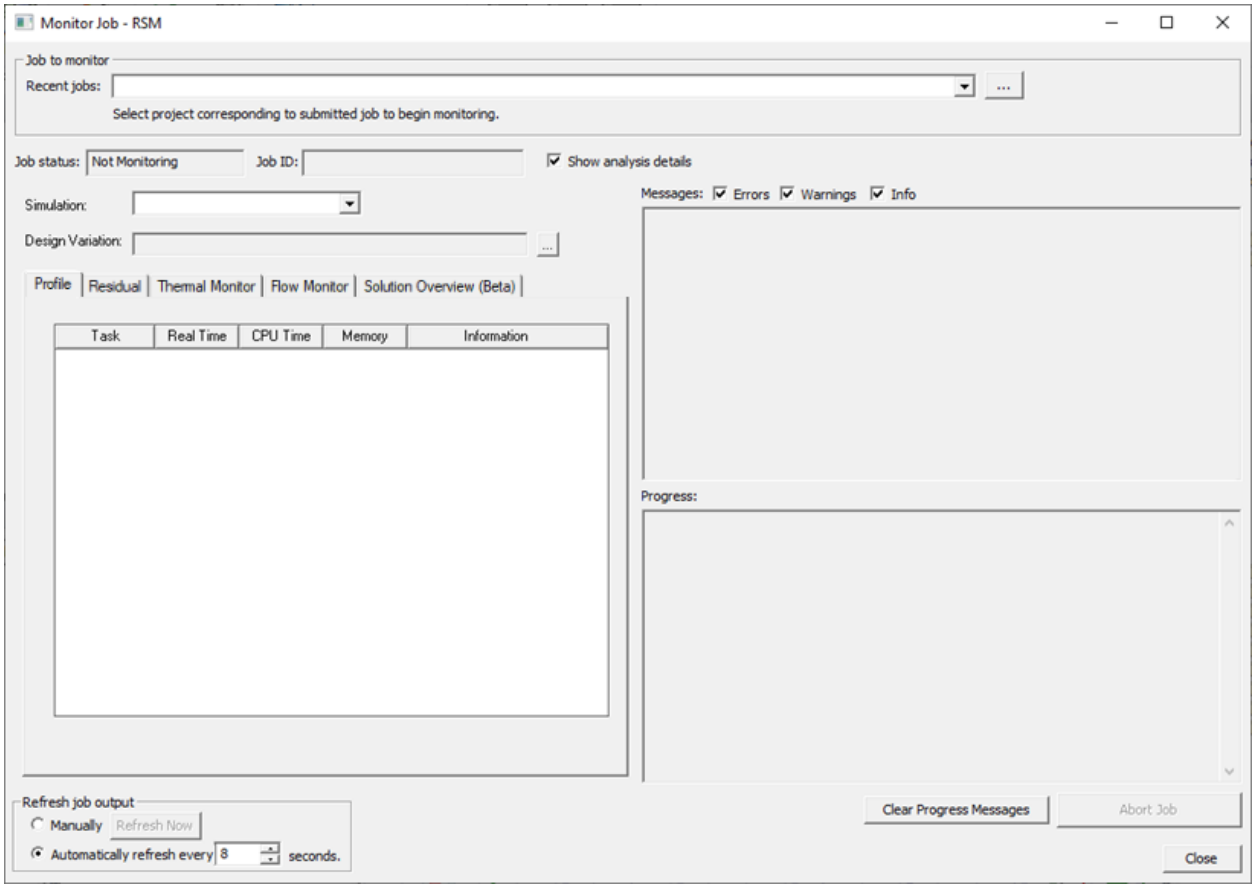
The **Monitor Job** window allows you to monitor the progress and status of jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

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 2026 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:

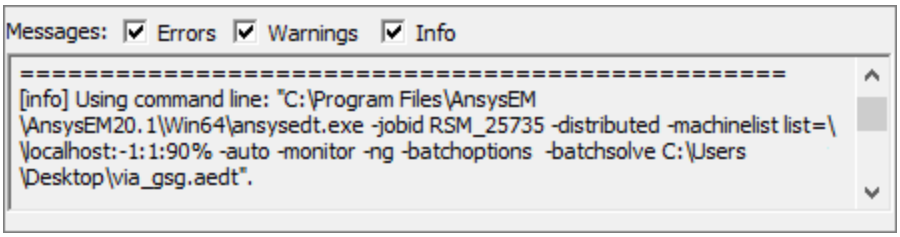
- **Job to Monitor** – Allows you to select either a recent job, or a project containing the job you want to monitor. You can select an archive.
- **Job Status** – The current job status. The normal progression is: Starting Monitoring, Queued, Running, Shutting Down, Completed. A status of Unknown may 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 2026 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

- **Residual Tab** – Displays the solution continuity, velocity, and energy residuals.
- **Thermal Monitor Tab** – Displays data for thermal monitors.
- **Flow Monitor Tab** – Displays data for flow monitors.
- **Solution Overview Tab** (beta) – Displays thermal and flow data for boundary conditions and fan components.
- **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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The arrow button opens a menu that allows you to either **Abort** that analysis, or perform a **Clean Stop**.

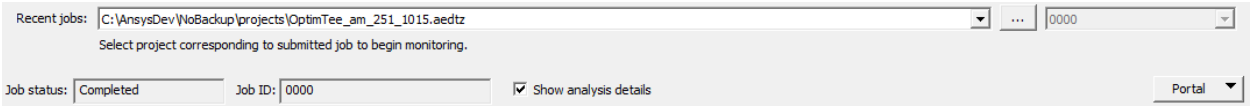
- **Refresh Job Output** – Allows you to select either manual or automatic refresh for the **Monitor Job** window.

To display less information, deselect **Show analysis details**. This removes the **Simulation**, **Design Variation**, **Profile**, and **Convergence** information, instead displaying only **Messages** and **Progress**.

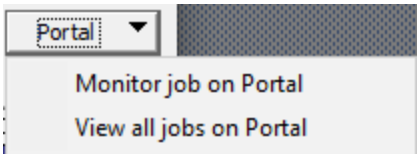
2. Click **Close** when you are done monitoring.

Monitoring Ansys Cloud Burst Jobs

When you submit a job to Ansys Cloud Burst Computing, 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**.



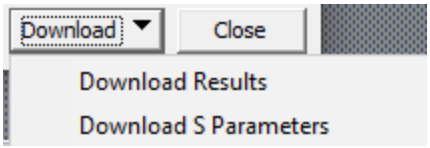
The **Portal** drop-down button provides a link to the Ansys Portal, where job details and additional monitoring information are available. For more information, see the Ansys Portal Guide.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



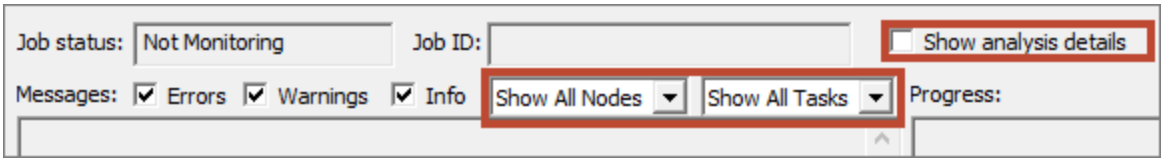
Monitoring Large Scale DSO Jobs

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.



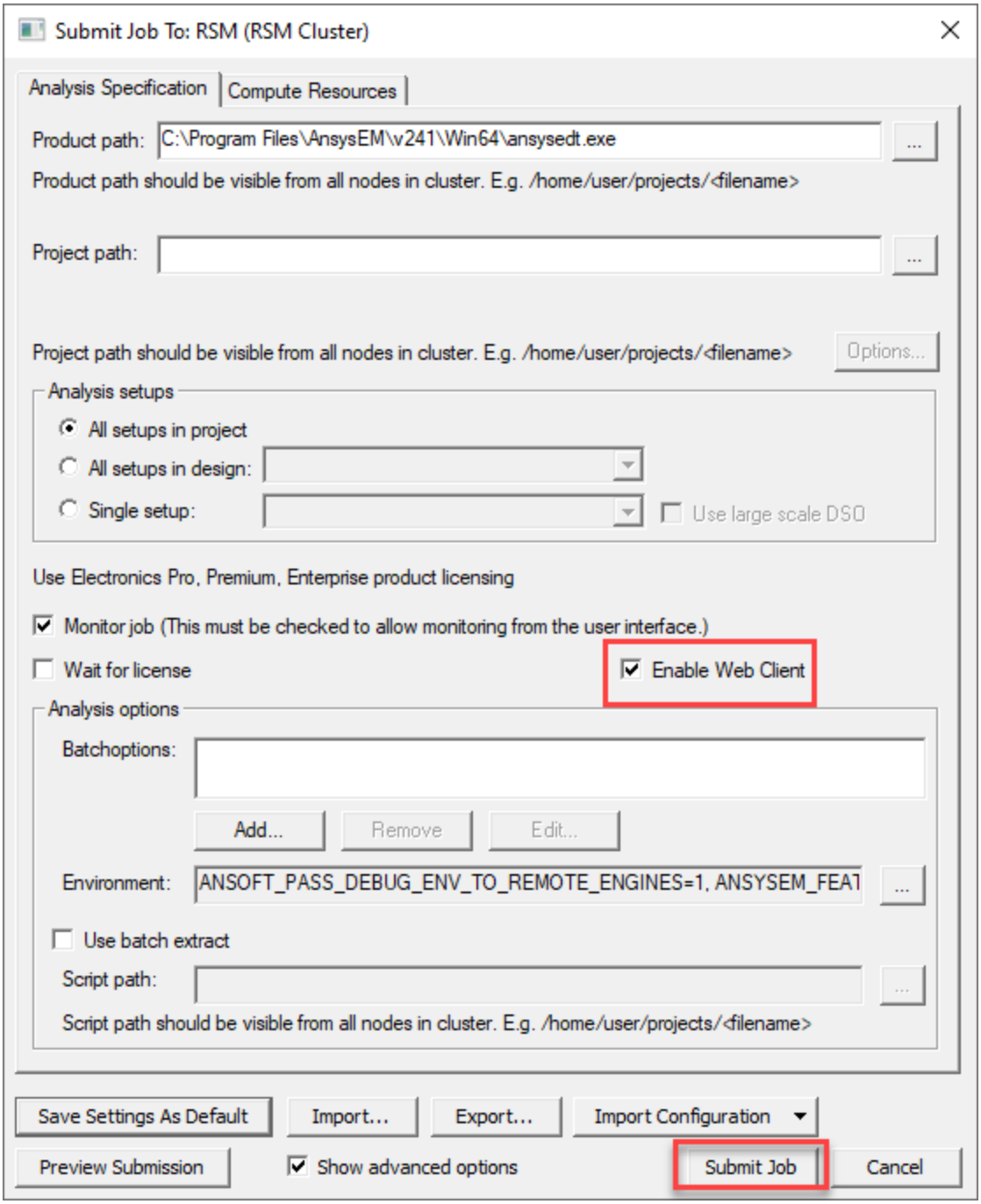
To leave this mode, select **Show analysis details**.

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 under **Tools>Options>General Options> Desktop Configuration** for "Enable Web Client for Monitoring," you will have access to the feature.

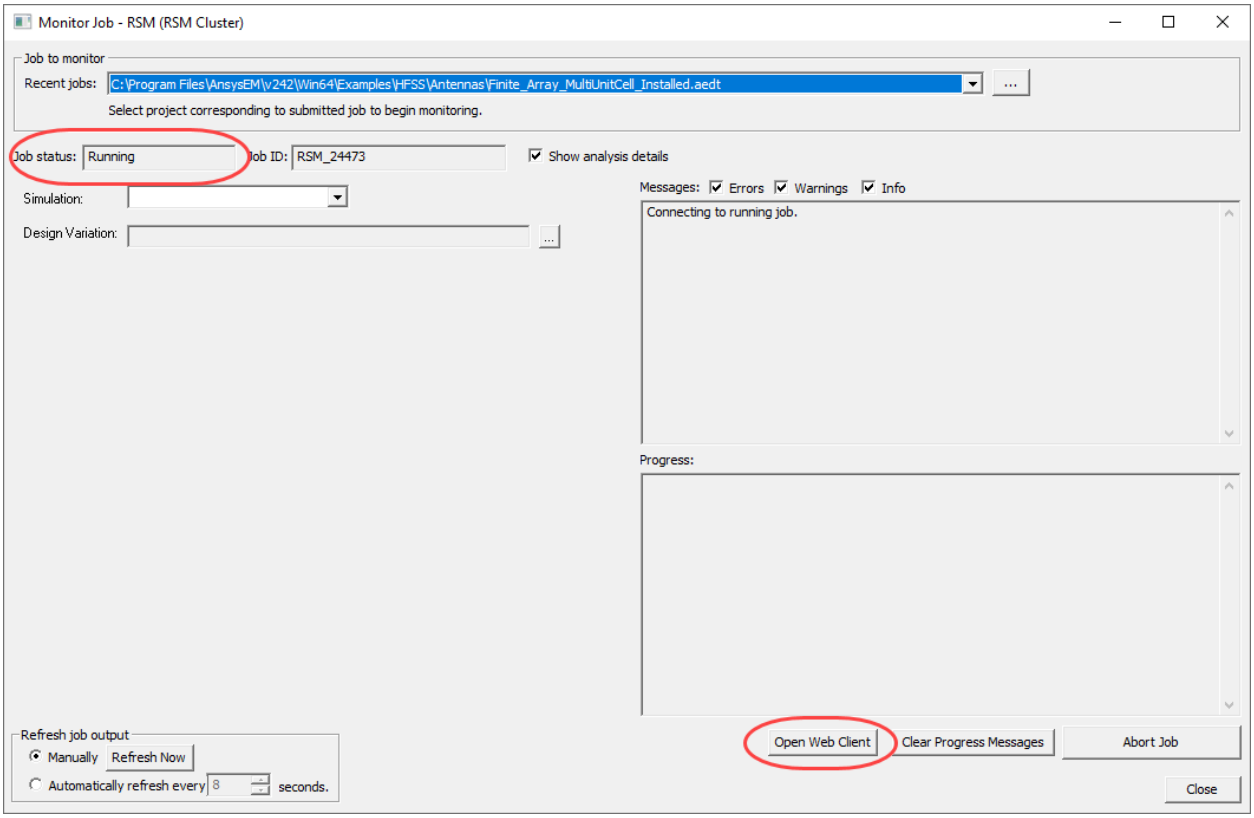
After you have selected the scheduler to use, when you access the **Submit Job To:** dialog, you well see Enable Web Client checked.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



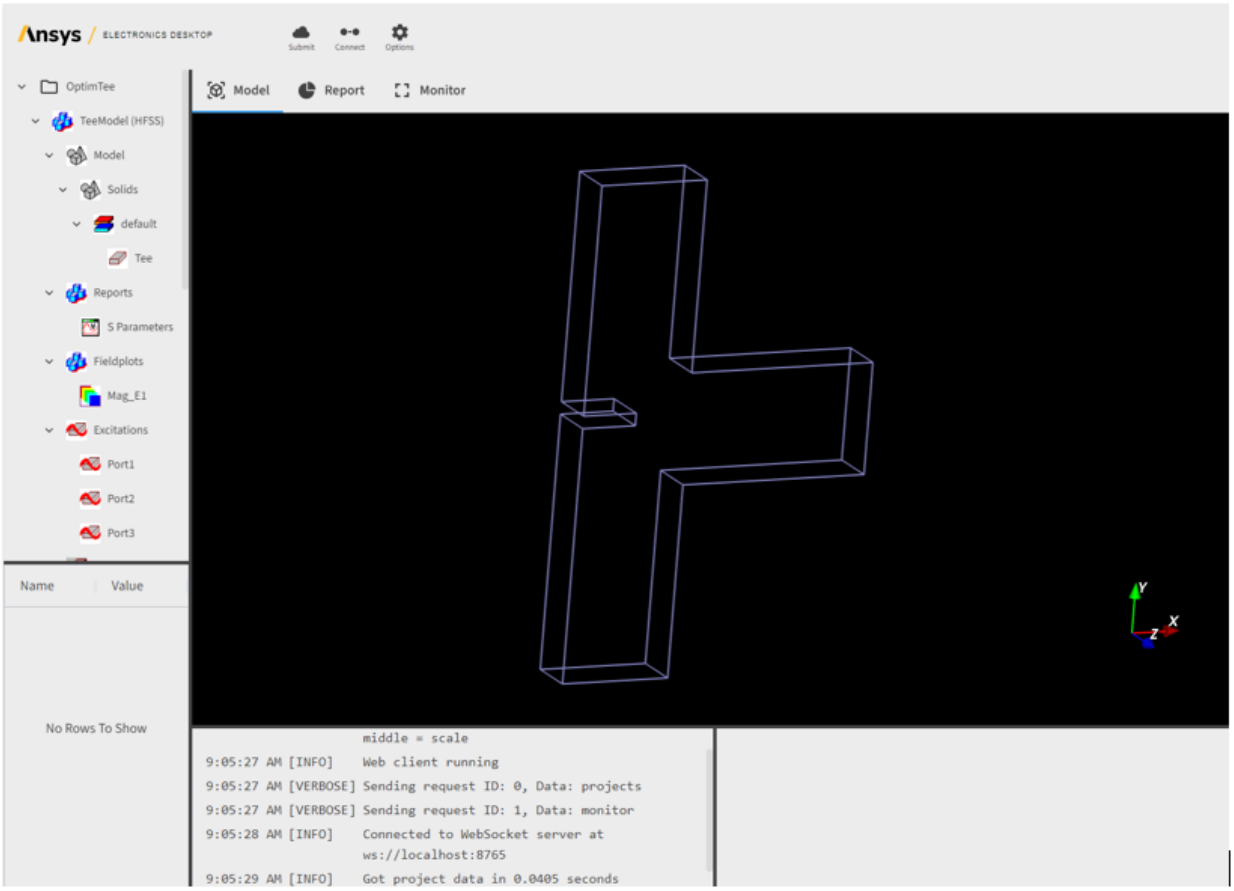
After specifying the project path and setting other parameters, you can then select **Submit Job**. This opens the Monitor Job dialog for your scheduler. Once the **Job Status** changes to Running, press **Open Web Client**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



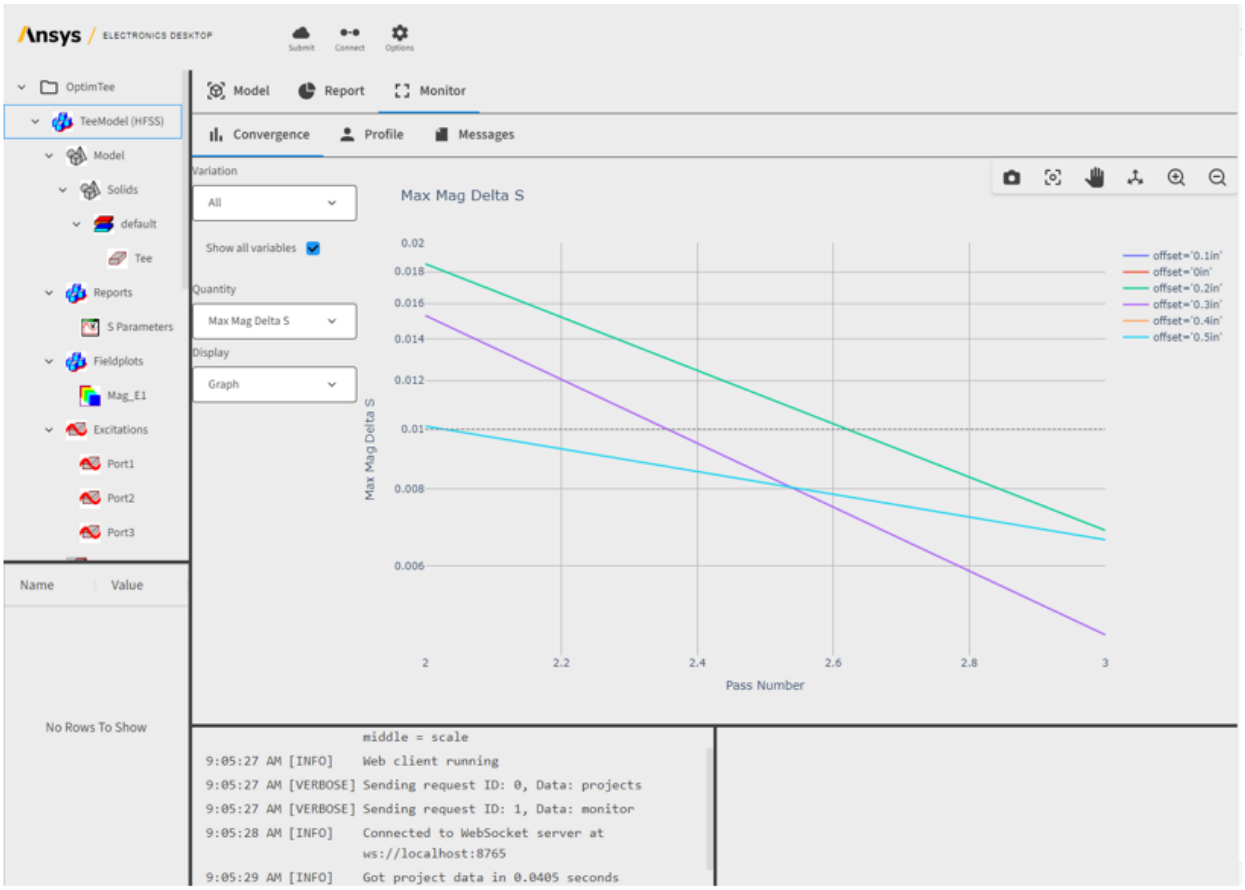
Click **Open Web Client** to open the Web Client in your default browser.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



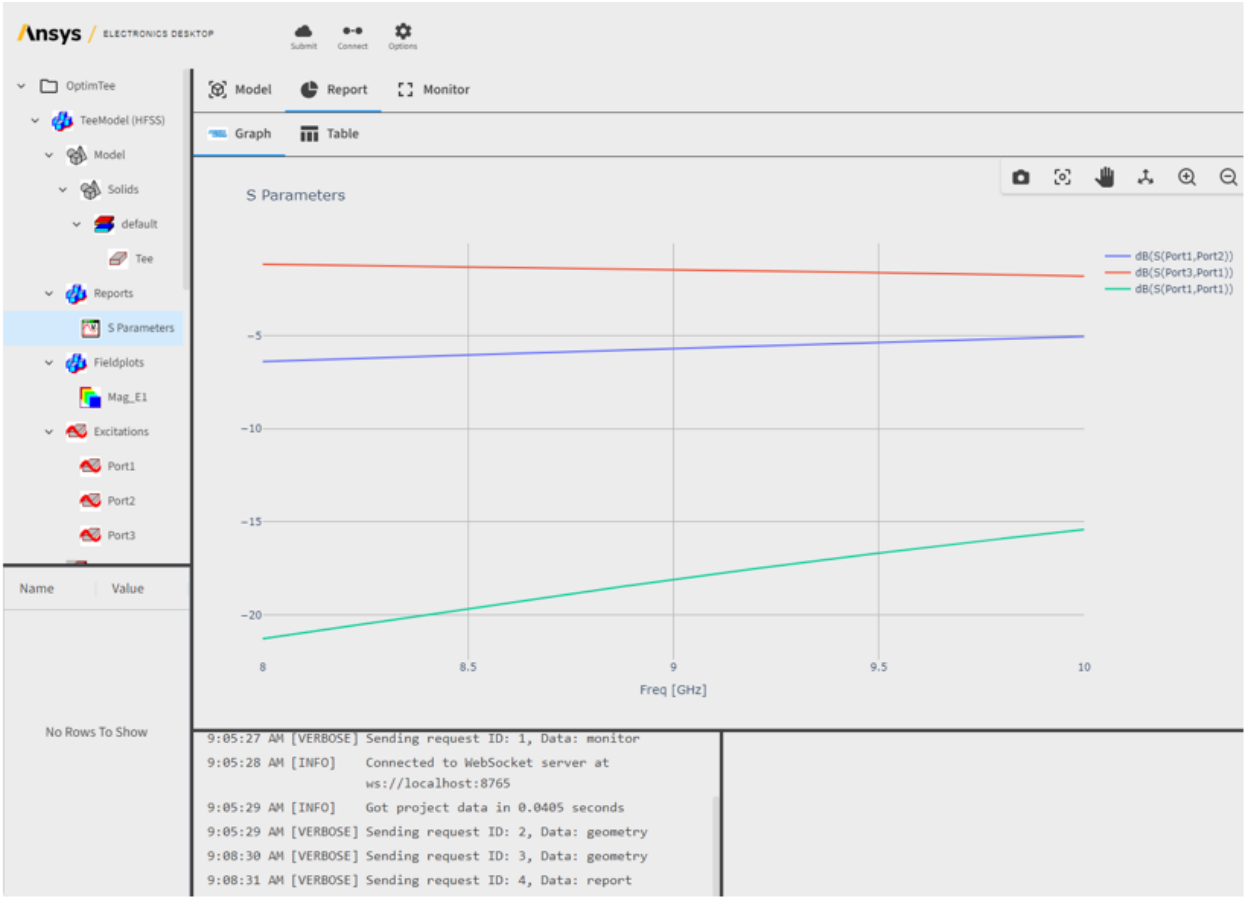
Switch to the **Monitor** tab to gain access to the subtabs for Convergence, Profile and Messages.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



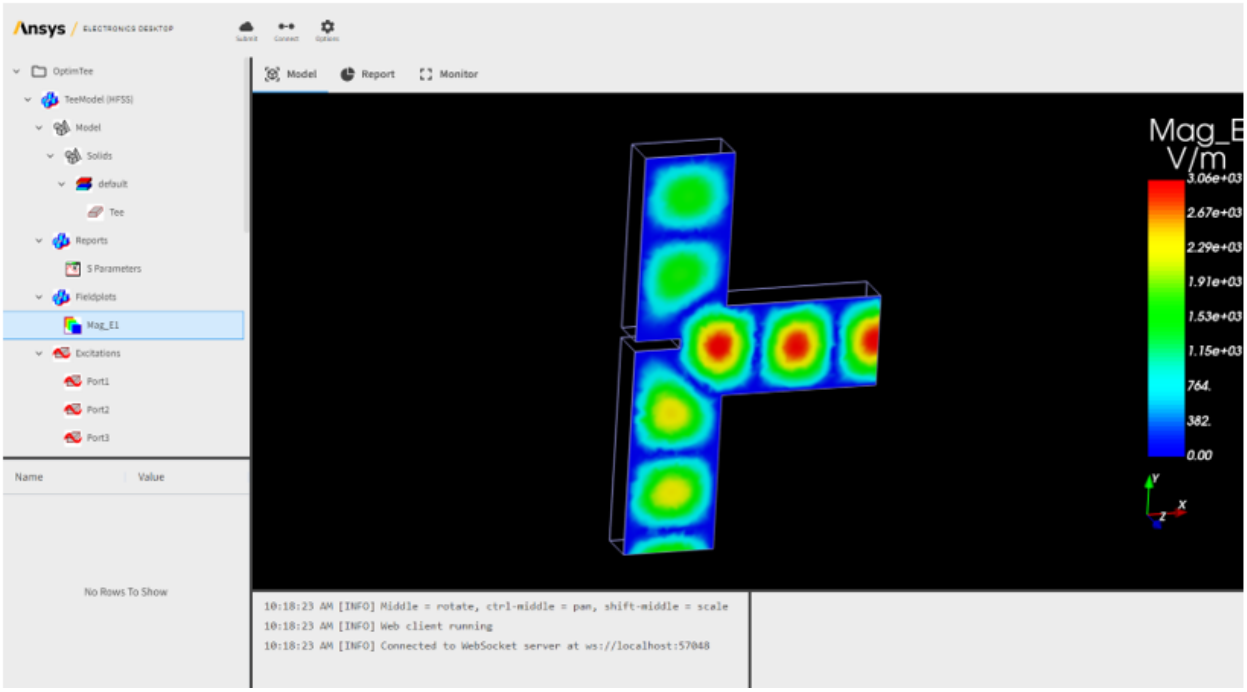
Select a Report in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aborting an Analysis

Opening the Abort Job menu

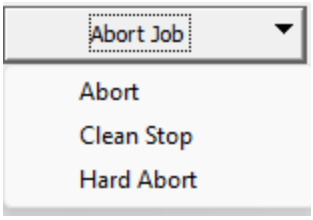
The **Abort Job** pop-up menu can be opened by any of the following options:

- Right click on the **Progress** window.
- Click on the drop-down button next to the progress bar(s).
- Click the **Abort Job** button in the **Job Monitor** dialog.

Abort Methods

To abort any running job submitted through either Ansys HPC Platform Services or Ansys Cloud Burst Compute, you can either **Abort**, **Clean Stop**, or **Hard Abort** the job. Using either of these abort types, debug logs can be collected from all compute nodes in the job.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To end the solution process before it is complete, open the **Abort Job** drop-down menu and select **Abort**.

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, open the **Abort Job** drop-down menu and select **Clean Stop**.

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.

If the selected scheduler is **HPC Platform Services** or **Ansys Cloud Burst Compute**, the **Hard Abort** option will be available under the **Abort Job** button drop-down menu in the **Job Monitor** dialog. Use **Hard Abort** if the simulation is hanging and does not respond to **Abort**. Any intermediate files that can be collected will be placed in the downloaded results under archive **abort_intermediate_results.zip**.

Linux

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 fluent.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

```
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.

Setting MPI Version

When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

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?

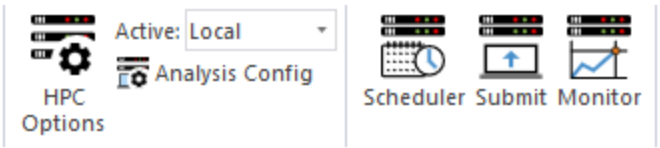
RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, 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...**

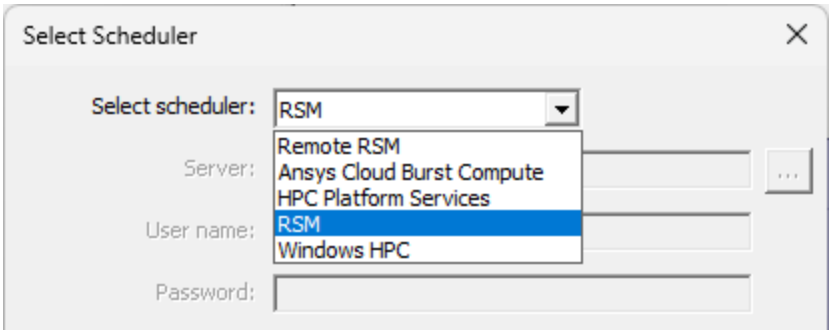


You can also select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



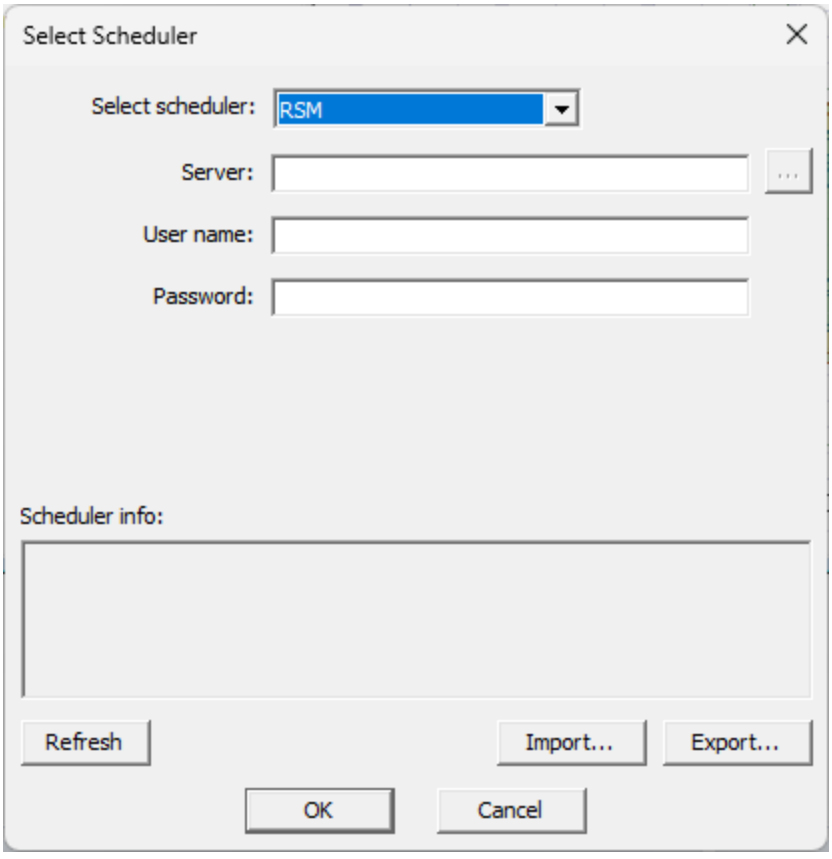
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (list is defined by your environment).



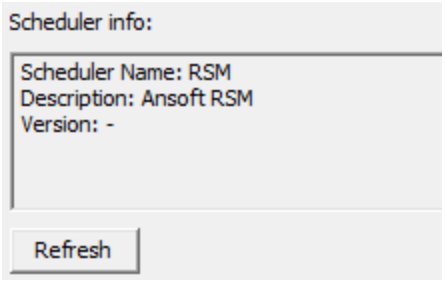
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured, you can select a computer, user name, and password.



After selecting a scheduler, click **Refresh** to display information for that scheduler.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

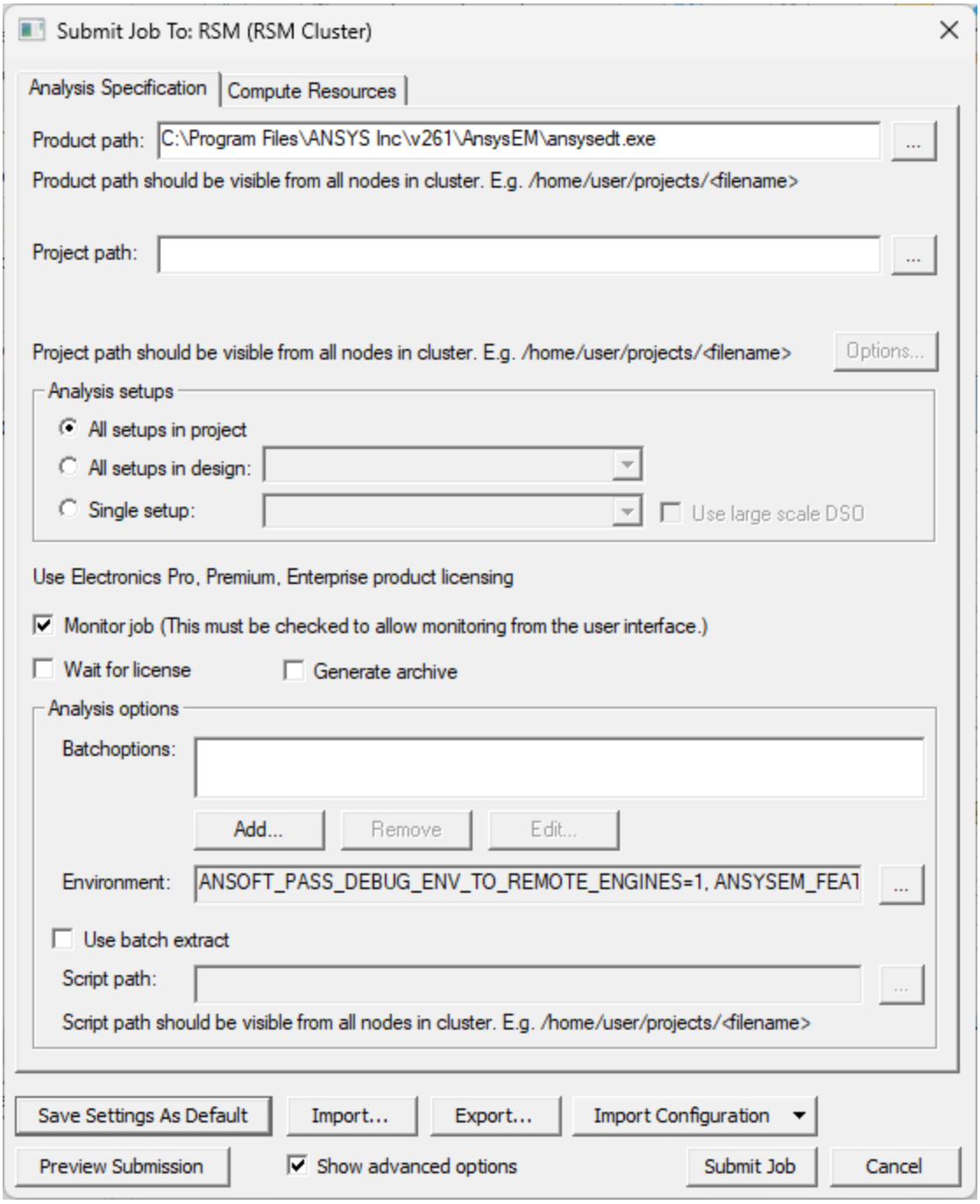
1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).
    - In the RSM environment, you are required to specify 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.

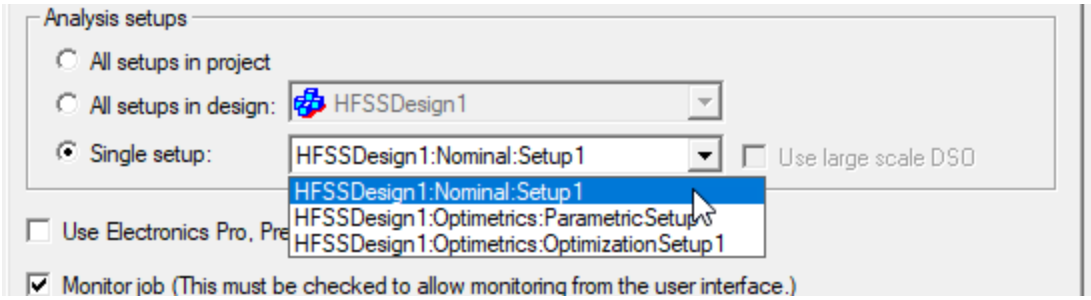
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job window from the right-click menus for Setup or Optimetrics Setup, this field can be pre-populated.

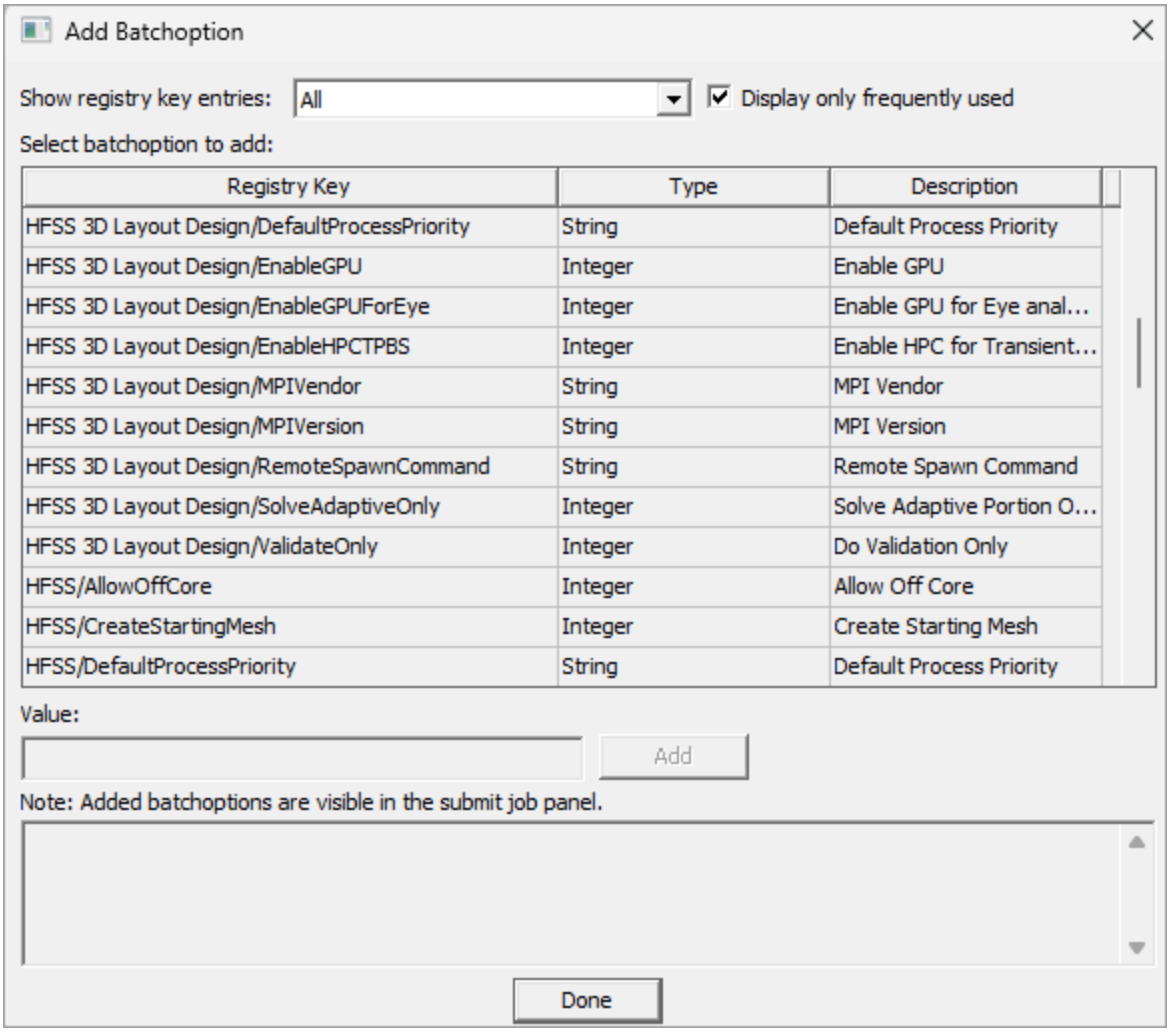


For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

7. Configure the desired analysis options:
   - **Monitor job** – Enable this option to monitor the job from the user interface. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** function or by checking the dialog box that opens when you submit the job.
   - **Wait for license** – specify whether to wait until a license if available before starting a simulation.
   - **Generate Archive** – Generates an archive for the submitted job once it is complete. The project file, batchinfo directory, any external files, and the results directory are combined into the archive. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc...* will be written instead. This function will not overwrite an existing archive.
     Note that if an archive file is selected, this option will be renamed **Repackage results into archive**.
   - **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 Batchoptions field allows you to add additional –`Batchoptions` parameters, either as text, or by using a dialog with selection

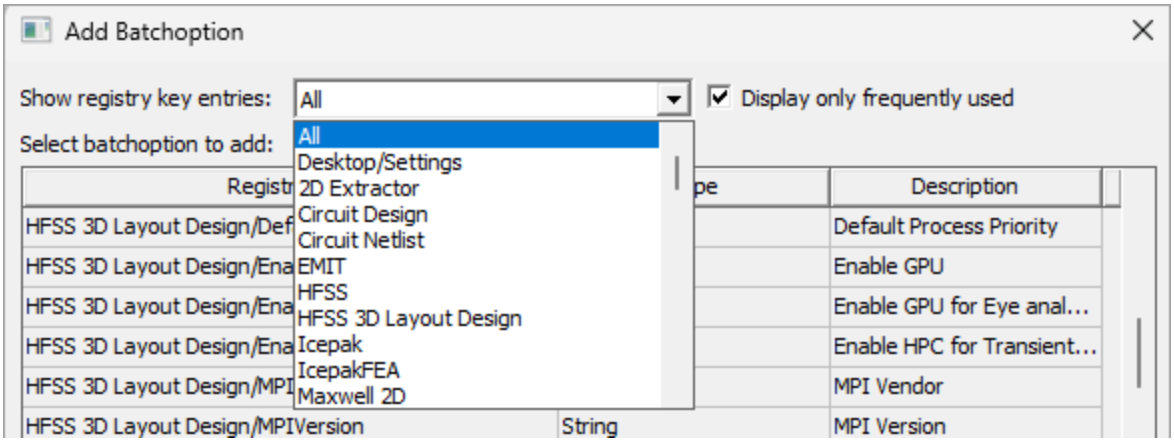
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menus. Click the **Add** button to view the **Add Batchoption** dialog box.

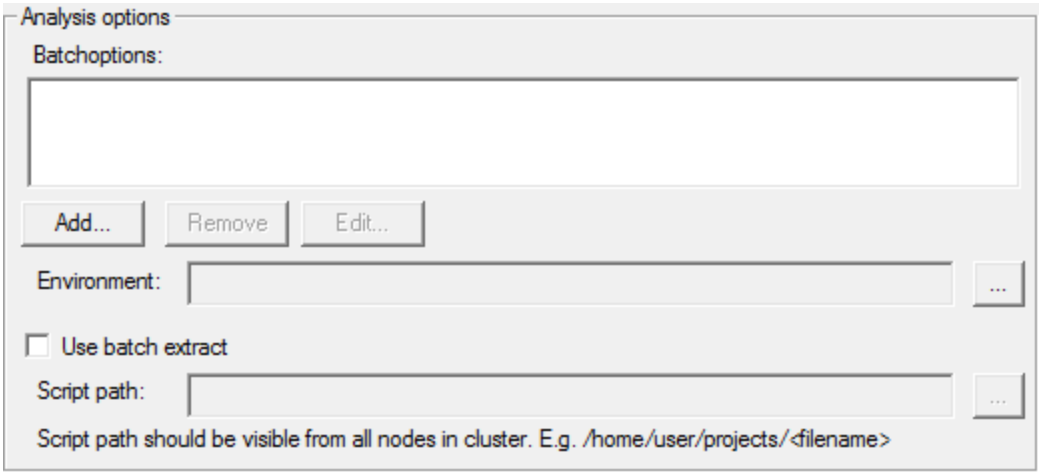


The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

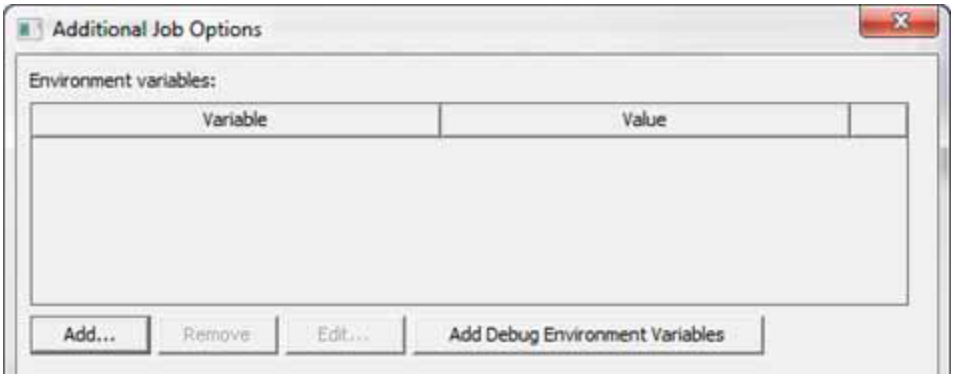


- When you have selected a batch option, you can type the value in the field, and click the **Add** button to add the option to the batch command.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and to display **Use batch extract** setting.

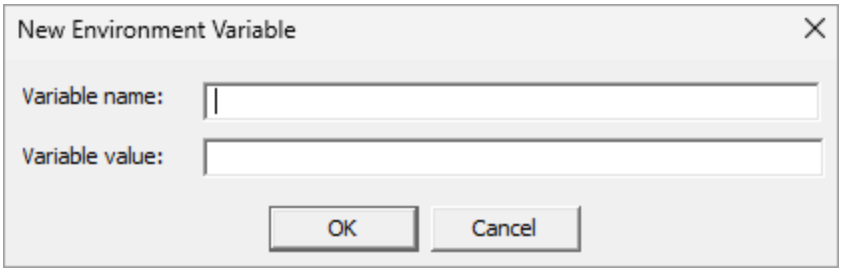


The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

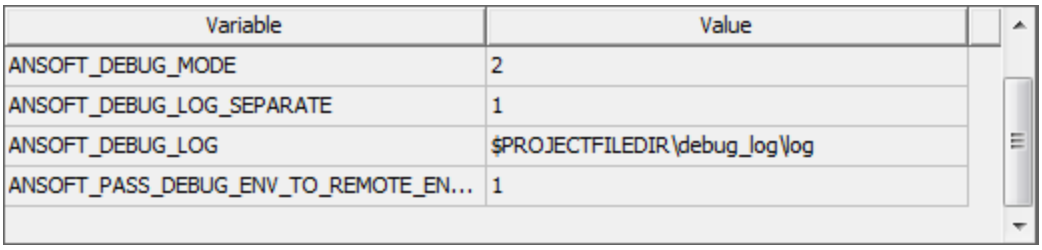
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



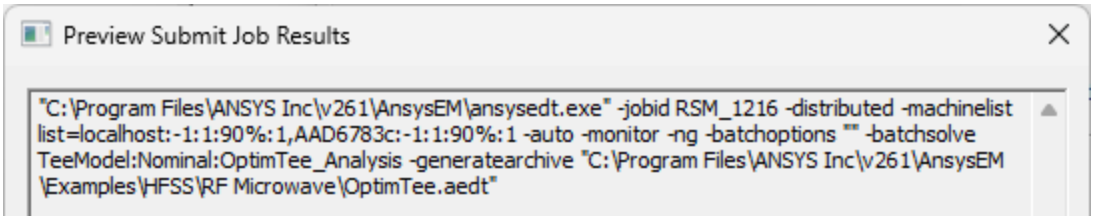
Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



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.

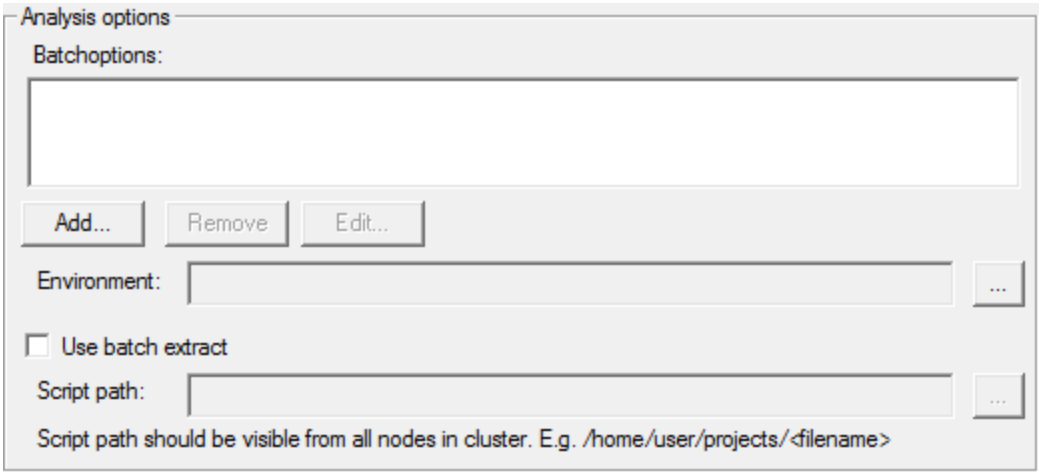
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

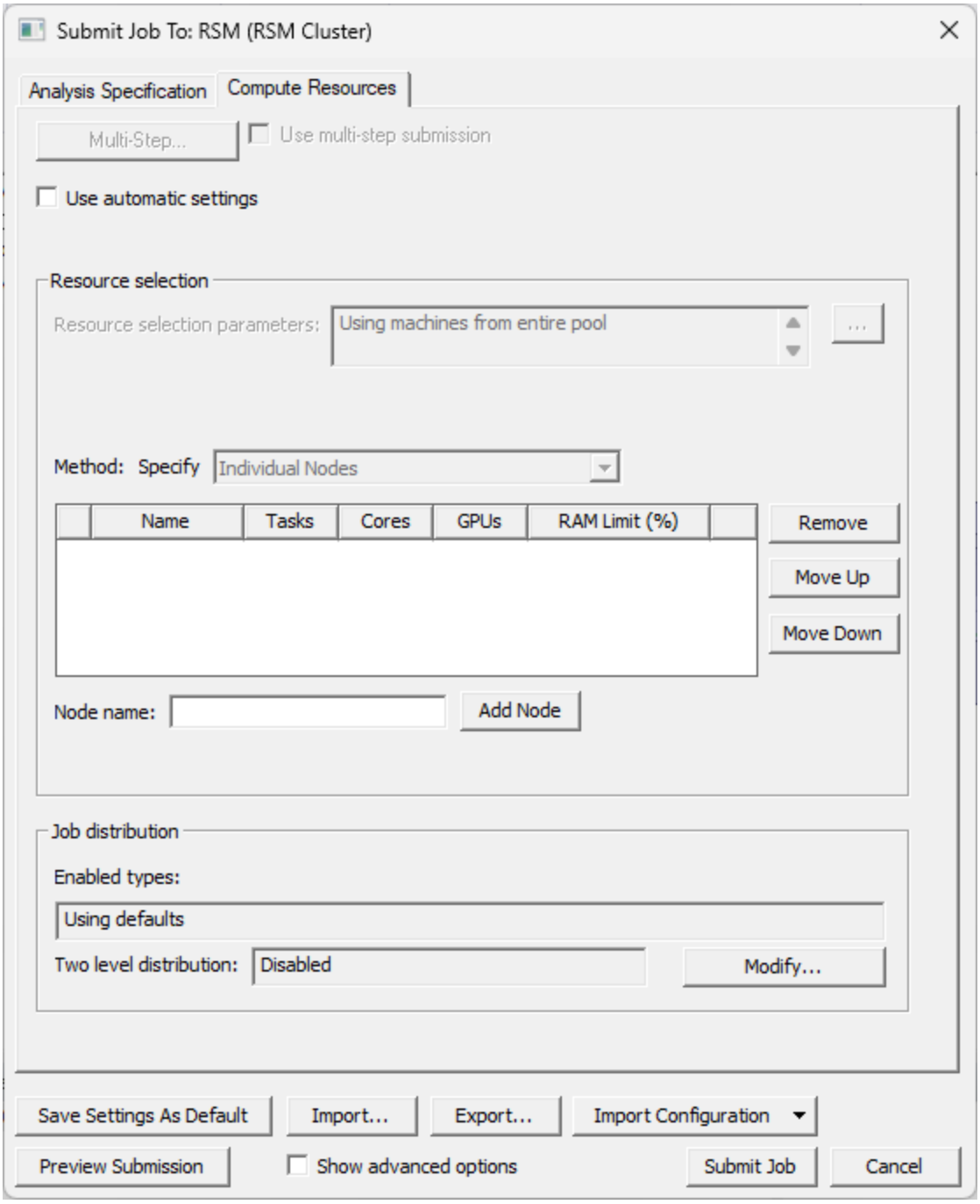
Selecting **Show advanced options for RSM** also shows the Use batch extract fields.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

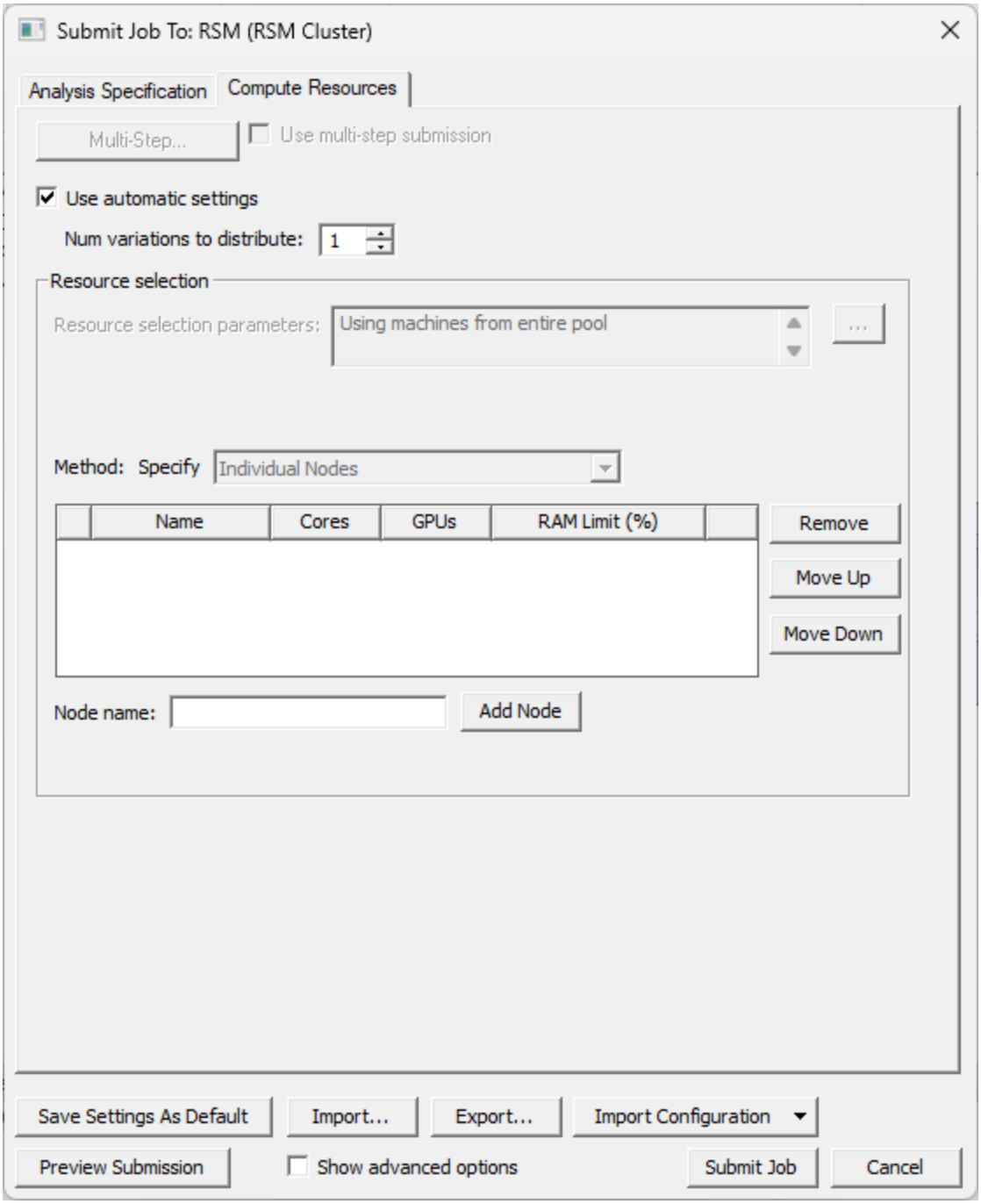
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

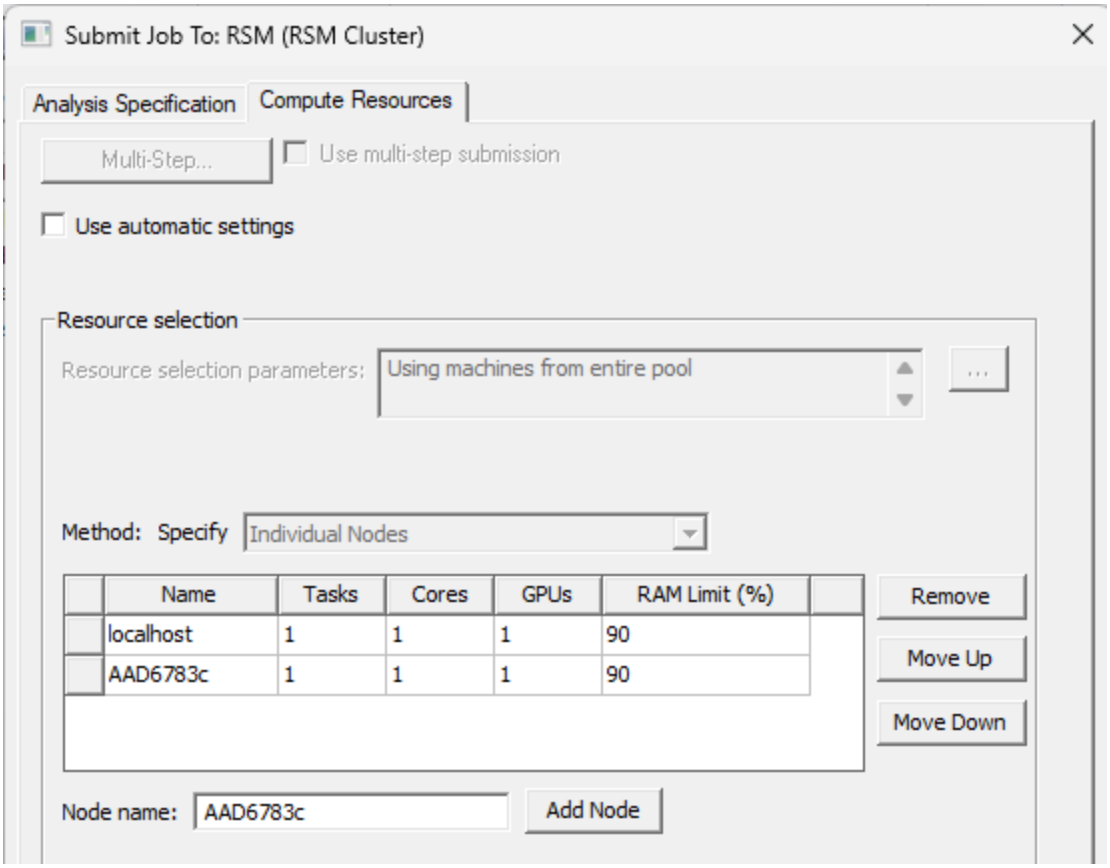
Note:

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 a 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.

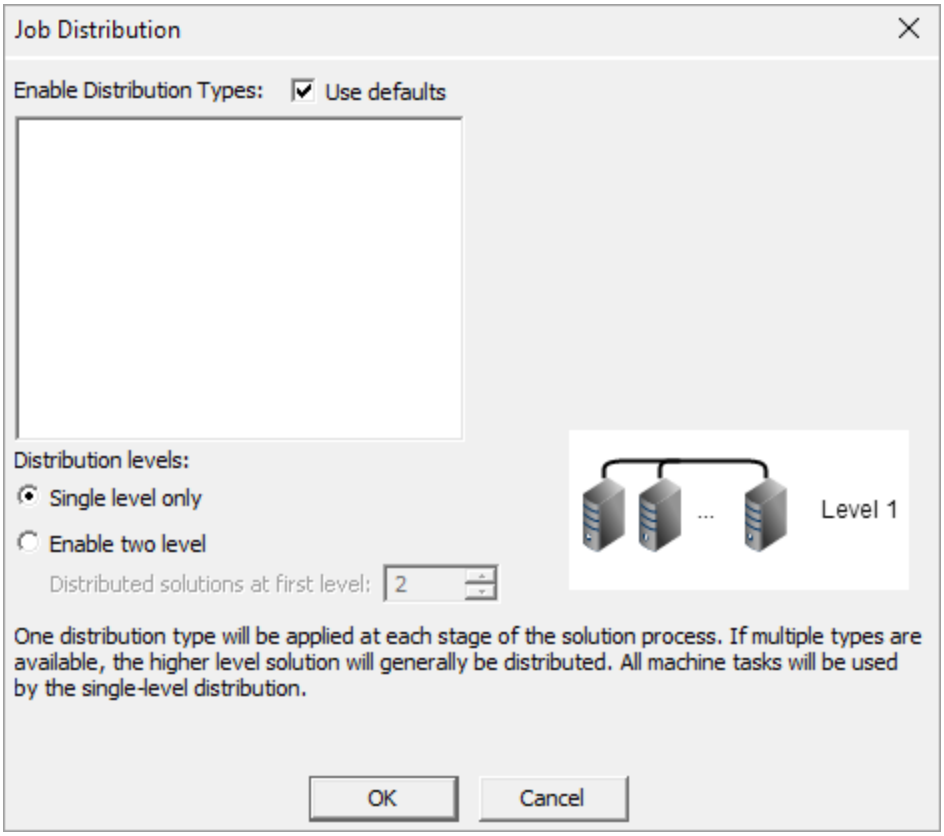


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Distribution

If you disable User automatic selection, you can modify the Job distribution settings.

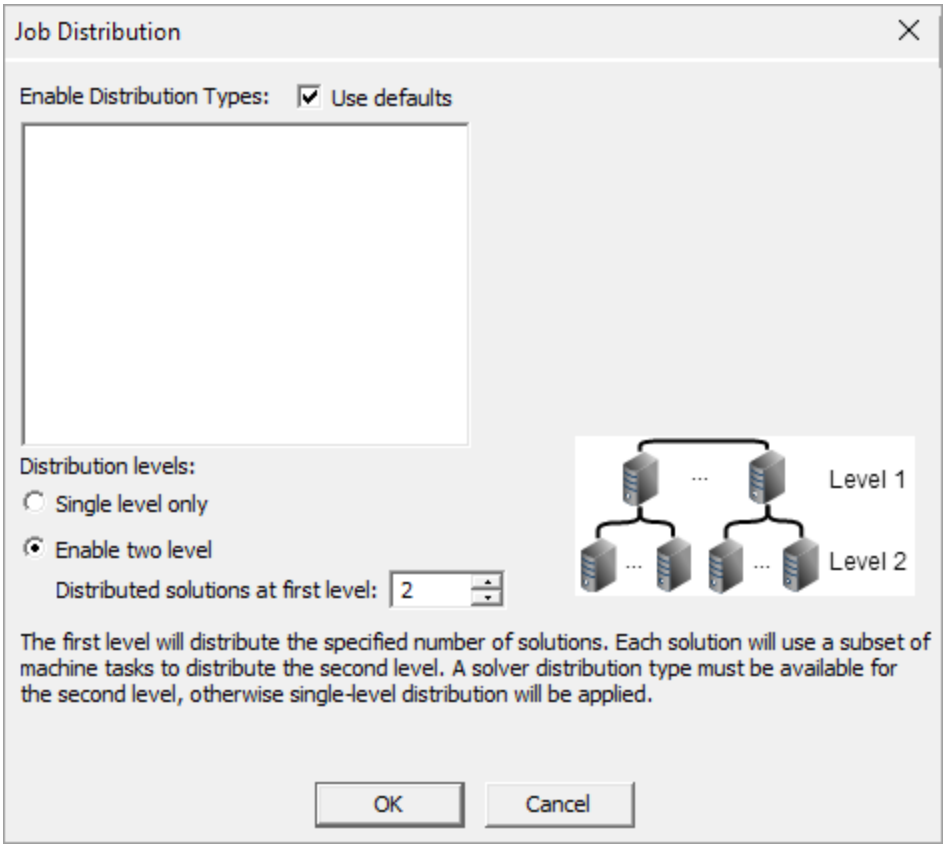
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



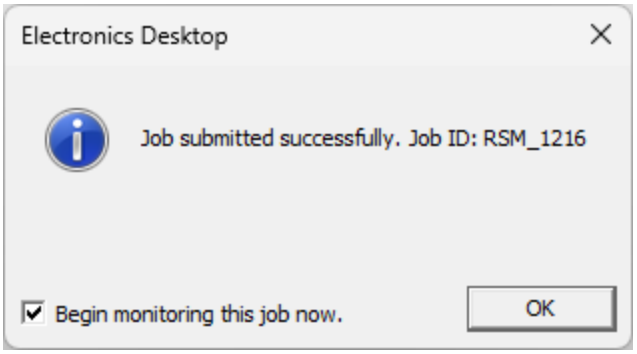
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, 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 2026 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AnsoftRSMService is used to start these remote processes. By default, the AnsoftRSMService listens for socket connections from the Ansys Electromagnetics Desktop on port 32958.

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\v261\EM_RSM" on Windows.

The default location of this directory is /opt/ansys_inc/v261/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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$begin 'AnsoftCOMDaemon'

$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'

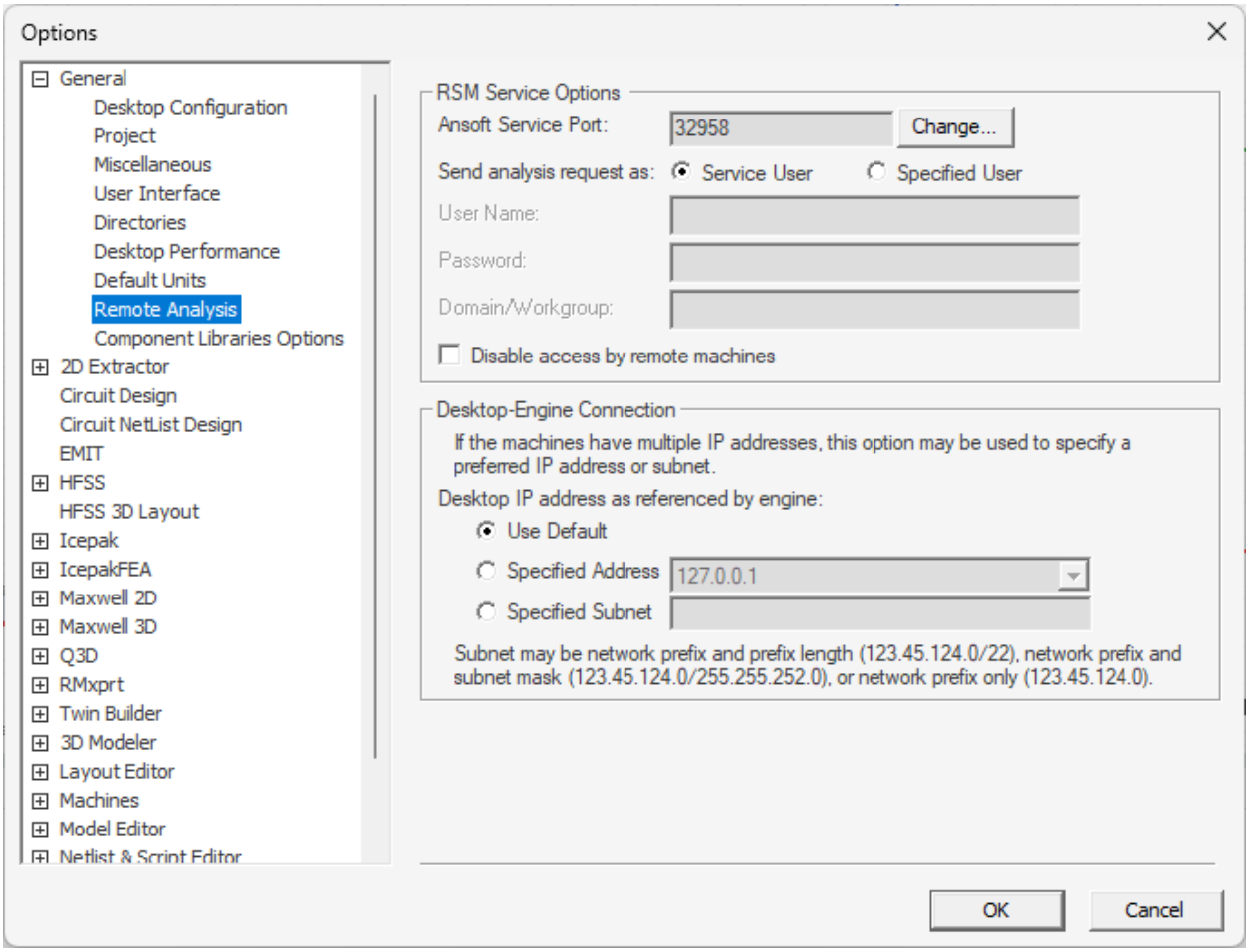
Ansoft Electromagnetics Desktop Configuration

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**.

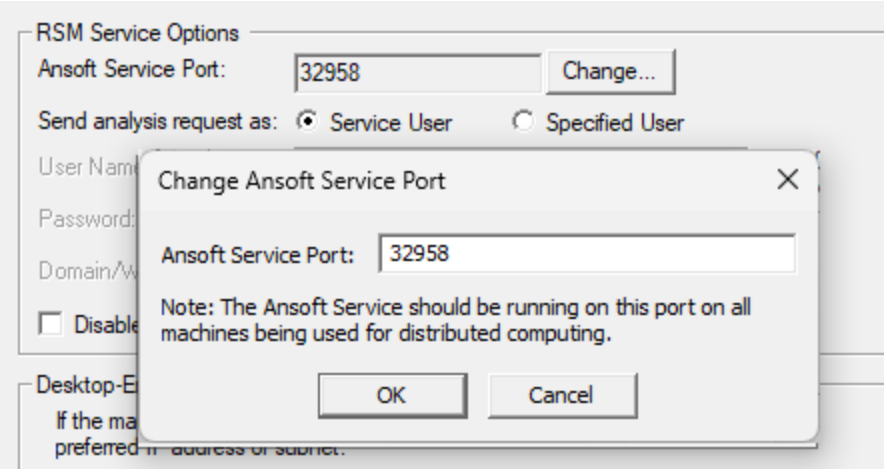
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

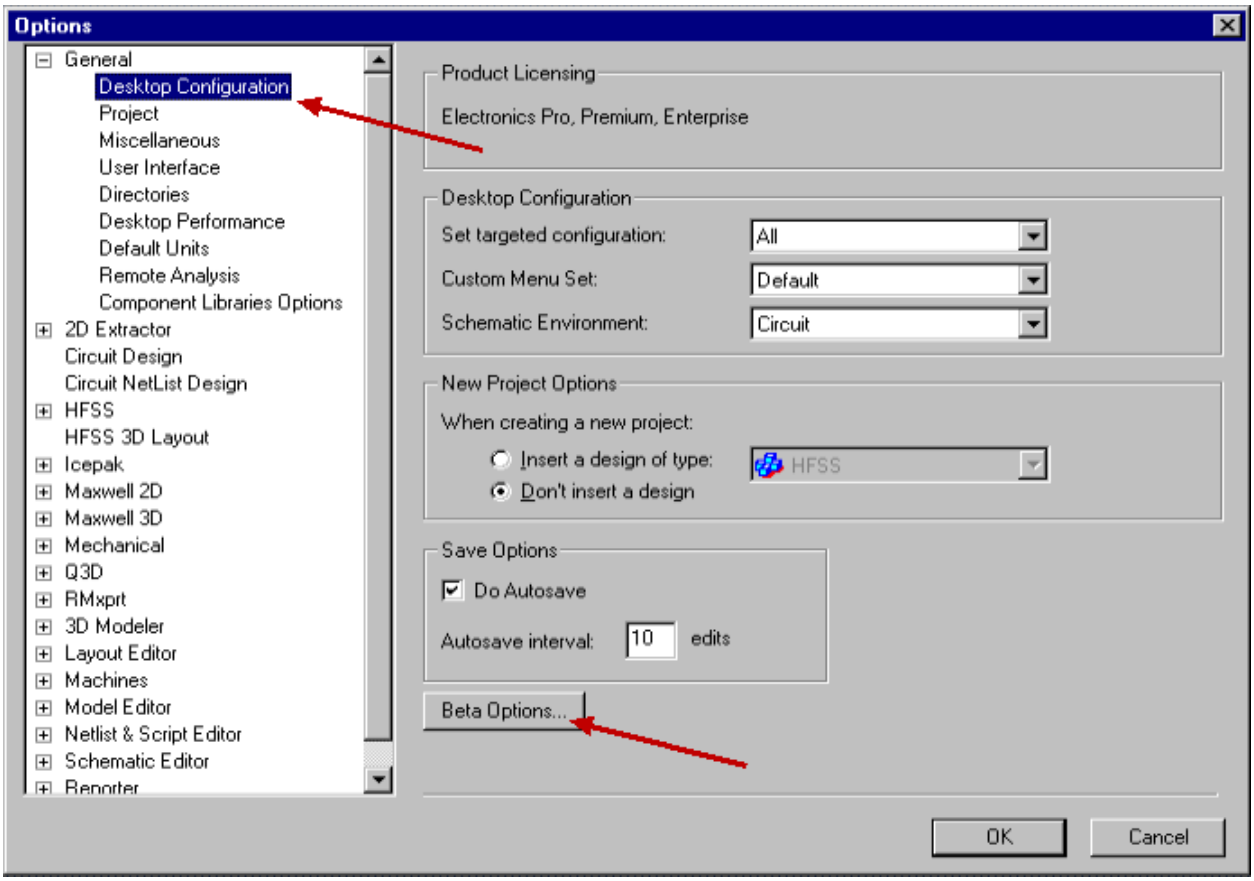
Support for Open MPI on Linux [Beta]

Support for Open MPI on Linux is available as a beta feature for HFSS, HFSS 3D Layout, Icepak, Maxwell (3D and 2D), and Q3D designs.

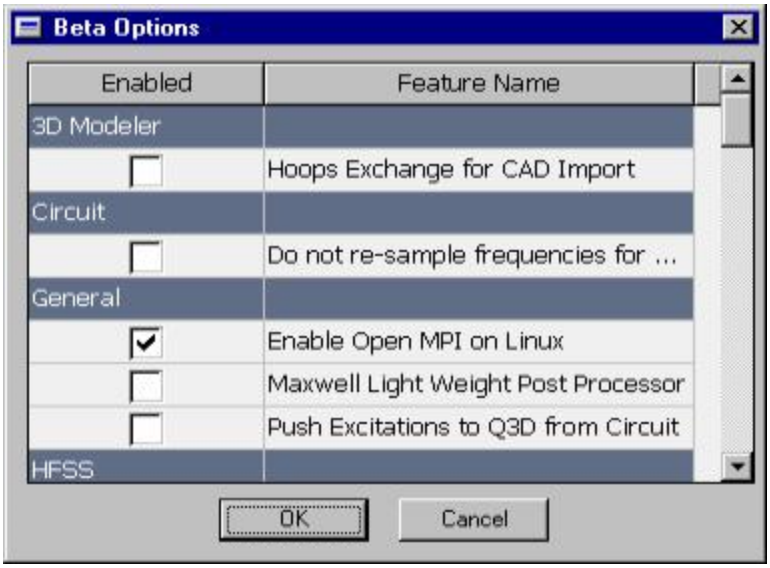
To enable the beta feature:

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Select **Enable Open MPI on Linux**.



5. When making a change to any beta option, you may be prompted to restart the Ansys Electronics Desktop application. If prompted, click **Yes** to do so.

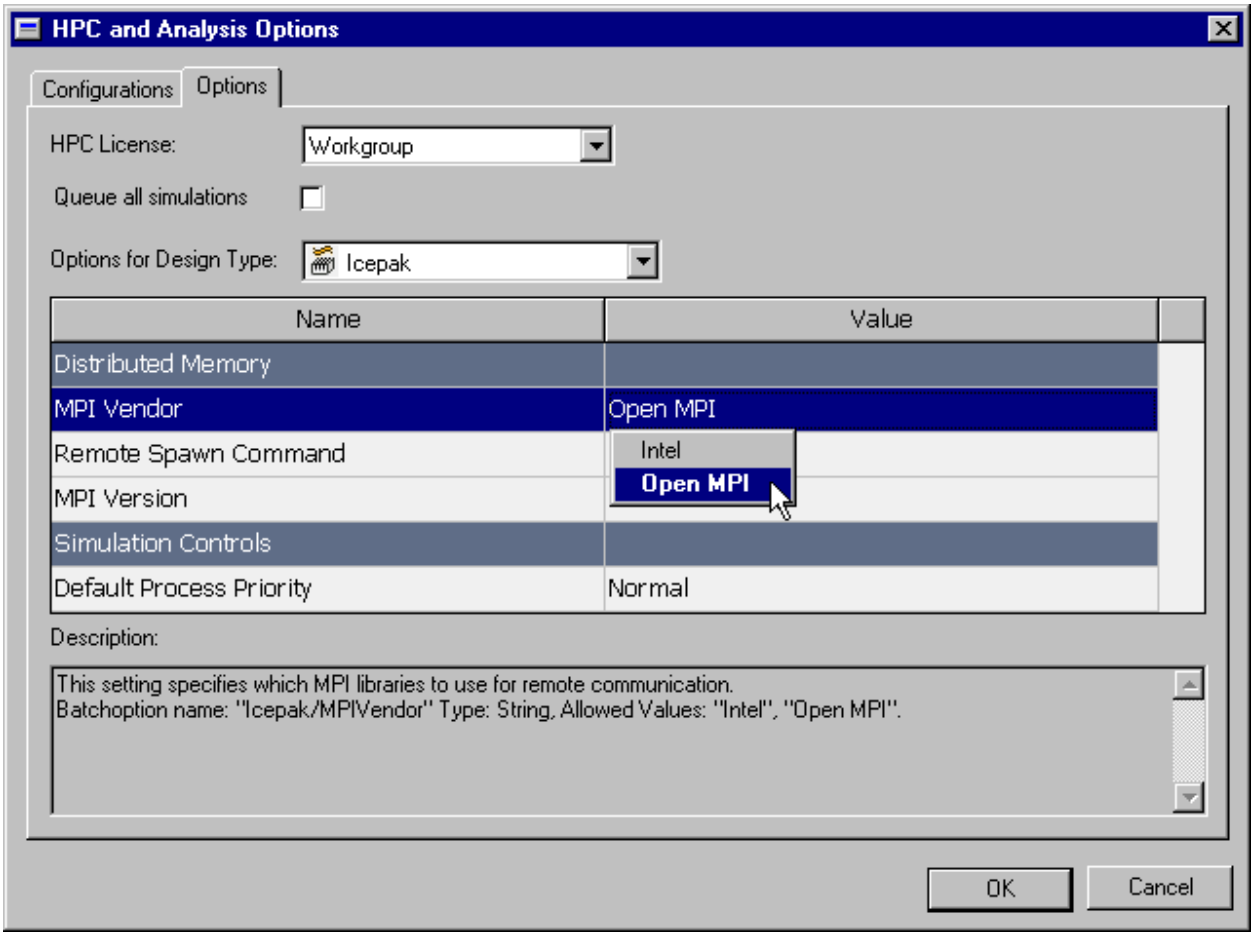
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Open MPI** option will now be available in the **MPI Vendor** drop-down menu within the *Options* tab of the *HPC and Analysis Options* dialog box. You only need to perform this procedure once; the Open MPI option will remain available for future sessions and projects.

To choose Open MPI as the MPI Vendor:

1. Open the project in the Ansys Electronics Desktop application (interactive mode).
2. From the **Simulation** ribbon tab, click  **HPC Options**
3. In the *HPC and Analysis Options* dialog box, select the **Options** tab.

   The contents of this tab vary by design type.
4. Select **Open MPI** from the **MPI Vendor** drop-down menu:



5. Click **OK**.

   You can then run the simulation on a Linux cluster as follows.

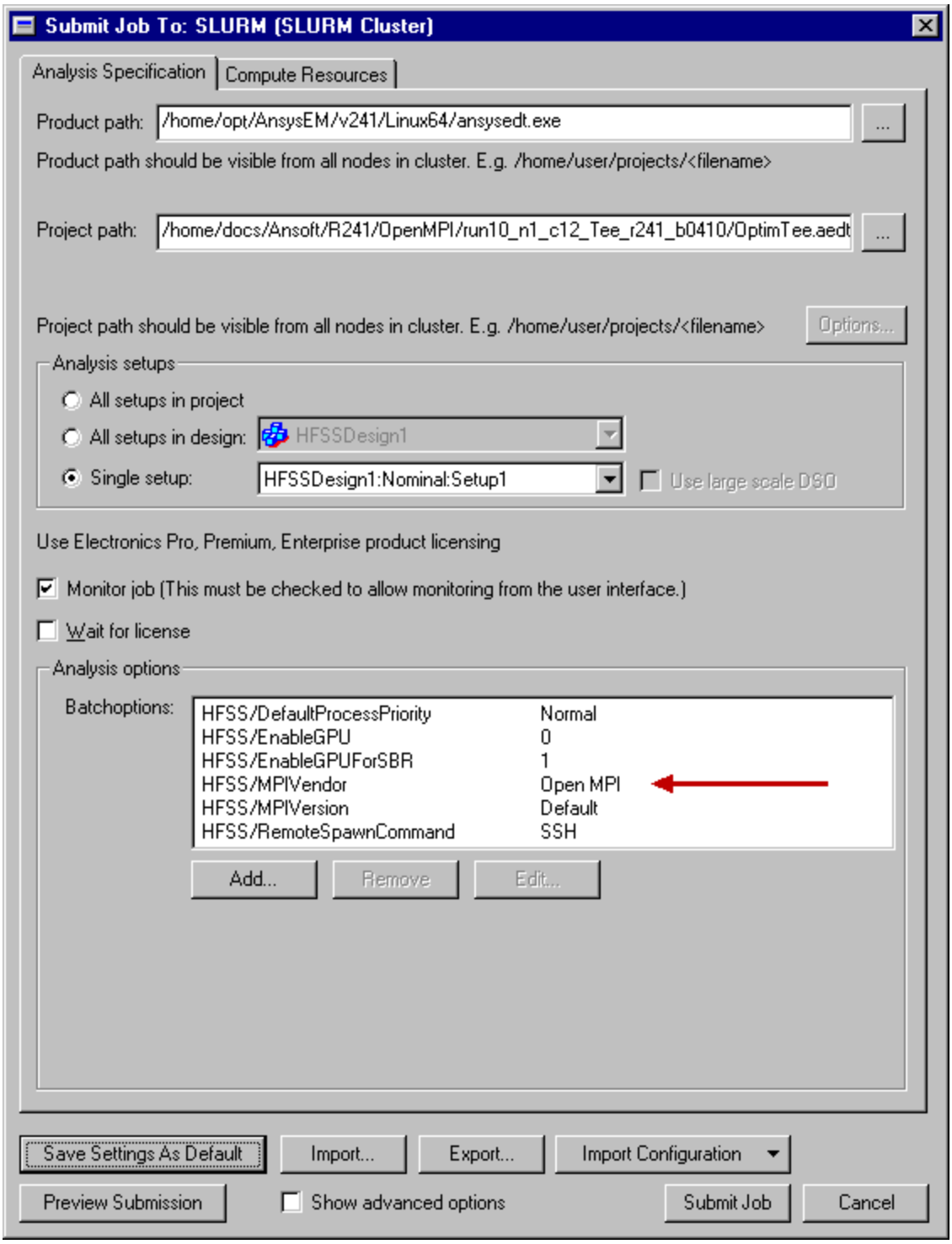
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Project Manager, right-click the solution setup to solve and choose **Submit Job** from the shortcut menu.
7. Specify the desired parameters and click **Submit Job**.

   In the *Submit Job* dialog box, *Open MPI* is included as a batch option for each supported design type along with other applicable batch options.

   The following example is for an HFSS design:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 topic 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

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,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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/v261/AnsysEM/schedulers/diagnostics
```

Windows:

```
 "C:\Program Files\ANSYS Inc\v261\AnsysEM\schedulers\diagnostics
```

Using Diagnostics Scripts on Linux Clusters

The following basic scripts are provided in the diagnostics directory (.../ansys_ inc/v261/AnsysEM/schedulers/diagnostics):

These job submission scripts are scheduler specific.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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\v261\AnsysEM"
Linux example	/shared/ansys_inc/v261/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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment variable	**ANSYSEM_DIAG_RESULTS_DIR**
	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
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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\v261/AnsysEM" on Windows, ".../ansys_inc/v261/AnsysEM" on Linux).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
```

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

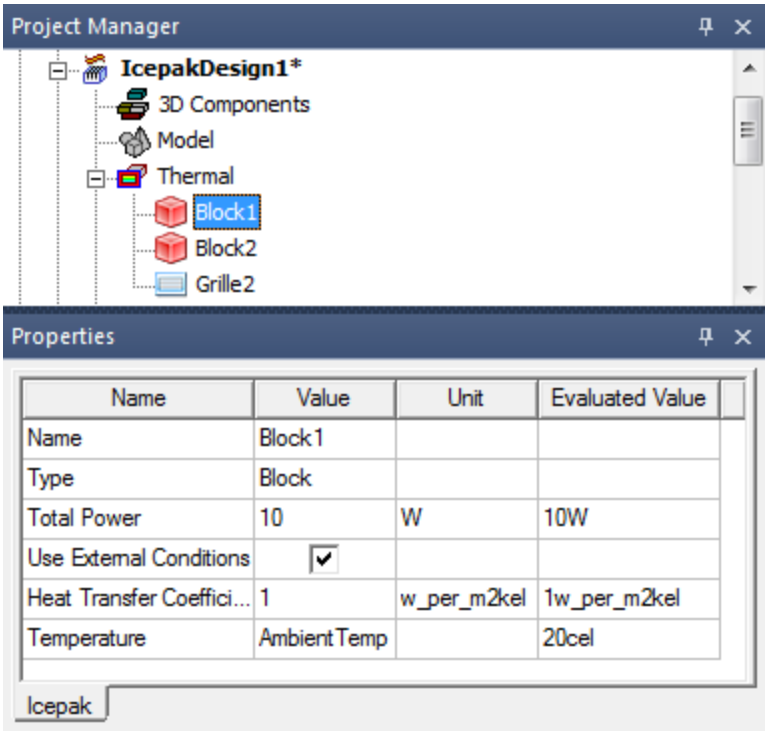
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - Assigning Thermal Boundary Conditions

> **Note:** Some thermal boundary condition parameters can be defined as transient. See Specifying Parameters as a Function of Time for more information.

To specify thermal properties in your model, you can assign Icepak thermal boundary conditions to the geometry you have drawn. When you assign a boundary condition to geometry, the **Properties** pane displays the object's thermal properties. Alternatively, you can either double-click the boundary condition under **Thermal** in the **Project Manager** or right-click it and select **Properties...** to display its properties in a dialog box.

Boundary conditions can be assigned to multiple objects or faces. When a boundary condition is assigned to multiple objects or faces, the properties defined are applied to each of them. For example, if a Block boundary condition with a **Total Power** of 5 W is assigned to multiple box geometries, 5 W is assigned to each box.



This chapter begins with an overview of assigning Icepak thermal boundary conditions and describes their associated thermal properties. Icepak thermal boundary conditions are as follows:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Block
- Grille
- Network
- Opening
- Plate (Adiabatic and Conducting)
- Resistance
- Source
- Wall (Stationary and Symmetry)
- EM Surface and Volume Loss
- Blower
- Heat Exchanger

Temperature-Dependent Thermal Properties

In the Electronics Desktop, you can create datasets, which are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. In Icepak, you can create a dataset to define temperature dependent thermal properties for specific boundary conditions.

The following table displays which thermal boundary condition properties can be defined as a temperature-dependent expression.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

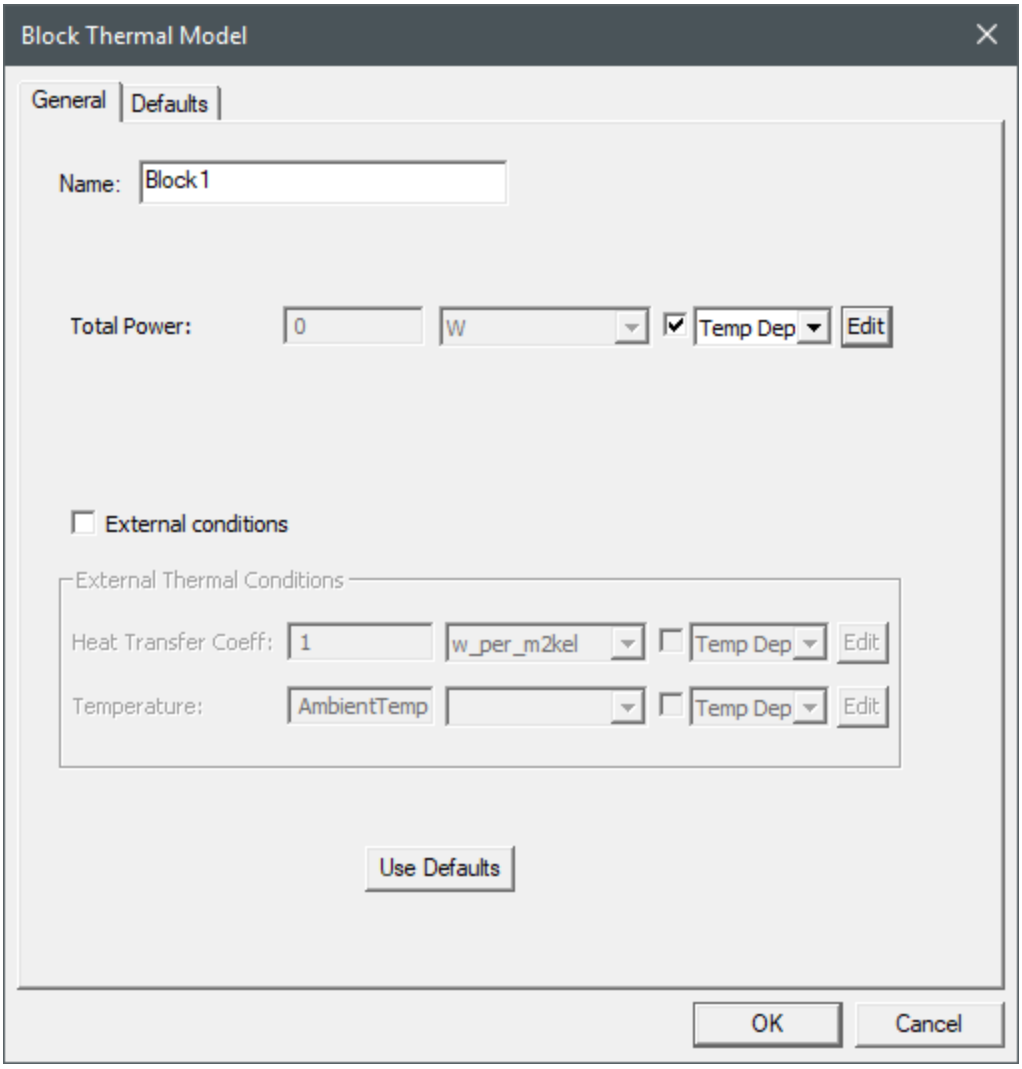
Boundary Condition	Thermal Property
Fluid Block	Total Power Heat Transfer Coefficient
Solid Block	Total Power Heat Transfer Coefficient
Hollow Block	Total Power Heat Flux Heat Transfer Coefficient
Plate (Conducting)	Total Power
2D and 3D Source	Total Power Surface Flux
Wall (Stationary)	Heat Transfer Coefficient
Network	Internal Node Power

Using Datasets to Define Temperature-Dependent Parameters

Some Icepak thermal boundary condition parameters can be assigned a temperature-dependent value using a dataset. For a complete list of temperature-dependent parameters, see Temperature-Dependent Thermal Properties. You can access the **Add Dataset** dialog two ways:

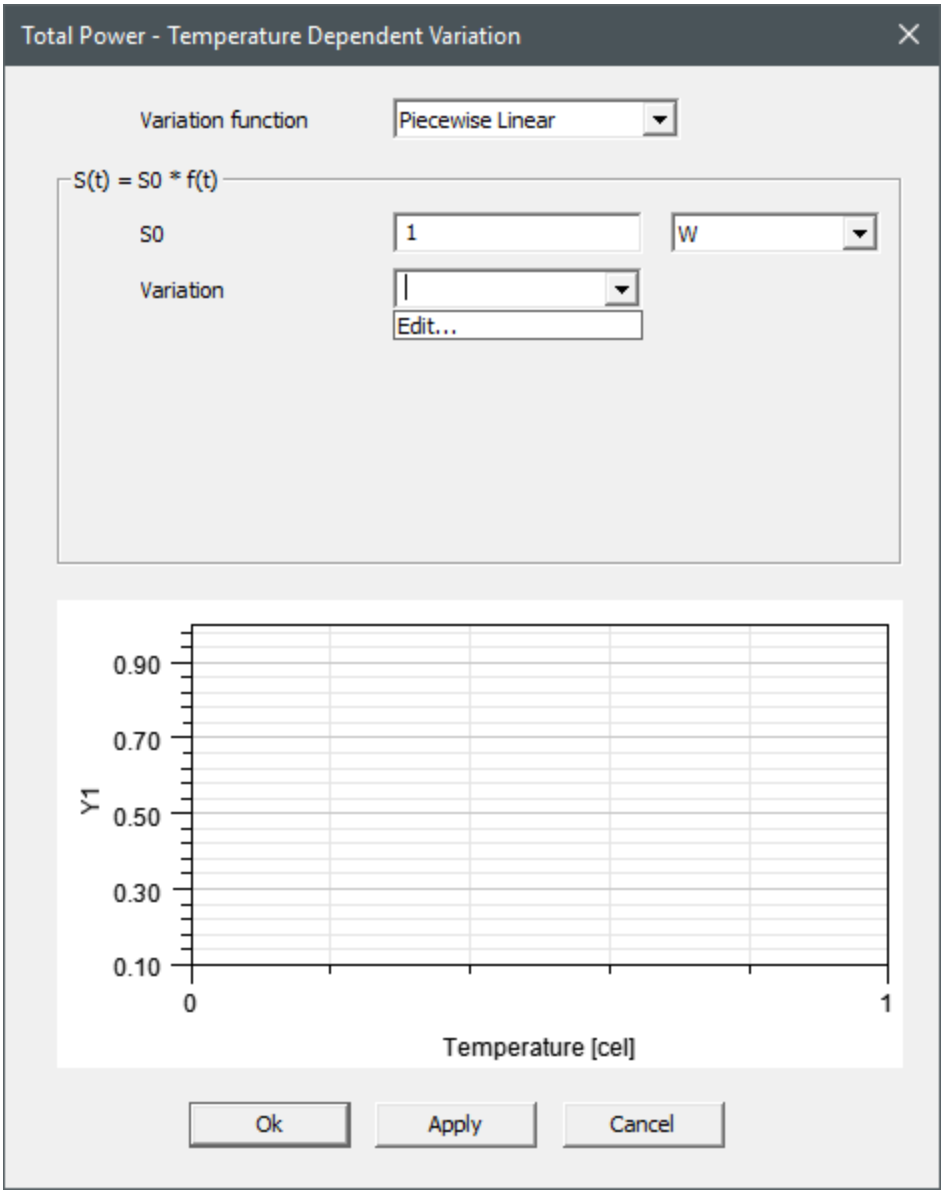
- Define a dataset prior to assigning boundary conditions by adding a project or design dataset. See Adding Datasets for more information.
- When assigning boundary conditions, enable the temperature-dependent option for the parameter and define the dataset. See "Defining a Dataset for Temperature-Dependent Parameters" on page 9-5 for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: Datasets you have previously created are available as options in the **Variation** drop-down list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

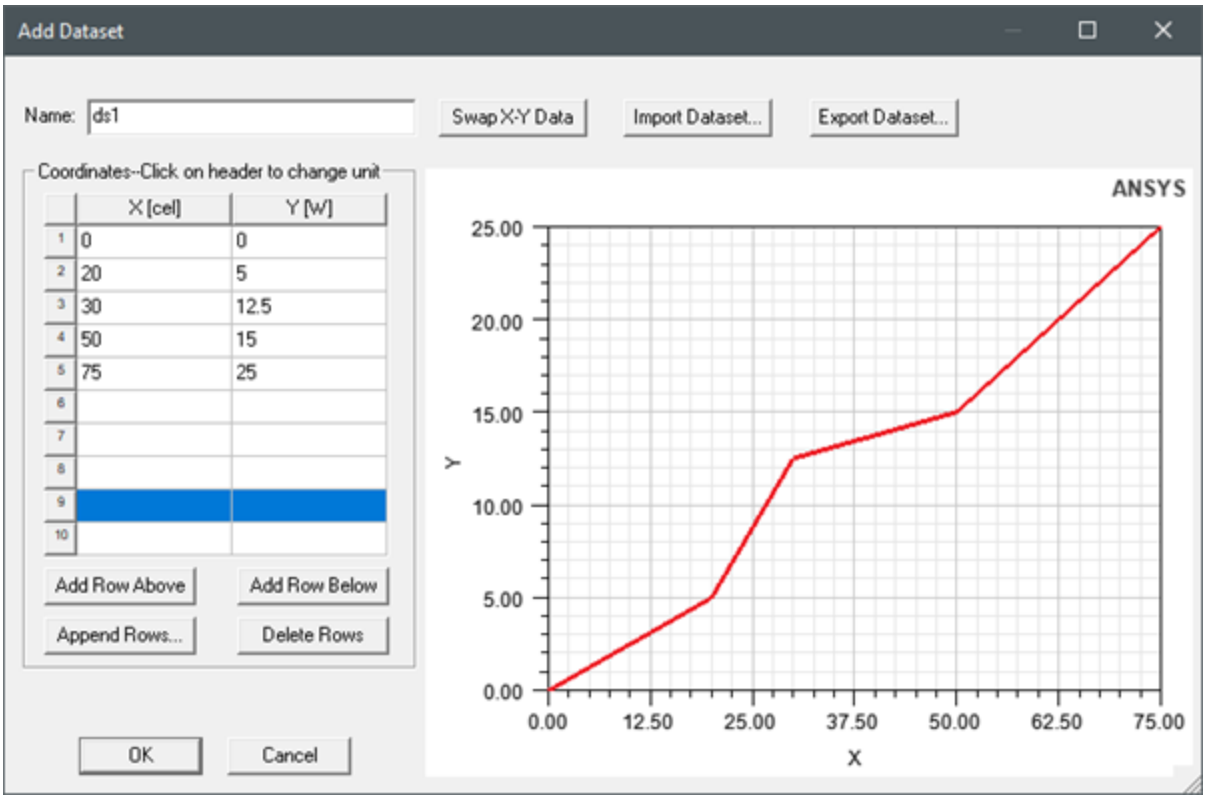


Defining a Dataset for Temperature-Dependent Parameters

1. In the boundary condition or native component dialog box, enable the check box next to the parameter.
2. Select **Temp Dep** from the drop-down list and click **Edit**.
3. In the **Advanced Time Variations** dialog box, select **Piecewise Linear** from the **Time Variation Function** drop-down list.
4. Specify a value for **S0**, which is the total power value used during the simulation when the temperature is not in the range specified in the dataset.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. From the **Variation** drop-down list. select **Edit**.
6. In the **Datasets** dialog box, click **Add** > **1D Dataset**.
7. In the **Add Dataset** dialog box, enter a **Name**.
8. In the **Coordinates** area, click the **X** column header and select **Temperature** and then click it again and select a unit of temperature.
9. Click the **Y** column header and select a parameter category (e.g., **Power**) and then click it again and select a unit for the parameters (e.g., **W**).
10. In the **X** and **Y** columns enter the values for time and the time-dependent parameter.



11. Click **OK** to save the dataset.
12. In the **Datasets** dialog box, click **Done**.

3D Datasets for Thermal Properties

In Icepak designs, you can create and assign a 3D dataset, which is a spatial, or volumetric, profile, to certain thermal boundary condition properties.

The following table displays which thermal boundary condition properties can be defined with a 3D dataset.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Condition	Thermal Property
Block	Total Power
	Volume Flux
Hollow Block	Total Power
Resistance	Total Power
Source	Surface Flux
Wall (Stationary)	Temperature
	Heat Transfer Coefficient
	Heat Flux

To define a thermal property with a 3D dataset, enter the following expression :

`clp(datasetName, X,Y,Z)`

where `datasetName` is the name given to the 3D dataset.

Joule Heating in Icepak

In the Electronics Desktop, you can model block and source boundary conditions using joule heating.

Joule Heating in Blocks

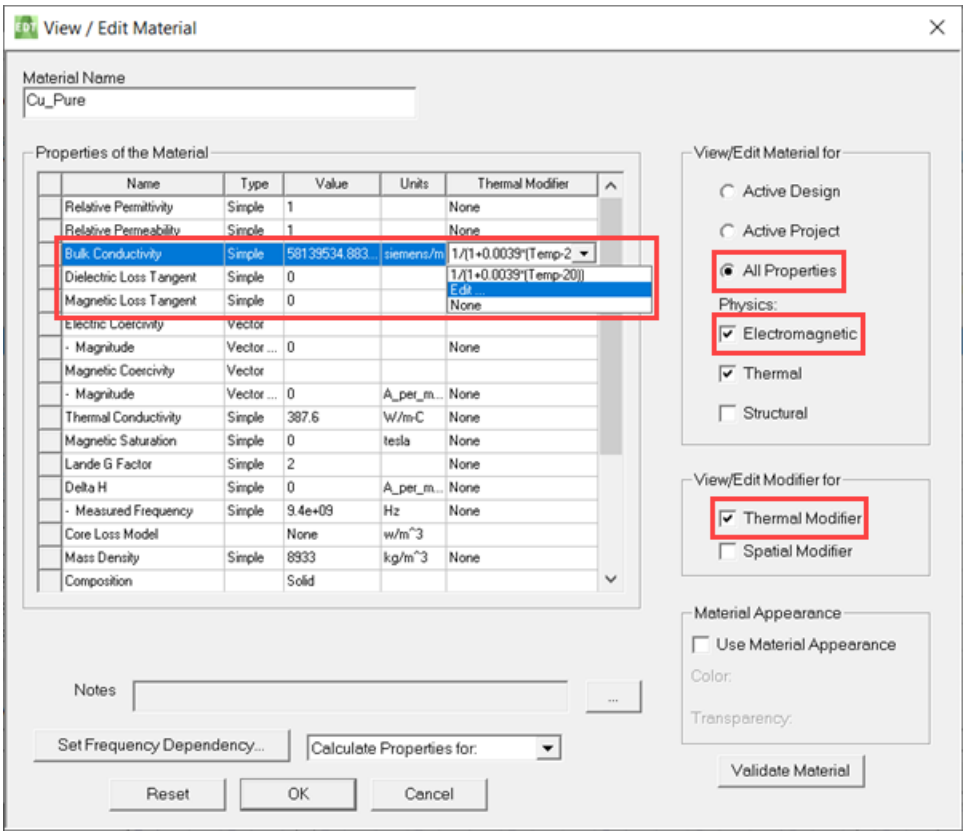
Blocks with solid materials can be assigned joule heating for **Total Power** in the **Block Thermal Model** dialog box.



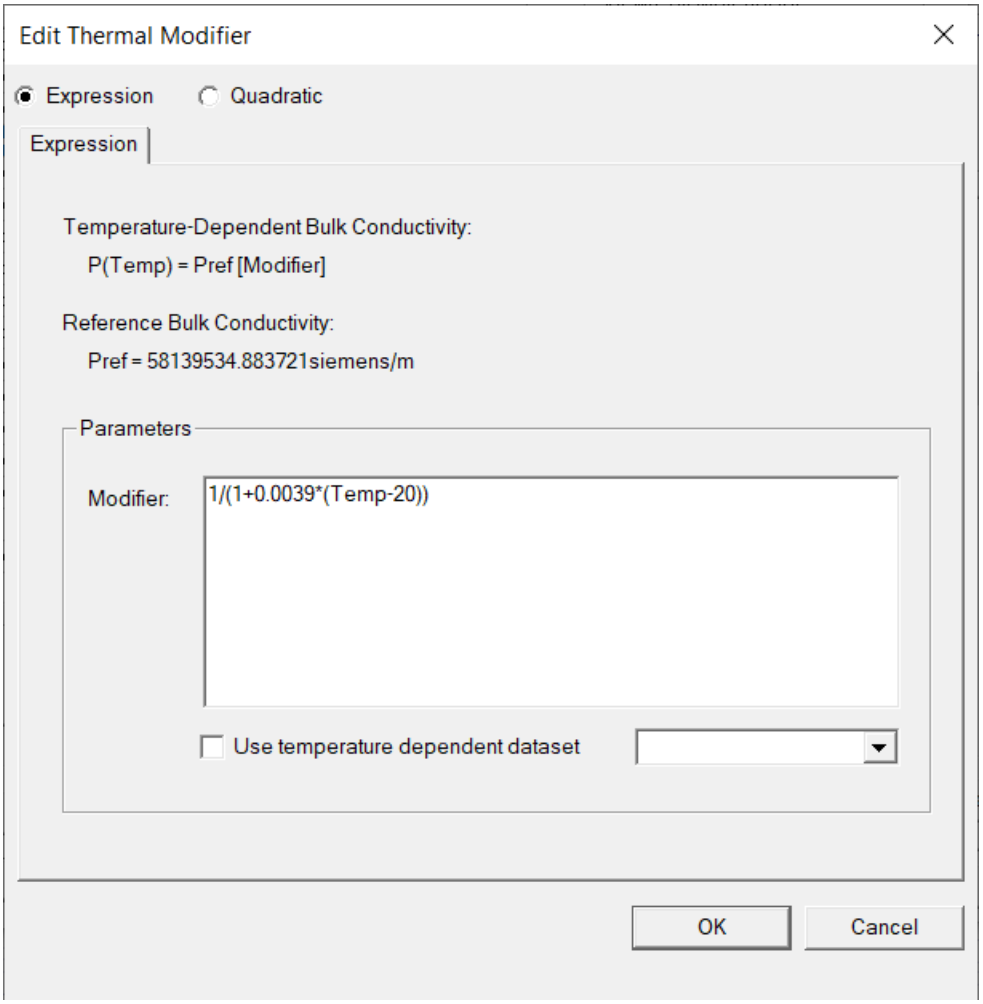
When assigned joule heating, blocks use the bulk conductivity of the assigned material to calculate the total power. The total power is computed by solving an additional conservation equation for the electric potential at every iteration. This equation is only solved in the current-containing objects. The joule heating power is accurately computed per cell in each object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **Select Definition** dialog box, click **View/Edit Material**. To display **Bulk Conductivity**, select **All Properties** and **Electromagnetic** under **View/Edit Material for**. If no thermal modifier is applied to bulk conductivity, it is assigned as a constant value. To assign bulk conductivity as a varying value, edit the material and apply a thermal modifier.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the example above, the thermal modifier equation is as follows:

$$1/(1+C(\mathrm{Temp}-\mathrm{Tref}))$$

where C is the constant, Temp is the intrinsic variable for the analysis temperature, and Tref is the reference temperature.

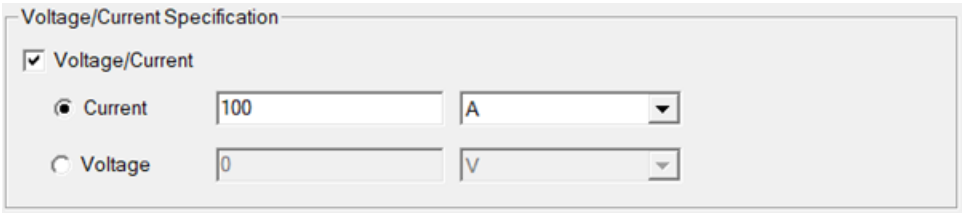
> **Note:**
>
> Quadratic definitions are not currently supported.

Joule Heating in Sources

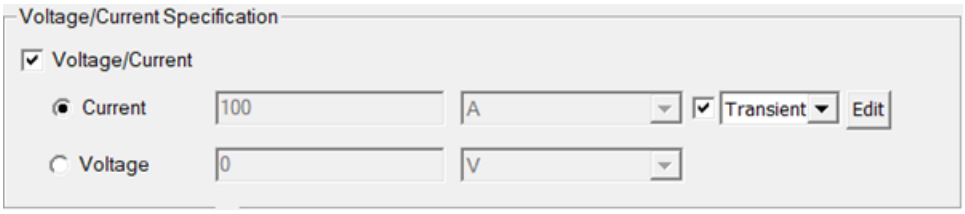
Sources can be assigned joule heating properties under **Voltage/Current Specification** in the **Source Thermal Model** dialog box. Select **Current** or **Voltage**. Enter a value and select the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

corresponding unit.



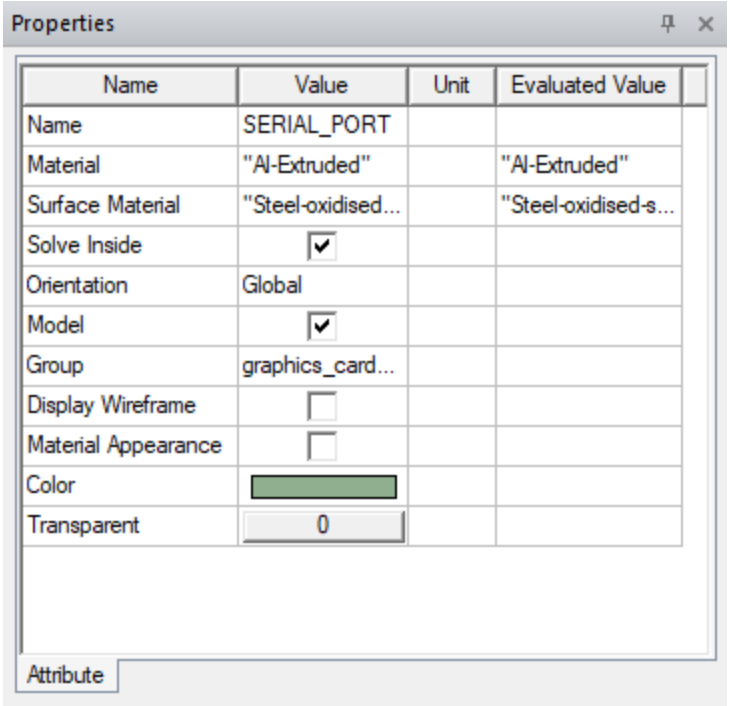
Time dependent variations are supported for **Current** and **Voltage**. See Specifying Parameters as a Function of Time for more information.



Assigning Block Properties

A block is a three-dimensional modeling boundary condition. To assign a block boundary condition, the selected geometry must be three-dimensional (for example, a box). There are three block types (solid, hollow, and fluid). Block type is determined by the **Material** and **Solve Inside** properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

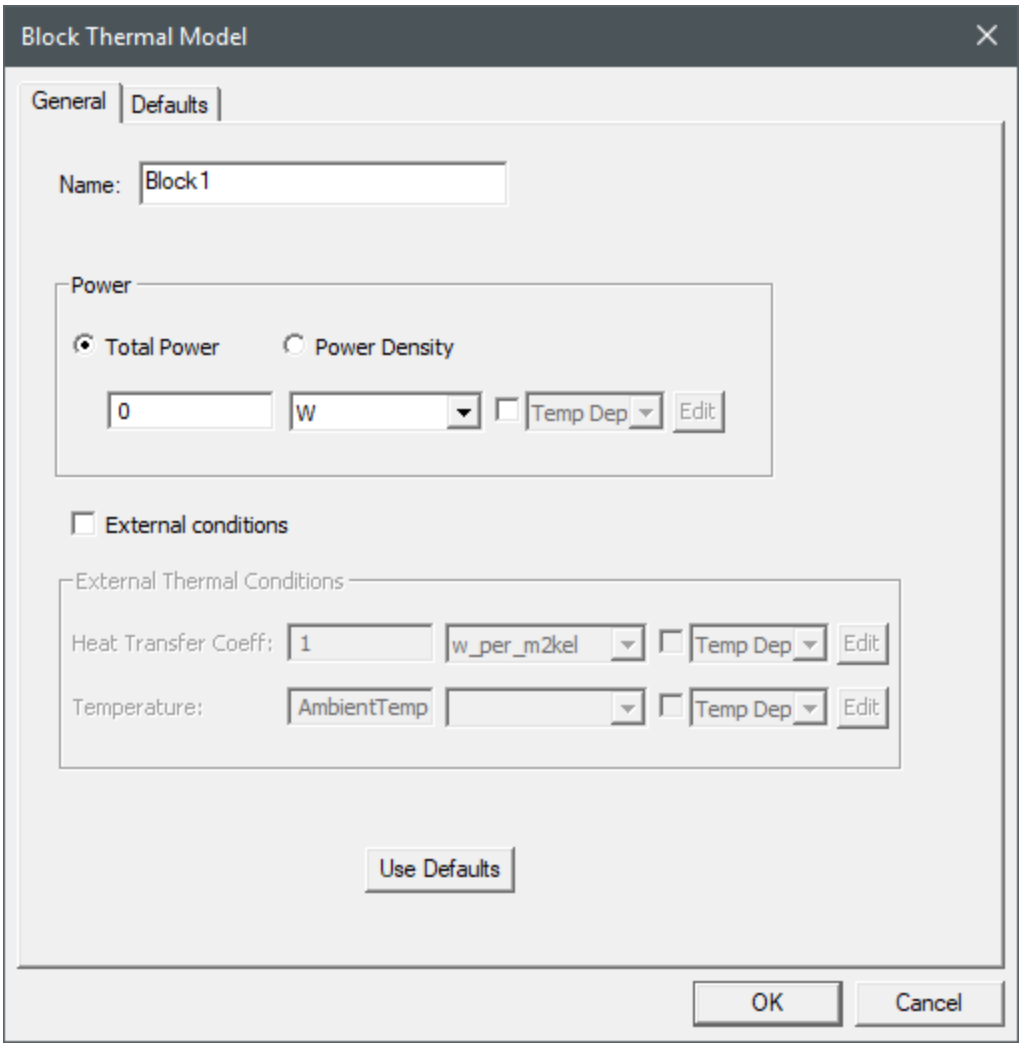


- **Solid**: A solid **Material** is assigned and the **Solve Inside** option is enabled.
- **Hollow**: A solid **Material** is assigned and the **Solve Inside** option is disabled.
- **Fluid**: A fluid **Material** is assigned. **Solve Inside** can be enabled or disabled.

Assigning Solid Block Properties

A solid block is a block with a solid material assigned and the **Solve Inside** option enabled. Solid blocks represent actual solid objects and can possess physical and thermal characteristics such as density, specific heat, thermal conductivity, and total heat flux. Icepak considers the interior of a solid block to be part of the computational domain and includes the block internal temperature distribution as part of the model solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Block Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Note:

- If needed, you can enter a temperature dependent expression for **Total Power**, **Heat Transfer Coefficient**, or **Temperature**. See Temperature Dependent Thermal Properties for more information.
- If needed, you can define **Total Power** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Power	The **Power** specifies type of power assigned to the block. **Total Power** specifies the total power dissipated by the solid block. For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For **Temp Dep**, click **Edit** to specify the temperature dependence of power. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. Alternatively, specify the power as **Power Density**. **Transient** allows you to specify power as a function of time. **DTM** (dynamic thermal modeling) allows you to specify a temperature-dependent power after each time step of a transient simulation.
Note: For more information on specifying transient power, see Specifying Parameters as a Function of Time. For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.	
External Thermal Conditions	**External Thermal Conditions** specifies a **Heat Transfer Coefficient** and a reference **Temperature** for the volume of space on the immediate exterior of the currently selected side of the solid block.

Assigning a Solid Block and Defining Thermal Properties

To assign a solid block boundary condition and define its properties:

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.

   > **Note:**
   >
   > To assign a solid block type, ensure that the material assigned is solid and the **Solve Inside** option is enabled.

3. From the right-click menu, select **Assign Thermal**>**Block**.
4. In the **Block Thermal Model** dialog box, enter a **Name** for the block.
5. Under **Power**, do one of the following:
   - For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. For **Temp Dep**, click **Edit** to open the **Temperature Dependent Variation** dialog box and specify the **Variation function** and values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> **Joule Heating** uses the block geometry's **Thermal Conductivity** to calculate the block's power. See Joule Heating in Icepak for more information.

- Select **Power Density** and enter a power density value for the block.

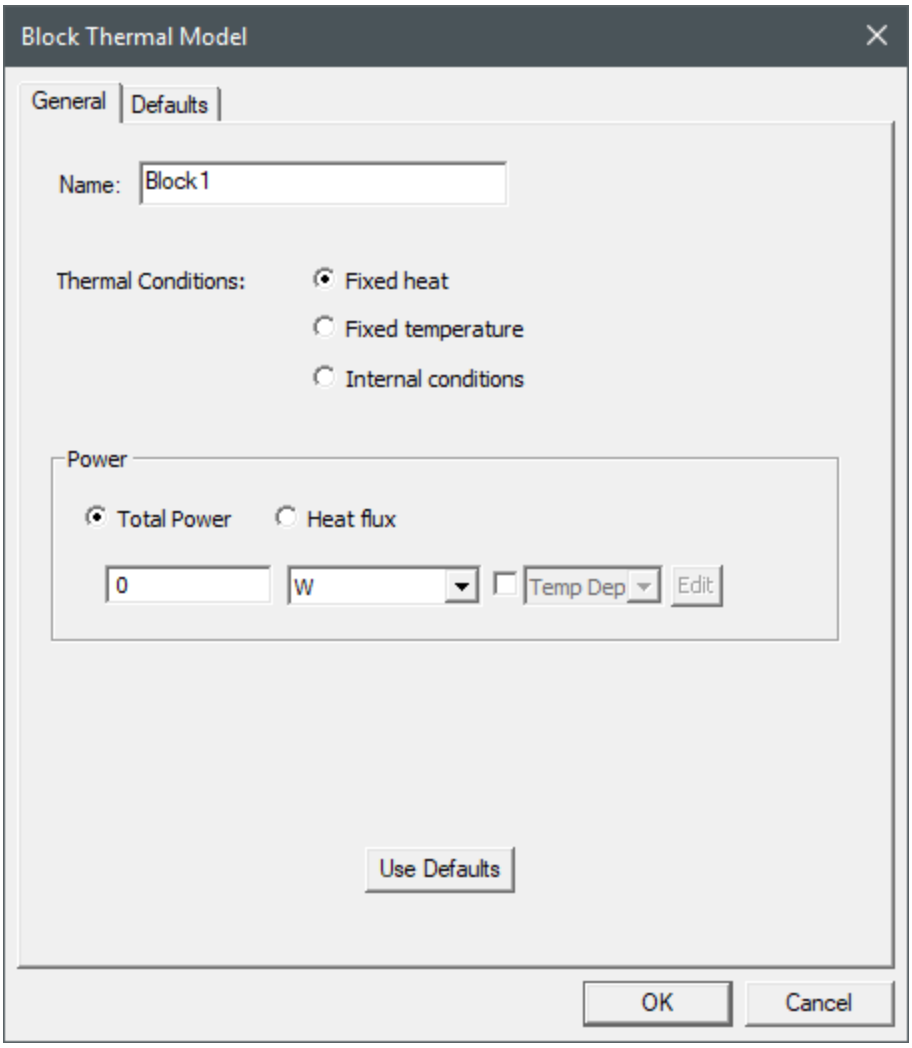
6. If needed, select the **External conditions** check box and enter the following values:
   - **Heat Transfer Coeff**
   - **Temperature**
7. Click **OK**.

> **Note:**
>
> The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:
>
> - **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
> - **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Hollow Block Properties

A hollow block is a block with a solid material assigned and the **Solve Inside** option disabled. Hollow blocks represent three-dimensional regions of the model for which only side characteristics are important. Icepak does not mesh or solve for temperature or flow within regions bounded by the sides of a hollow block. Hollow-block surfaces can be specified as adiabatic (impervious to heat flow) or as possessing a fixed, uniform temperature or heat flux.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Block Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Note:

- If needed, you can enter a temperature-dependent expression for **Total Power**, **Heat Flux**, and **Heat Transfer Coefficient**. See Temperature Dependent Thermal Properties for more information.
- If needed, you can define **Total Power** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Thermal Conditions: Fixed heat	
Total Power	**Total Power** specifies the total power dissipated by the solid block. For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For **Temp Dep**, click **Edit** to specify the temperature dependence of power. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. Alternatively, specify the power as **Power Density**. **Transient** allows you to specify power as a function of time. DTM (dynamic thermal modeling) allows you to specify a temperature-dependent power after each time step of a transient simulation.
Note: For more information on specifying transient power, see Specifying Parameters as a Function of Time. For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.	
Heat flux	**Heat flux** specifies a constant uniform heat flux into or out of all block surfaces.
Thermal Conditions: Fixed temperature	
Fixed Temperature	**Fixed Temperature** specifies a constant uniform temperature for all the block surfaces.
Thermal Conditions: Internal conditions	
Internal Thermal Conditions	**Internal Thermal Conditions** specifies a **Heat Transfer Coefficient** and a reference **Temperature** for the internal volume of space inside the block.

Assigning a Hollow Block and Defining Thermal Properties

To assign a hollow block boundary condition and define its properties:

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.

> **Note:**
>
> To assign a hollow block type, ensure that the material assigned is solid and the **Solve Inside** option is disabled.

3. From the right-click menu, select **Assign Thermal**>**Block**.
4. In the **Block Thermal Model** dialog box, enter a **Name** for the block.
5. Select a **Thermal Conditions** and enter the associated values:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fixed heat**: **Total Power** or **Heat flux**. For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. For **Temp Dep**, click **Edit** to open the **Temperature Dependent Variation** dialog box and specify the **Variation function** and values.
- **Fixed temperature**: **Fixed Temperature**.
- **Internal conditions**: **Heat Transfer Coefficient** and a reference **Temperature**.

6. Click **OK**.

Note:

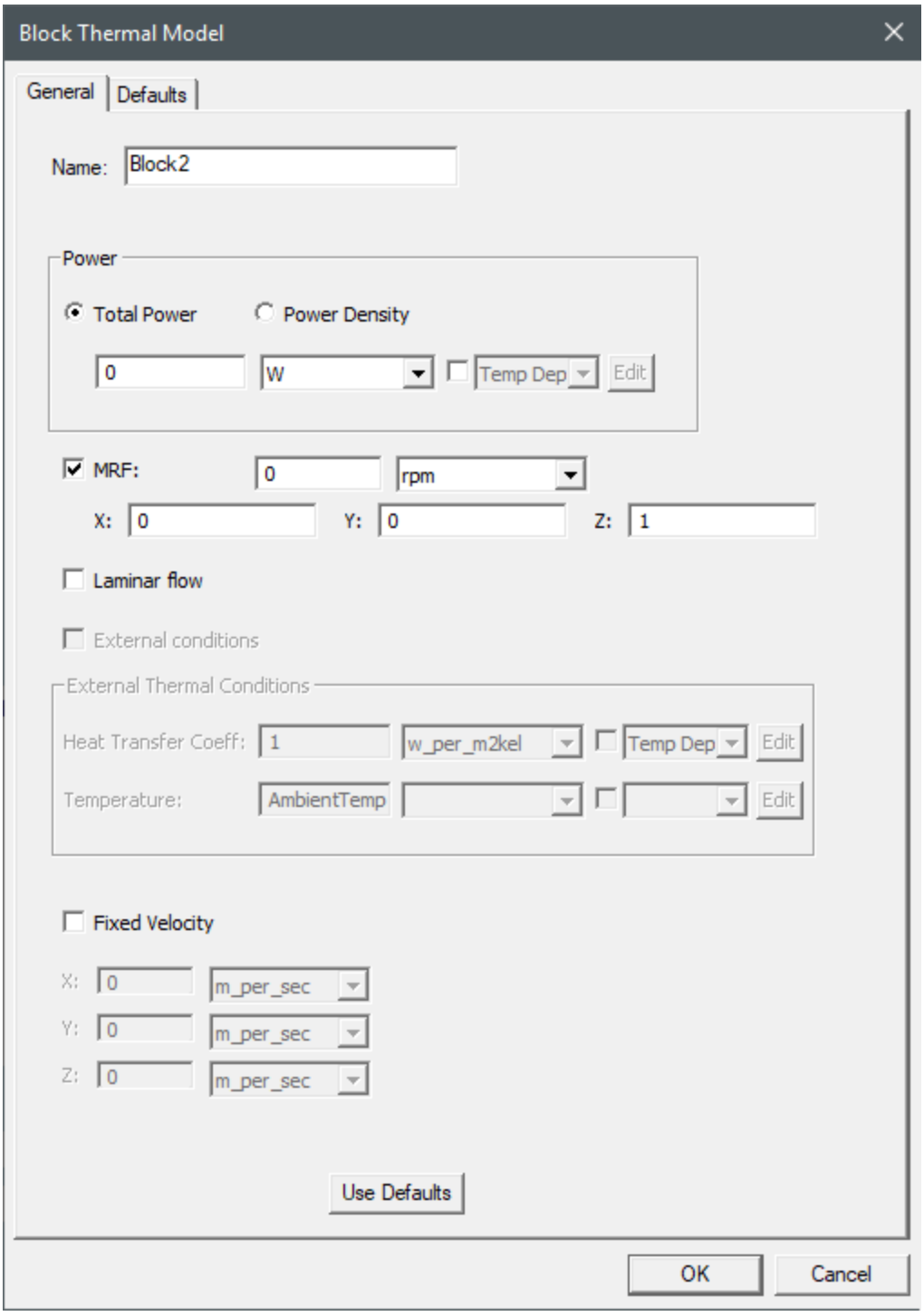
The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Fluid Block Properties

A fluid block is a block with a fluid material assigned. Fluid blocks are regions of the model where fluid properties can be specified independently of those specified for the default fluid.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Block Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- If needed, you can enter a temperature dependent expression for **Total Power**, **Heat Transfer Coefficient**, or **Temperature**. See Temperature Dependent Thermal Properties for more information.
- If needed, you can define **Total Power** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Power	The **Power** specifies type of power assigned to the block. **Total Power** specifies the total power dissipated by the solid block. For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For **Temp Dep**, click **Edit** to specify the temperature dependence of power. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. Alternatively, specify the power as **Power Density**. **Transient** allows you to specify power as a function of time. DTM (dynamic thermal modeling) allows you to specify a temperature-dependent power after each time step of a transient simulation. Alternatively, specify the power as **Power Density** (**Solve Inside** enabled) or **Heat Flux** (**Solve inside** disabled).
Note: For more information on specifying transient power, see Specifying Parameters as a Function of Time. For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.	
MRF	Moving Reference Frame **MRF** is available when a fluid material (gaseous fluids only) is assigned to the block's geometry. MRF specifies an angular velocity to model rotating parts in a steady state simulation. See Moving Reference Frame for more information.
Laminar flow	**Laminar flow** specifies that interior of the block is to be modeled as a laminar zone.
External Thermal Conditions	**External Thermal Conditions** specifies a **Heat Transfer Coefficient** and a reference **Temperature** for the volume of space on the immediate exterior of the currently selected side of the solid block.
Fixed Velocity	**Fixed Velocity** allows you to fix values of velocities to essentially set a boundary condition for the velocities within the cells of the fluid block. When a velocity is fixed in a given cell, the transport equation for that velocity is not solved in the cell. The fixed value is used for the calculation of face fluxes between the cell and its neighbors. The result is a smooth transition between the fixed value of a velocity and the values at the neighboring cells.

Assigning a Fluid Block and Defining Thermal Properties

To assign a fluid block boundary condition and define its properties:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.

   > **Note:**
   >
   > To assign a fluid block type, ensure that the material assigned is fluid.

3. From the right-click menu, select **Assign Thermal**>**Block**.
4. In the **Block Thermal Model** dialog box, enter a **Name** for the block.
5. Under **Power**, do one of the following:

   - For steady-state simulations, if needed, select the check box and select **Temp Dep** or **Joule Heating**. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. For **Temp Dep**, click **Edit** to open the **Temperature Dependent Variation** dialog box and specify the **Variation function** and values.

     > **Note:**
     >
     > **Joule Heating** uses the block geometry's **Thermal Conductivity** to calculate the block's power. See Joule Heating in Icepak for more information.

   - If **Solve Inside** is enabled, select **Power Density** and enter a power density value for the block.
   - If **Solve Inside** is disabled, select **Heat Flux** and enter a heat flux value for the block.

6. If needed, select the **Laminar flow** check box.
7. If needed, select the **External conditions** check box and enter the following values:
   - **Heat Transfer Coeff**
   - **Temperature**
8. If needed, select the **Fixed Velocity** check box and enter velocity values for each coordinate direction.
9. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Moving Reference Frame (MRF)

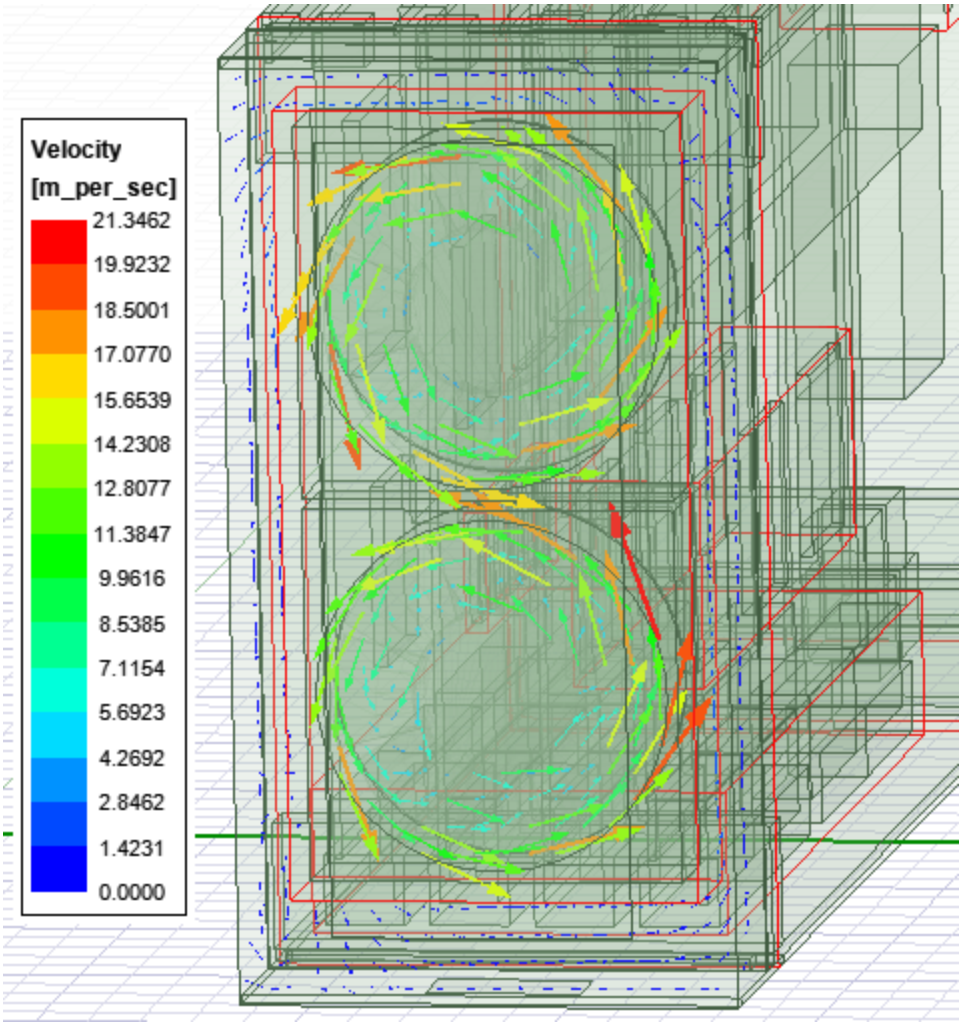
Moving Reference Frame (MRF) allows users to easily model steady state problems with rotating parts. It is computationally inexpensive and accurate enough for most industrial problems.

An Icepak 3D component fan may sometimes behave differently from real fan behavior. Typically this is due to flow features like swirl and centrifugal spread resultant from the spinning fan rotor as well as the rotor blade geometry. In an Icepak fan-based setup, this swirl is estimated and calculated off a 2D annular flow face rather than off the physical surfaces of the blade.

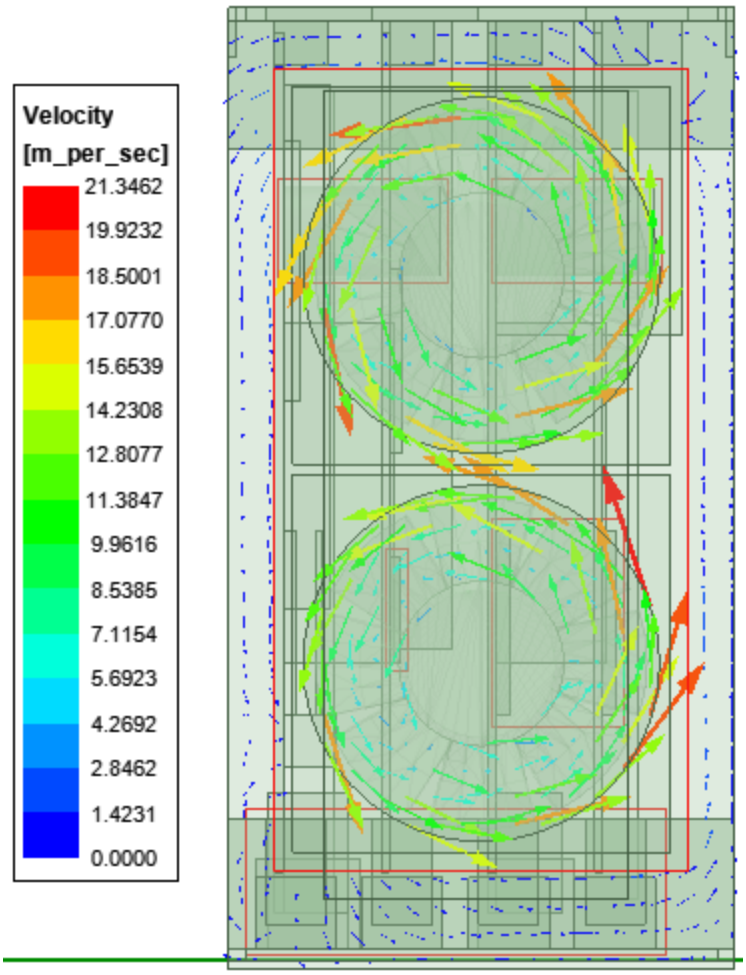
The MRF approach allows for a more accurate representation of the internal geometry of the fan and therefore results in a better modeling of the flow features. In Icepak MRF simulations, a fluid block rotates into contact with surfaces to create a spinning flow.

Below are images that illustrate spinning flow using a cut plane with speed vectors.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



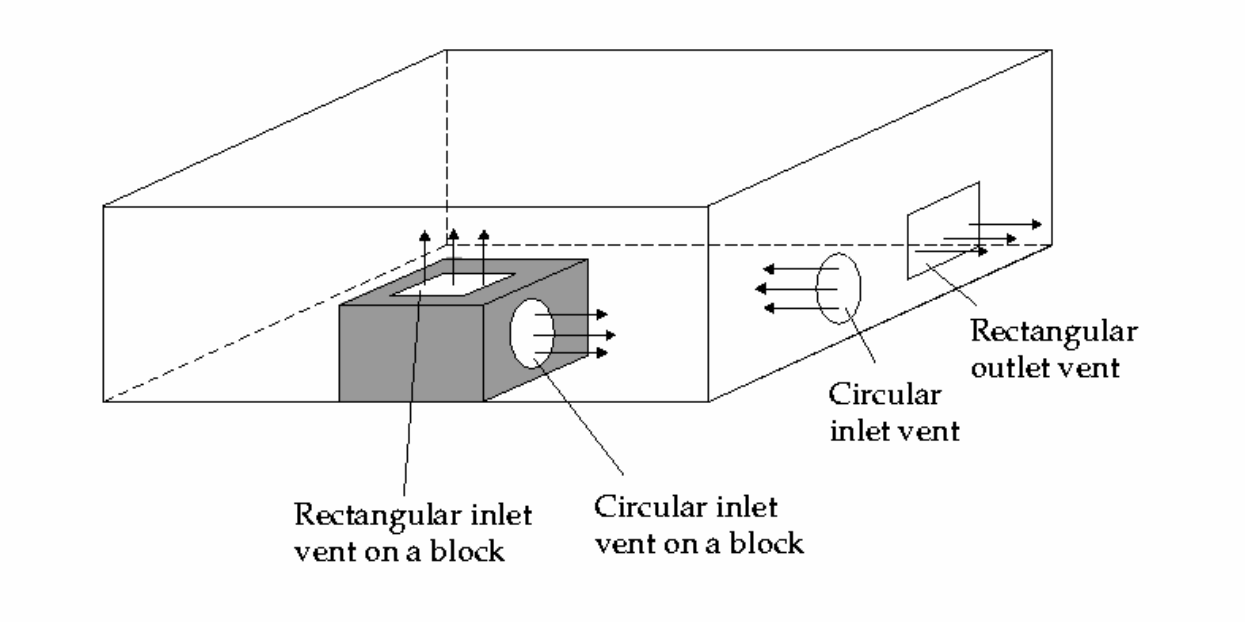
Assigning Grille Properties

A grille is a two-dimensional modeling boundary condition that can represent either vents or planar resistances depending on the location of the boundary assignment. A grille is treated as a vent if the boundary is assigned on a mesh region surface, whereas it is treated as a planar resistance if the boundary is assigned inside the mesh region. The effects of a grille is modeled as a pressure drop across its surface.

Modeling Vents in Icepak

Vents represent holes through which fluid can enter or leave the mesh region. They are always located on enclosure boundaries (either mesh region or hollow block surfaces).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In most real-world cases, vents have coverings (for example, screen meshes, angled slats, wire grilles). These coverings result in a pressure drop across the plane of the vent. Icepak treats the pressure drop in the same way that it treats a pressure drop through a resistance modeling boundary condition.

Fluid can enter or exit the mesh region through a vent. The actual flow direction through the vent is computed by Icepak. In some cases, fluid can enter and exit through different areas of a single vent. In cases where the amount of flow in both directions is nearly equal, the vent may be modeled as two separate but adjoining vents.

Icepak computes both the temperature and the direction of the fluid exiting the mesh region through a vent. By default, the temperature of the fluid entering the mesh region through a vent is assumed to be the **Temperature** specified in the **Icepak Design Settings** dialog box. For best results, the vent size and geometry should closely match the size and geometry of the actual vent.

Modeling Planar Resistances in Icepak

Planar resistances represent partial planar flow obstructions within the mesh region, such as screens and permeable baffles. For a planar resistance in Icepak, the effect of any resistance is modeled as a pressure drop through its area or volume.

Pressure Drop Calculations for Grilles

The pressure drop across grilles is determined based on the selected pressure loss type - either loss coefficient or loss curve.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loss Coefficient

Icepak calculates the loss coefficients based on the free area ratio of the grill and the type of grille. For the given grille type, the loss coefficient is computed as follow, where A is the free area ratio:

- Perforated thin vent, with a loss coefficient of

$$l_c = \frac{1}{A^2}\left[0.707(1-A)^{0.375} + 1 - A\right]^2$$

- Circular metal wire screen, with a loss coefficient of

$$l_c = 1.3\,(1-A) + \left(\frac{1}{A} - 1\right)^2$$

- Two-plane screen with cylindrical bars, with a loss coefficient of

$$l_c = \frac{1.28\,(1-A)}{A^2}$$

From the calculated loss coefficient above, the pressure drop is calculated as:

$$\Delta p = \rho\left(\frac{l_c}{2}\right)(v^2)$$

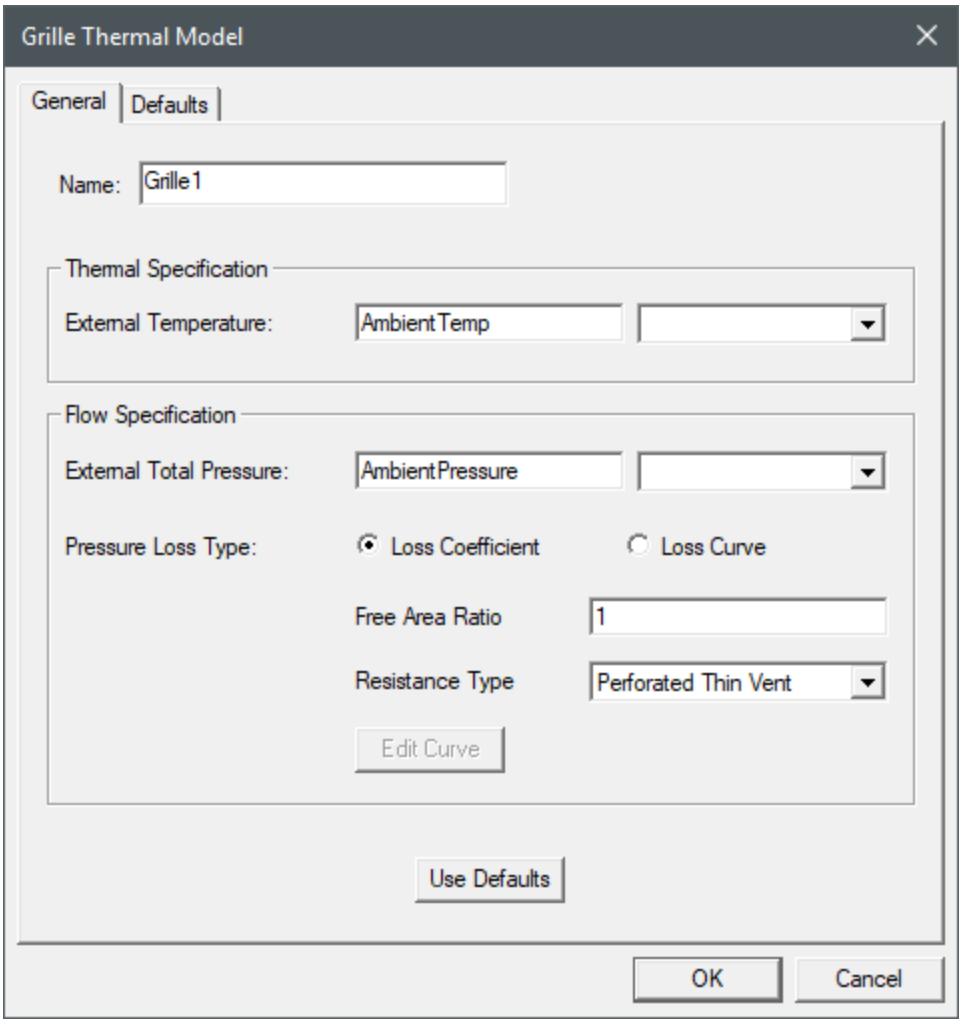
where v is the velocity that Icepak calculates based on the surrounding environment.

Loss Curve

You specify a piecewise-linear profile for the pressure drop as a function of speed. The pressure drop is then interpolated from the loss curve profile and the velocity that Icepak calculates based on the surrounding environment.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Grille Properties



The **Grille Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Thermal Specification	
External Temperature	**External Temperature** specifies the temperature of the external fluid region. This temperature is used if fluid flows into the mesh region through the vent, and is ignored if fluid flows out of the mesh region through the vent. The default **AmbientTemp** is the **Temperature** specified in the **Icepak Design Settings** dialog box.
Flow Specification	
External Total Pressure	**External Total Pressure** specifies the external pressure. For flow into a grille, the external pressure is the stagnation pressure. For flow out of a grille, the external pressure is the static pressure. he default **AmbientPressure** is the **Gauge Pressure** specified in the **Icepak Design Settings** dialog box.
Pressure Loss Type: Loss Coefficient	
Free Area Ratio	**Free Area Ratio** is the ratio of the area through which a fluid can flow unobstructed to the total planar area of the obstruction.
Resistance Type	**Resistance Type** specifies the type of resistance, which includes the options **Perforated Thin Vent**, **Circular Metal Wire Screen**, and **Two-Plate Screen Cyl. Bars**.
Pressure Loss Type: Loss Curve	
Edit Curve	Clicking **Edit Curve** displays the **Edit Dataset** dialog box. In the **Edit Dataset** dialog box, specify coordinates for velocity and pressure to create the pressure loss curve. See Adding Datasets for more information.

Assigning a Grille and Defining Thermal Properties

To assign a grille boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Grille**.
4. In the **Grille Thermal Model** dialog box, enter a **Name** for the grille.
5. Under **Thermal Specification**, enter the **External Temperature**.
6. Under **Flow Specification**, enter the **External Total Pressure**.
7. Select one of the following for **Pressure Loss Type**.

- **Loss Coefficient**: Enter the **Free Area Ratio** and **Resistance Type**.
- **Loss Curve**: Click **Edit Curve** to create pressure loss curve.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **OK**.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Network Properties

A network is a two-dimensional modeling boundary condition that can be used to approximate the thermal behavior complex components or systems. They consist of nodes linked together using either thermal resistance (R link) or unidirectional mass flow heat transfer (MC link) connections or links. The network schematic editor allows you to make fairly complicated, arbitrarily connected thermal network topology for modeling various components, like an IC package, a heat pipe, a cold plate, or a desktop server. Each network can consist of a set of external nodes (faces) that are connected to internal network nodes, which, in turn, can be connected to other internal nodes or boundary nodes. External face nodes can be connected to each other as well.

Steady-state or transient thermal analysis can be performed by adding thermal mass properties (Mass, Specific Heat) to the network internal nodes.

The Icepak network boundary condition is designed to be used inside a CFD model, that is, it cannot be used in a "standalone" mode. Within a network, the governing flow and heat transfer equations are reduced to simple algebraic equations. The node temperatures are related to the conduction heat transfers through the R-links and to mass flow heat transfers through the C-links for given mass flow rates. Each defined network node must be connected. Network object communicates with the rest of the Icepak model through the two-dimensional face nodes of a three-dimensional solid block. Since there is no need to solve CFD equations within network object, there is no need to mesh inside its embedding three-dimensional solid block. All volumetric CFD mesh must only exist in the rest of the Icepak model outside the network object. To do this, deselect **Solve Inside** in Properties Window. So, volumetric mesh inside is disabled as it is considered a hollow block.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

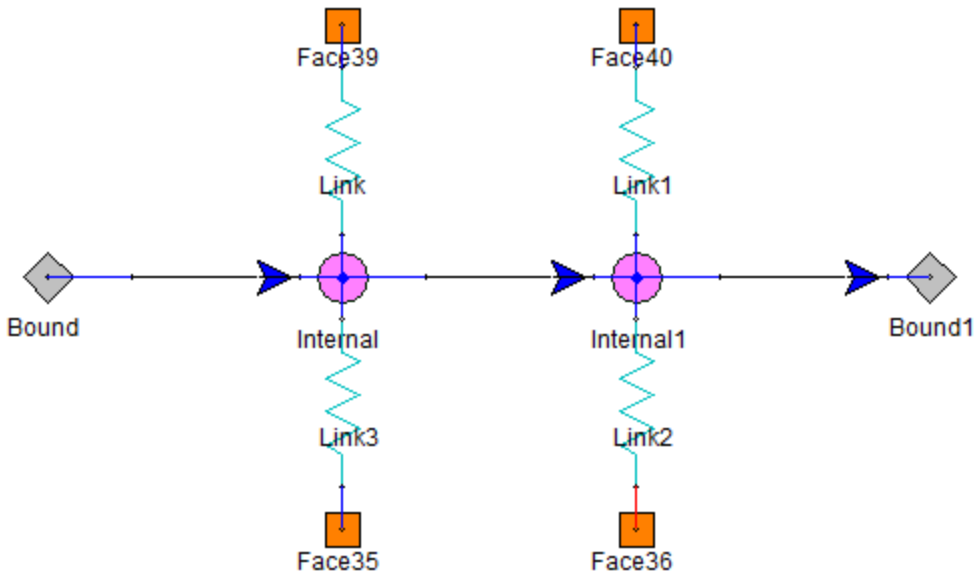
A simple way to create a network object in Icepak is to create a solid block with network volume dimensions. Then, in face-selection mode, you can select one or more faces of the network volume and assign a network boundary condition (**Icepak** > **Thermal** > **Network**).Then, edit it in the **Network Thermal Model** dialog box. Based on the number of selected faces, the schematic is initially populated with their R links to a single internal node. This basic network topology can be easily revised to adapt network object to more complex model.

Since there is no need to solve CFD equations within a network, there is no mesh inside. All volumetric CFD mesh must only exist in the rest of the Icepak model outside the network object. This is easily achieved by creating a 3D geometry that has:

- higher object priority than other intersecting and contacting objects.
- its Properties > "Solve Inside" is disabled to ensure no volumetric mesh inside

You can refer to this 3D geometry as the hollow block.

In face-selection mode, you can select one or more faces of the network volume and create a network boundary condition using the schematic in the **Network Thermal Model** dialog box. Based on the number of faces you select, the schematic is initially populated with nodes and links.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network Nodes and Links

Networks consist of a set of external nodes, boundary nodes, and internal nodes. External nodes represent the geometry of the network face, so that you can visualize the geometric shape to which the internal nodes are connected.

- Boundary nodes represent boundary conditions for the network node. They are just like regular boundary conditions that you would apply at a wall, such as fixed heat flux or fixed temperature.
- Internal nodes represent objects that you want to lump together to have a single temperature. For example, if you are modeling an IC package or a chip, the junction would be an internal node. The assumptions here are that the internal details of the chip are not important and that the most important characteristic that would be of interest in modeling is the junction node. The rest of the chip is modeled through resistance connections to surfaces.
- External nodes and internal nodes can be connected by resistive links (R links) or thermal mass flow links (MC links).
    - For IC packages, R-links can represent sophisticated thermal networks, for instance, modeling a package with multiple heat sources. The internal nodes will be dedicated for observing the various junction temperatures of these heat sources. Their connections to the external nodes, positioned on the faces of the hollow block, or to other passive internal nodes, are composing the interlinking thermal heat flow paths of the package.
    - For cold plate equipment, the created network model is combining flow and thermal heat transfers. Unidirectional MC links allow to specify mass flow rate as well as the flow direction. The mass flow rate entering and leaving any internal node must sum to zero to account for conservation of mass. Face nodes/surfaces must be located on domain boundaries (for example, a cabinet boundary or surfaces of objects with Solve Inside disabled) since they represent a connection to an external domain that you do not want to model in detail.

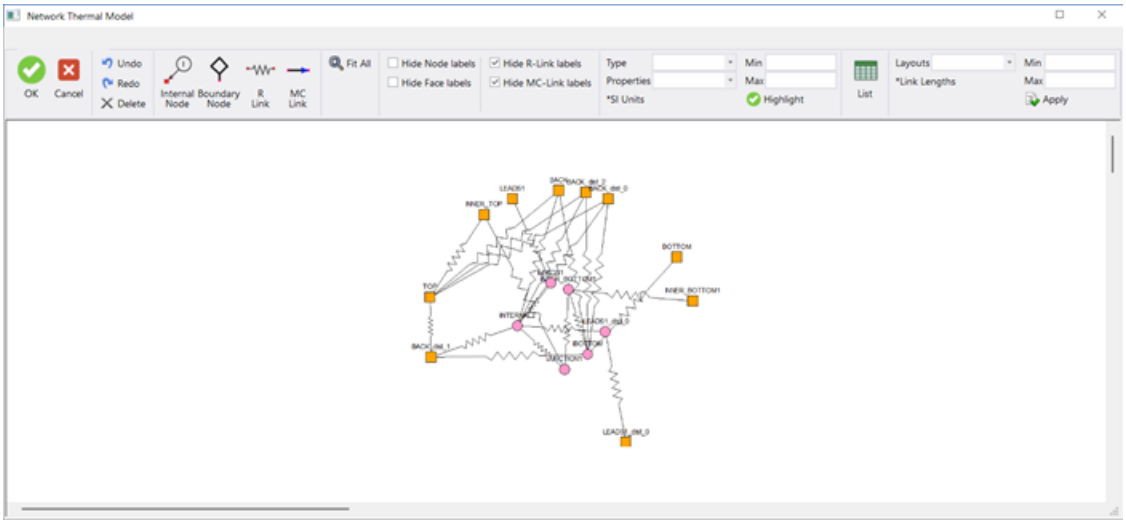
> **Note:**
>
> For complementary information about the building of a complex thermal network for IC package, the JEDEC standard JESD15-4 could be useful: JEDEC-JESD15-4 DELPHI Compact Thermal Model Guideline.

The Network Thermal Model Dialog

The **Network Thermal Model** dialog box is an editing and schematic tool, which allows you to create, view, or update any design of network boundary conditions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Boundary nodes and internal nodes with a power definition greater than zero appear red in the schematic.



The **Network Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition. Double-click a node or link to edit its properties. See Network Node and Link Properties for more information about face, boundary, and internal nodes, and R and MC links.

Note:

You can draw a box in the schematic or press **Ctrl** to select multiple components.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK	**OK** accepts the changes to the network and closes the dialog.
Cancel	**Cancel** closes the dialog without saving any changes.
Undo	**Undo** reverts the previous action.
Redo	**Redo** reapplies the previous action that was reverted.
Delete	**Delete** removes the selected component from the schematic. You can also right-click one or more components and select **Delete Selections**.
Internal Node	**Internal Node** allows you to place an internal node in the schematic.
Boundary Node	**Boundary Node** allows you to place a boundary node in the schematic.
R Link	**R Link** allows you to place a resistive link in the schematic. Select two nodes and click **R Link** to create a link.
MC Link	**MC Link** allows you to place a convective link in the schematic. Select two nodes and click **MC Link** to create a link. To reverse the direction of the MC link, right-click it in the schematic and select **Reverse Flow**.
Fit All	**Fit All** zooms to fit all or only selected components.
Hide Labels	The hide options allow you to hide or show the names for nodes, faces, R Links, and MC Links.
Highlight Filters	To highlight links or nodes in the schematic in red, select a **Type** and one of its **Properties** from the drop-down lists. Then specify the **Min** and **Max** value range to highlight. Click **Highlight** to display links in red and nodes with a red outline.
List	**List** opens the **Network Properties** dialog box, which displays link and node properties. See Network Properties Table for more information.
Layouts	In the network schematic, the **Circular** layout is selected by default. Optionally, select the **Fruchterman-Reingold** layout, which considers network nodes, network display area and minimum and maximum lengths for the links. The Fruchterman-Reingold layout is recommended for networks with fewer than 50 nodes.

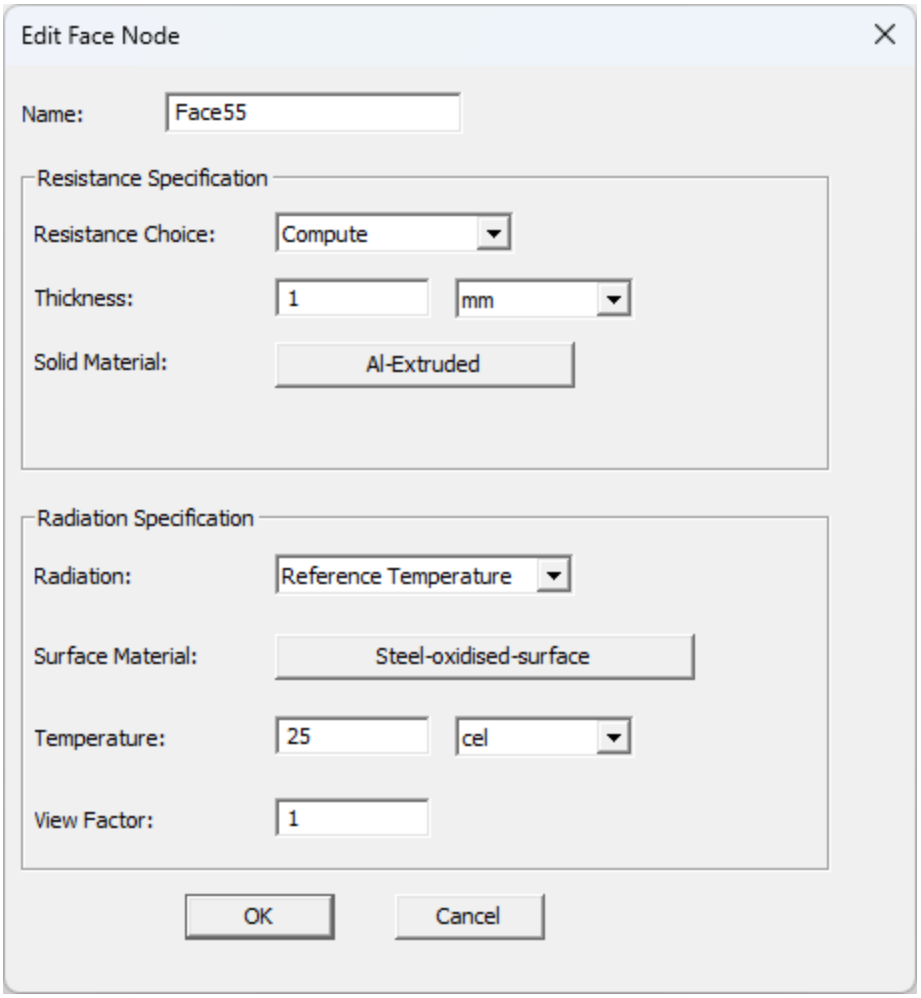
Network Node and Link Properties

Network node and link properties are defined in dialog boxes, which you can open by double-clicking the component in the schematic.

Face Node Properties

Face node properties are defined in the **Edit Face Node** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



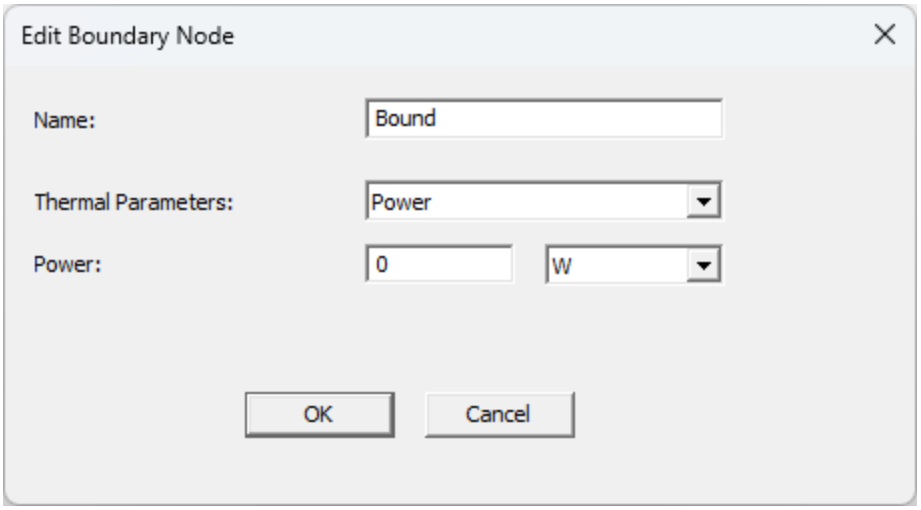
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resistance Specification	
Name	The **Name** appears in the schematic editor.
Resistance Specification	Select **Compute** or **Specified**. **Compute** uses the specified **Thickness** value and the specified **Solid Material** thermal conductivity to calculate the thermal resistance. **Specified** allows you to directly specify the **Thermal Resistance** value if it is known.
Thickness	When **Compute** is selected, **Thickness** allows you to specify the thickness of the node. Also specify the **Unit** of measurement.
Solid Material	When **Compute** is selected, **Solid Material** allows you to open the **Select Definition** dialog box to define the node's material.
Thermal Resistance	When **Specified** is selected, **Thermal Resistance** allows you to directly specify the node's thermal resistance if it is known.
Radiation Specification	
Radiation	Select **None**, **All Objects** or **Reference Temperature**. specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. **Reference temperature** specifies a fixed temperature to which the selected sides of the object radiate heat.
Surface Material	**Surface Material** specifies the material type for the side of the network using the **Select Definition** dialog box.
Temperature	When **Reference Temperature** is selected, **Temperature** specifies a fixed temperature to which the selected sides of the object radiate heat.
View Factor	When **Reference Temperature** is selected, enter a **View factor**, which is the proportion of radiation that leaves the network and contacts surrounding objects. The default view factor is 1.0.

Boundary Node Properties

Boundary node properties are defined in the **Edit Boundary Node** dialog box.

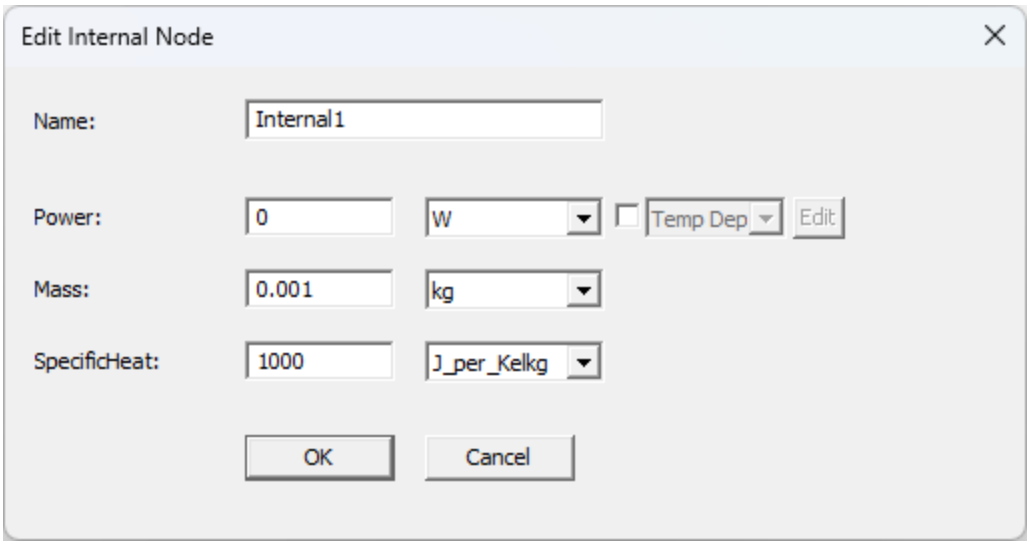
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameter Values	
Name	The **Name** appears in the schematic editor.
Thermal Parameters	Select **Power** or **Temperature**.
Power	When **Power** is selected, specify the power of the node. Also specify the **Unit** of measurement.
Temperature	When **Temperature** is selected, specify the temperature of the node. Also specify the **Unit** of measurement.

Internal Node Properties

Internal node properties are defined in the **Edit Internal Node** dialog box.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

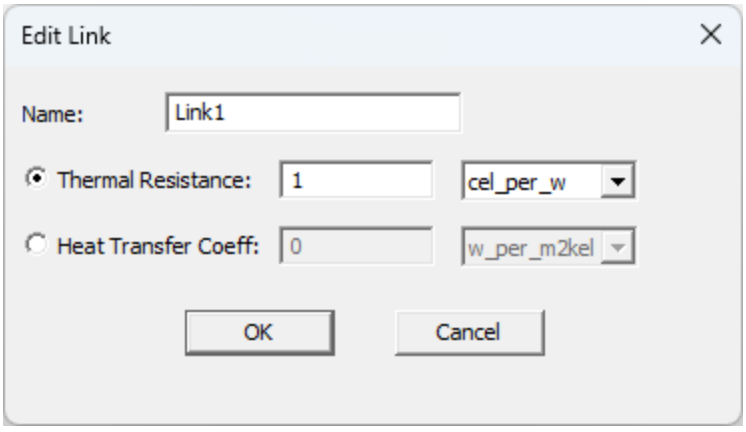
Parameter Values	
Name	The **Name** appears in the schematic editor.
Power	**Power** allows you to specify the power of the node. Also specify the **Unit** of measurement. Enable the check box specify temperature-dependent or transient power. For steady-state simulations, if needed, select the check box and select **Temp Dep**. For **Temp Dep**, click **Edit** to specify the temperature dependence of power. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. **Transient** allows you to specify power as a function of time. **DTM** (dynamic thermal modeling) allows you to specify a temperature-dependent power after each time step of a transient simulation.
Mass	**Mass** allows you to specify the mass of the node. Also specify the **Unit** of measurement.
Specific Heat	**Specific Heat** allows you to specify the heat of the node. Also specify the **Unit** of measurement.

Note:

- For more information about specifying temperature-dependent power, see Using Datasets to Define Temperature Dependent Parameters.
- For more information on specifying transient parameters, see Specifying Parameters as a Function of Time.
- For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.

R Link Properties

R link properties are defined in the **Edit Link** dialog box.

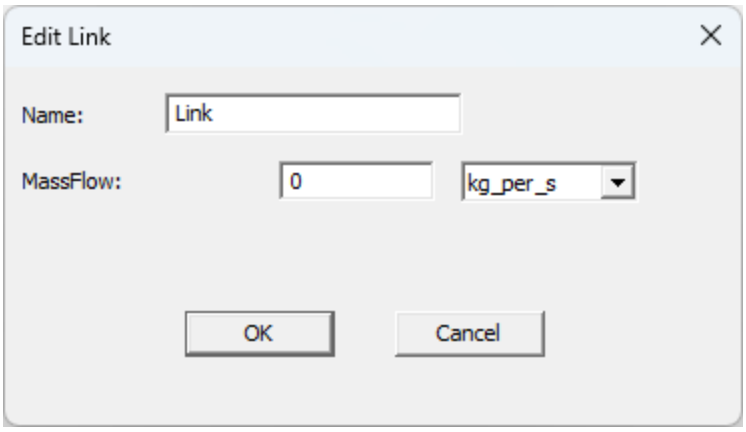


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter Values	
Name	The **Name** appears in the schematic editor.
Thermal Resistance	When **Thermal Resistance** is selected, specify the node's thermal resistance if it is known.
Heat Transfer Coeff	When **Heat Transfer Coeff** is selected, specify the link's convective heat transfer. Icepak computes the thermal resistance as a $R = \frac{1}{ha}$, where *h* is the heat transfer coefficient and *a* is the area of the face.

MC Link Properties

MC link properties are defined in the **Edit Link** dialog box.



Parameter Values	
Name	The **Name** appears in the schematic editor.
Mass Flow	**Mass Flow** allows you to specify the link's mass flow rate.

> **Note:**
>
> Net incoming and outgoing mass flow from MC Links at an internal node should be 0.

Network Properties Table

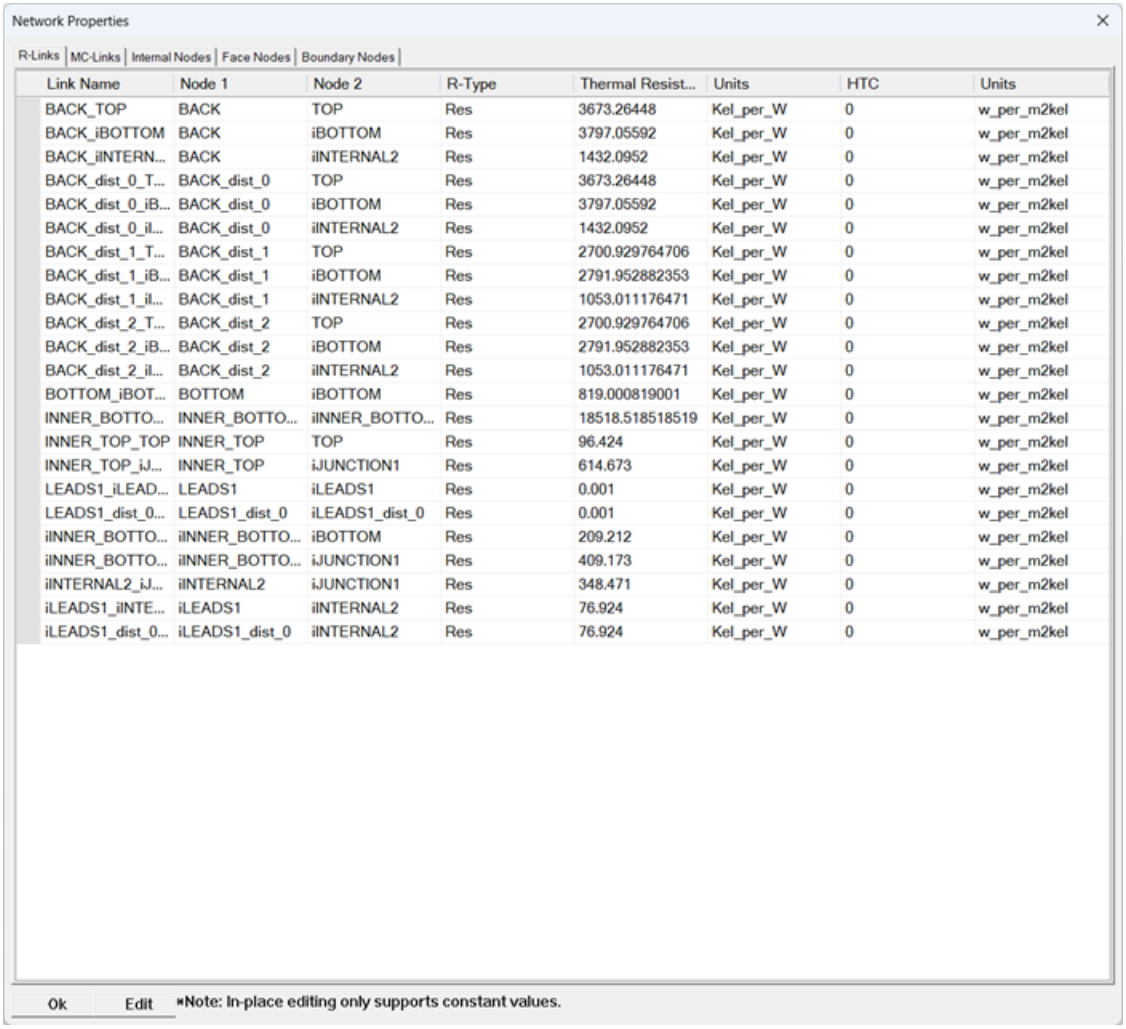
In the **Network Thermal Model** dialog box, click **List** to open the **Network Properties** dialog box, which displays link and node property information on up to five tabs (if the link and node types exist in the schematic):

- R Links
- MC Links

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Internal Nodes
- Face Nodes
- Boundary Nodes

Click any column heading to sort the property table. You can edit link and node properties directly in each cell. Alternative, select a row and click **Edit** to define properties in the dialog boxes described in Network Node and Link Properties.



Link Name	Node 1	Node 2	R-Type	Thermal Resist...	Units	HTC	Units
BACK_TOP	BACK	TOP	Res	3673.26448	Kel_per_W	0	w_per_m2kel
BACK_iBOTTOM	BACK	iBOTTOM	Res	3797.05592	Kel_per_W	0	w_per_m2kel
BACK_iINTERN...	BACK	iINTERNAL2	Res	1432.0952	Kel_per_W	0	w_per_m2kel
BACK_dist_0_T...	BACK_dist_0	TOP	Res	3673.26448	Kel_per_W	0	w_per_m2kel
BACK_dist_0_iB...	BACK_dist_0	iBOTTOM	Res	3797.05592	Kel_per_W	0	w_per_m2kel
BACK_dist_0_iI...	BACK_dist_0	iINTERNAL2	Res	1432.0952	Kel_per_W	0	w_per_m2kel
BACK_dist_1_T...	BACK_dist_1	TOP	Res	2700.929764706	Kel_per_W	0	w_per_m2kel
BACK_dist_1_iB...	BACK_dist_1	iBOTTOM	Res	2791.952882353	Kel_per_W	0	w_per_m2kel
BACK_dist_1_iI...	BACK_dist_1	iINTERNAL2	Res	1053.011176471	Kel_per_W	0	w_per_m2kel
BACK_dist_2_T...	BACK_dist_2	TOP	Res	2700.929764706	Kel_per_W	0	w_per_m2kel
BACK_dist_2_iB...	BACK_dist_2	iBOTTOM	Res	2791.952882353	Kel_per_W	0	w_per_m2kel
BACK_dist_2_iI...	BACK_dist_2	iINTERNAL2	Res	1053.011176471	Kel_per_W	0	w_per_m2kel
BOTTOM_iBOT...	BOTTOM	iBOTTOM	Res	819.000819001	Kel_per_W	0	w_per_m2kel
INNER_BOTTO...	INNER_BOTTO...	iINNER_BOTTO...	Res	18518.518518519	Kel_per_W	0	w_per_m2kel
INNER_TOP_TOP	INNER_TOP	TOP	Res	96.424	Kel_per_W	0	w_per_m2kel
INNER_TOP_iJ...	INNER_TOP	iJUNCTION1	Res	614.673	Kel_per_W	0	w_per_m2kel
LEADS1_iLEAD...	LEADS1	iLEADS1	Res	0.001	Kel_per_W	0	w_per_m2kel
LEADS1_dist_0...	LEADS1_dist_0	iLEADS1_dist_0	Res	0.001	Kel_per_W	0	w_per_m2kel
iINNER_BOTTO...	iINNER_BOTTO...	iBOTTOM	Res	209.212	Kel_per_W	0	w_per_m2kel
iINNER_BOTTO...	iINNER_BOTTO...	iJUNCTION1	Res	409.173	Kel_per_W	0	w_per_m2kel
iINTERNAL2_iJ...	iINTERNAL2	iJUNCTION1	Res	348.471	Kel_per_W	0	w_per_m2kel
iLEADS1_iINTE...	iLEADS1	iINTERNAL2	Res	76.924	Kel_per_W	0	w_per_m2kel
iLEADS1_dist_0...	iLEADS1_dist_0	iINTERNAL2	Res	76.924	Kel_per_W	0	w_per_m2kel

Assign Network Properties

To assign a network boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select and right-click one or more faces in the **3D Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the right-click menu, select **Assign Thermal**>**Network**.

> **Note:**
>
> Face nodes are created for each face you select. When you select one or two faces, an internal node is automatically created. When selecting three or more faces, no internal nodes are automatically created.

4. In the **Network Thermal Model** dialog box, enter a **Name** for the network.
5. Use the network schematic editor to create the network structure. See The Network Thermal Model Dialog for more information.
    - Edit the node and link properties as needed. See Network Node and Link Properties for more information.
6. When you are finished creating and defining network nodes and links, click **OK**.

Assigning Opening Properties

An opening is a two-dimensional modeling boundary condition that represents an area on the surface of a solid object (such as a block or plate) or an area on a planar object such as a wall through which a fluid is free to flow in any direction. Opening types include free and recirculating. Free openings are specified individually, and recirculating openings must be specified in pairs. Recirculating opening pairs consist of two sections:

- an extract section, representing the location at which fluid is removed from the enclosure
- a supply section, representing the location at which fluid is returned to the enclosure

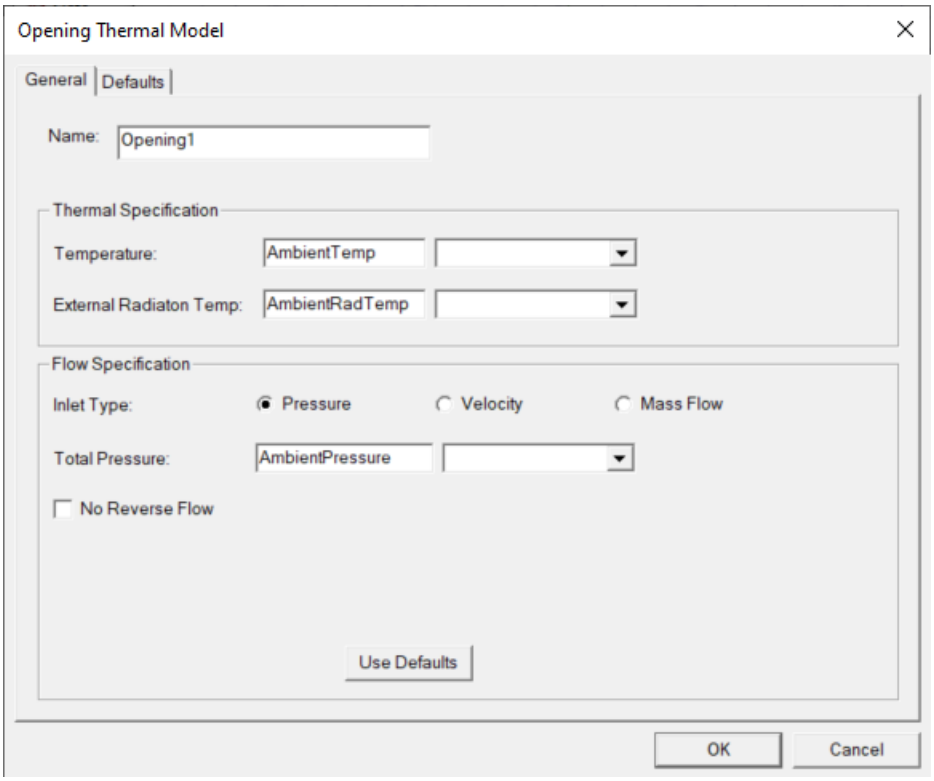
Openings should be located on an enclosure boundary, that is, either a mesh region wall or the surface of a hollow block that has been used to mask a portion of the enclosure. Free openings can be located on the surfaces of blocks or plates within the enclosure.

Assigning Free Opening Properties

A free opening represents an area on the surface of a solid object (e.g., a block or plate) or an area on a planar object (e.g., a wall) through which a fluid is free to flow in any direction. In most cases, a free opening represents a hole on the boundary of the global mesh region where the model fluid is exposed to the external environment. They can also represent holes either on the boundary of the enclosure or on blocks or plates.

For openings located on mesh region walls, Icepak computes the rate of flow through the opening based on an external static pressure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Opening Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the opening appears in the **Project Manager** window and **3D Modeler** window.
Thermal Specification	
Temperature	**Temperature** is the temperature of the fluid external to the mesh region. For transient simulations, select **Transient**, to specify temperature as a function of time.
External Radiation Temperature	**External Radiation Temperature** specifies the external radiation temperature. By default, the external radiation temperature is the **Radiation temp** specified in the **Icepak Design Settings** dialog box.
Flow Specification (Inlet Type: Pressure)	
Inlet Type: Pressure	**Pressure** specifies the pressure of the flow as a single value.
Total Pressure	**Total Pressure** is the pressure of the flow at the inlet plane. For transient simulations, select **Transient**, to specify pressure as a function of time.
No Reverse Flow	**No Reverse Flow** prevents reverse flow for the opening. When enabled, the mass flow is set to zero when reverse flow is detected.
Flow Specification (Inlet Type: Velocity)	
Inlet Type: Velocity	**Velocity** specifies the flow by defining the static pressure and speed of the flow in the X, Y, and Z planes of the opening.
Static Pressure	**Static Pressure** specifies the pressure of the fluid. For transient simulations, select **Transient**, to specify pressure as a function of time.
X, Y, Z Velocity	The **X**, **Y**, and **Z Velocity** values specify a uniform flow speed across the respective plane of the opening. For transient simulations, select **Transient** or **DTM**. **Transient** allows you to specify velocity as a function of time. **DTM** (dynamic thermal modeling) allows you to specify a temperature-dependent velocity after each time step of a transient simulation.
Flow Specification (Inlet Type: Mass Flow)	
Inlet Type: Mass Flow	**Mass Flow Rate** specifies the flow by defining the static pressure and mass flow rate at the opening.
Static Pressure	**Static Pressure** specifies the pressure of the fluid. For transient simulations, select **Transient**, to specify pressure as a function of time.
Mass Flow Rate	**Mass Flow Rate** specifies the mass flow rate of the fluid. For transient simulations, select **Transient**, or **DTM**. **Transient** allows you to specify mass flow as a function of time. **DTM** (dynamic thermal modeling) allows you to specify a temperature-dependent mass flow after each time step of a transient simulation.
Flow Condition	For **Flow Condition**, if the fluid flows into the opening, select **In**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Direction	**Flow**. Or, if the fluid flows out of the opening, select **Out Flow**. For **Flow Direction**, if the fluid flows into the computational domain normal to the opening, select **Normal to Boundary**. To specify the direction of the flow, select **Direction Vector**.

Note:

For more information on specifying transient parameters, see Specifying Parameters as a Function of Time. For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.

Assign a Free Opening and Define Thermal Properties

To assign a free opening boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry face in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Opening** > **Free**.
4. In the **Opening Thermal Model** dialog box, enter a **Name** for the opening.
5. Under **Thermal Specification**, enter values for **Temperature** and **External Radiation Temperature**.
6. Under **Flow Specification**, select an **Inlet Type** of **Pressure** or **Velocity**.
   - For **Pressure**, enter a value for **Total Pressure** and, if needed, select **No Reverse Flow**.
   - For **Velocity**, enter values for **Static Pressure**, **X Velocity**, **Y Velocity**, and **Z Velocity**. For transient simulations, if needed, enable the check box and select **Transient** or **DTM**. Define a variation function f or **Transient**.
   - For **Mass Flow Rate**, enter values for **Static Pressure** and **Mass Flow Rate**. Also, specify a **Flow Condition**, **Flow Direction**, and, if needed, **Direction Vector** values. For transient simulations, if needed, enable the check box and select **Transient** or **DTM**. Define a variation function f or **Transient**.
7. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

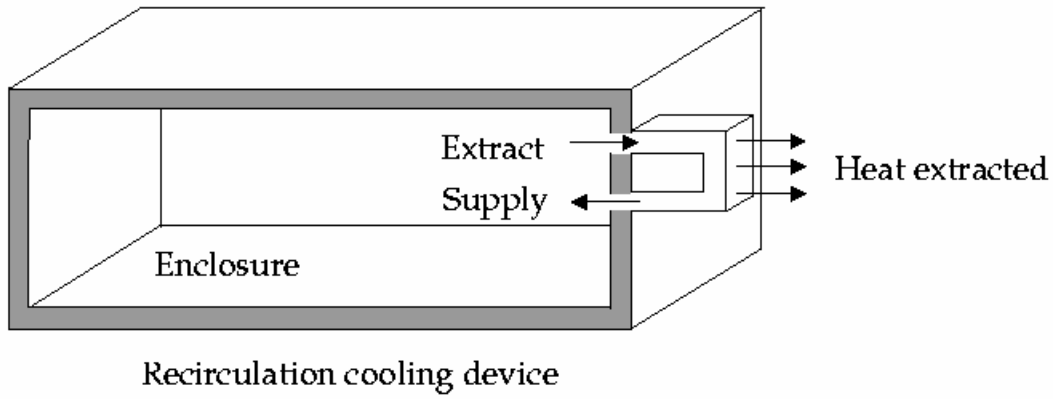
Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Recirculating Opening Properties

Recirculating openings model recirculation devices such as heating or cooling units. In such a device, the fluid is withdrawn from the cabinet through the extract section of the opening and returned to the cabinet through the supply section, as shown in the following image. The extract and supply sections of the recirculation opening can differ from one another in both size and geometry.

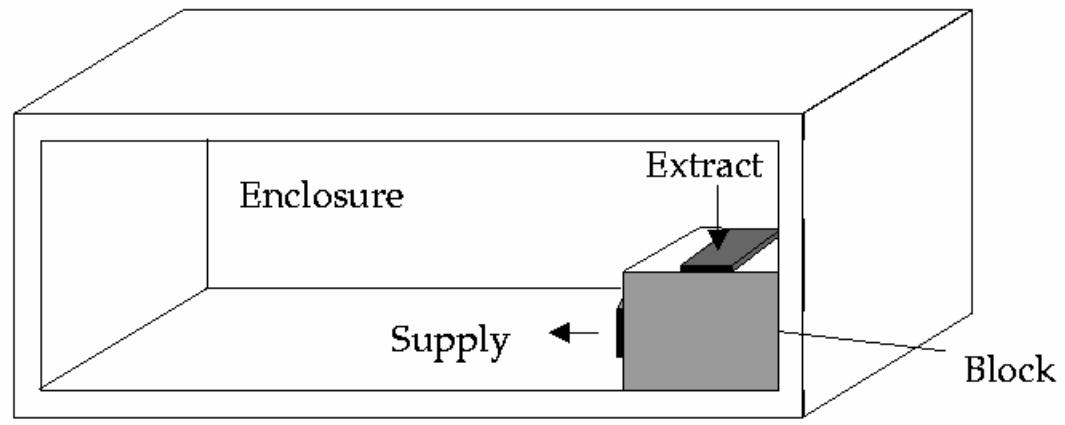


Recirculation cooling device

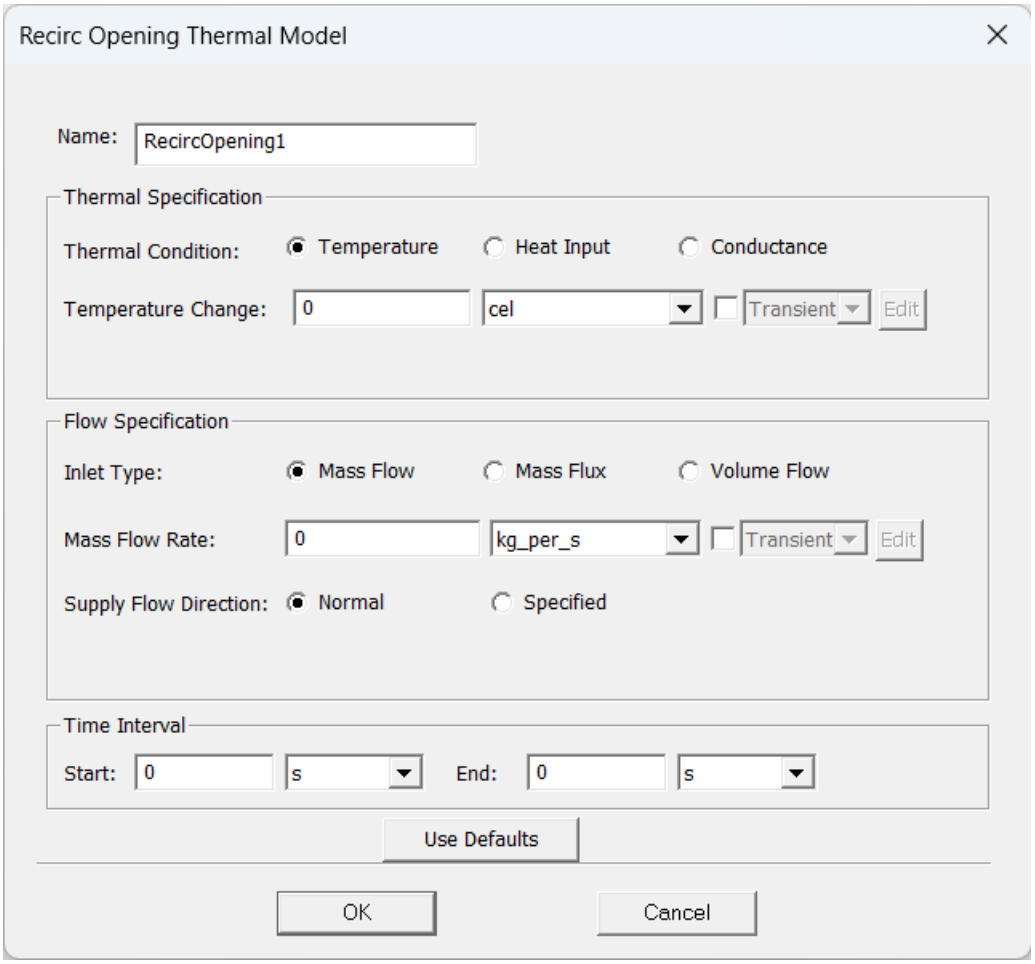
You can model an internal recirculation device by placing the extract and supply sections of a recirculating opening on two different sides of an adiabatic block, as shown in the following image.

Note: A conducting solid block cannot be used to represent an internal recirculation device.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For openings located on mesh region walls, Icepak computes the rate of flow through the opening based on an external static pressure.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

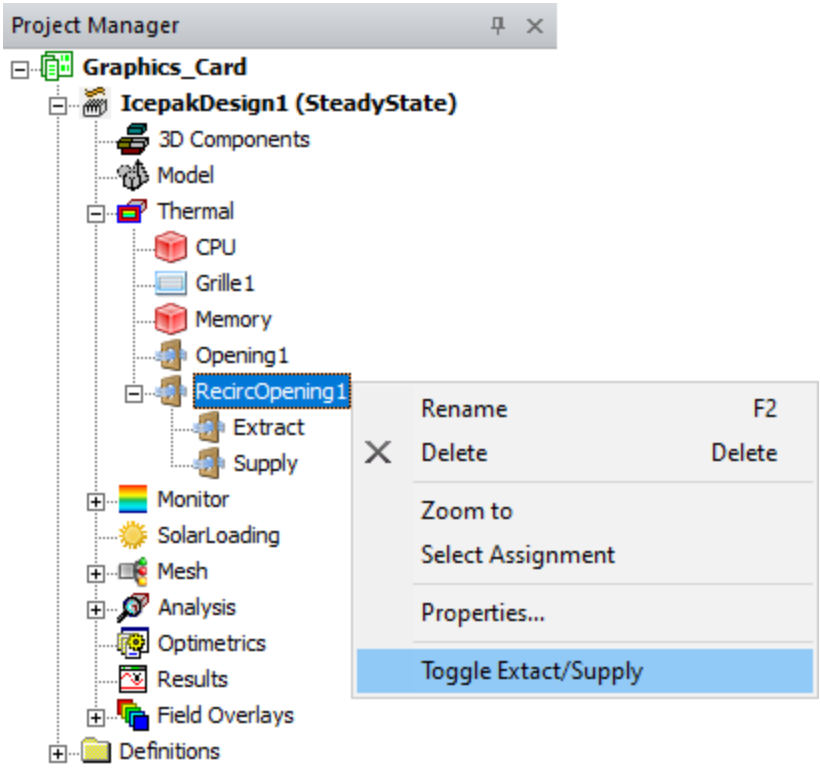
The **Recirc Opening Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Name	The **Name** of the opening appears in the **Project Manager** window and **3D Modeler** window.
Thermal Specification	
Temperature	**Temperature Change** specifies a constant increase or decrease of temperature applied to the fluid in the recirculation loop.
Heat Input	**Heat Flow** specifies a constant heat flow into or out of the fluid as it passes through the recirculation loop.
Conductance	**Conductance** specifies a conductance to model the heat input/output of the fluid from your system. Also enter an **External Radiation Temperature**, which specifies the external radiation temperature. By default, the external radiation temperature is the **Radiation temp** specified in the **Icepak Design Settings** dialog box.
Flow Specification (Inlet Type: Mass Flow)	
Inlet Type: Mass Flow	**Pressure** specifies the pressure of the flow as a single value.
Flow Specification (Inlet Type: Mass Flux)	
Inlet Type: Mass Flux	**Velocity** specifies the flow by defining the static pressure and speed of the flow in the X, Y, and Z planes of the opening.
Flow Specification (Inlet Type: Volume Flow)	
Inlet Type: Volume Flow	**Mass Flow Rate** specifies the flow by defining the static pressure and mass flow rate at the opening.
Supply Flow Direction	For **Supply Flow Direction**, if the fluid flows into the recirculating opening normal to the opening, select **Normal**. To specify the flow angle of the fluid entering the computational domain through the supply section of the opening, select **Specified**. Enter values for the direction vector (**X**, **Y**, and **Z**).
Time Interval (Transient Only)	Under **Time Interval**, the **Start** time and the **End** time specify when the thermal and mass flow specifications for the opening are active.

Recirculating Opening Project Manager Options

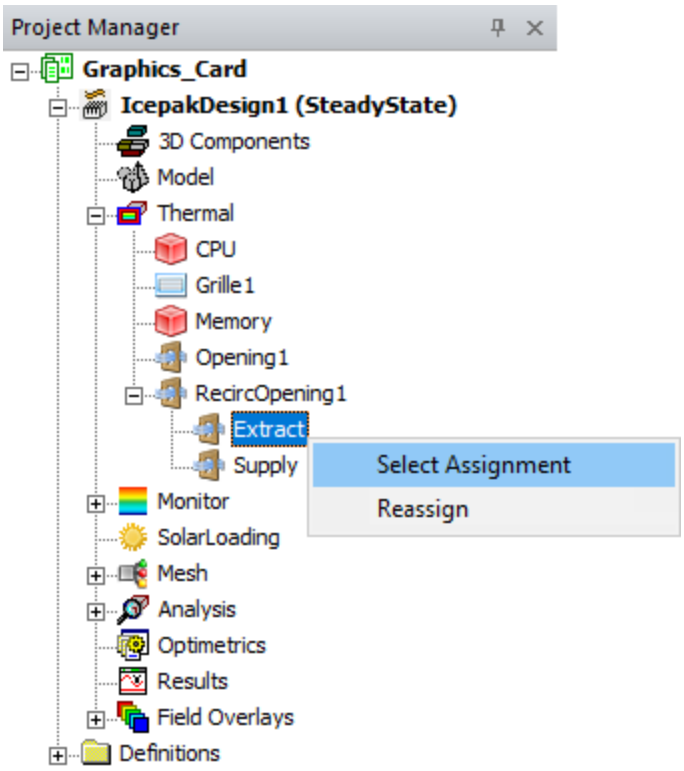
In the **Project Manager** under **Thermal**, right-click on the recirculating opening boundary condition to view the menu. Select **Toggle Extract/Supply** to reverse the assignments of the **Extract** and **Supply** geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Expand the recirculating opening to view the **Extract** and **Supply** assignments. Right-click either of these to view the menu. Select Assignment selects the assigned geometry in the **3D Modeler** window. To reassign the **Extract** or **Supply** opening, select the desired geometry in the **3D Modeler** window, right-click **Extract** or **Supply**, and select **Reassign**.

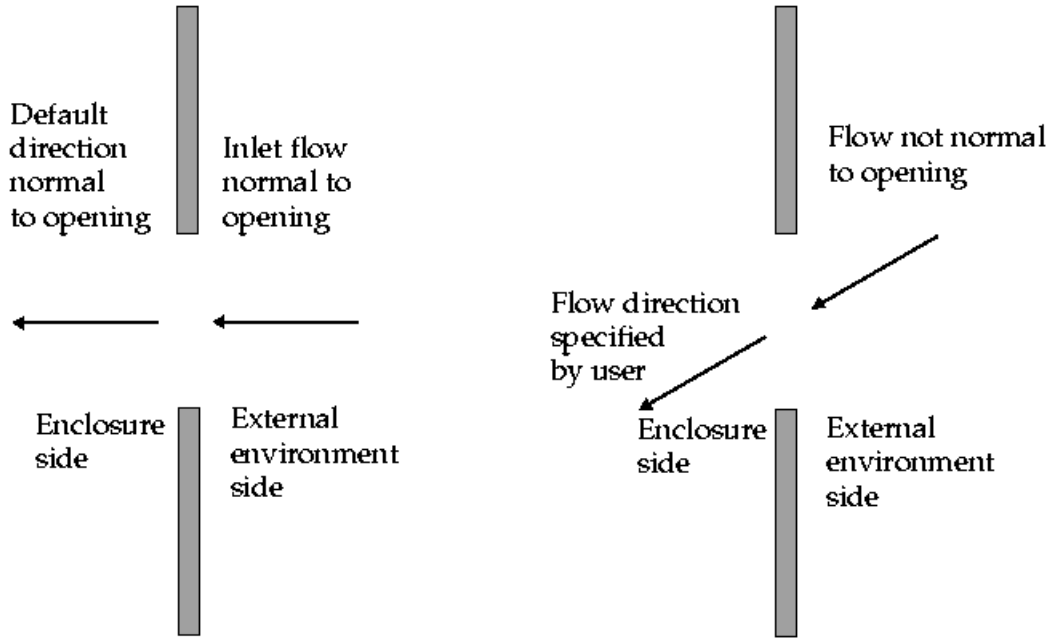
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Flow Direction for Recirculation Openings

By default, the flow direction of the fluid passing through the extract or supply section of a recirculation opening is normal to the plane of the section. However, you can specify that the flow leaves the supply section at an angle, as shown in the following image. Note that Icepak uses the direction parameters only to determine the direction of fluid flow; they do not affect the magnitude of the velocity.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Recirculation Mass Flow Rate

Icepak provides two methods of specifying the flow rate through a recirculation device:

- total mass flow through the opening
- mass flow rate per unit area of the opening

Because the extract and supply sections can differ in size, the per-unit-area specification can result in different mass fluxes but the same mass flow rate for each section of the opening. Such specifications can be used to model devices that extract or supply a fluid anywhere in the recirculation loop.

Recirculating Opening Thermal Specifications

When fluid exits and re-enters the enclosure through a recirculation device, its temperature can increase or decrease. Icepak computes the temperature of the re-entering fluid (T_{supply}) based on the temperature of the exiting fluid ($T_{extract}$) and the thermal change applied to the fluid as it passes through the device. Icepak provides three methods for computing T_{supply}. The first method requires the specification of a constant temperature change (ΔT) applied to the fluid within the device. In this case, T_{supply} is computed by

$$T_{supply} = T_{extract} + \Delta T$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $T_{extract}$ is the temperature of the enclosure fluid averaged over the face of the extract section. The second method requires the specification of the amount of heat (ΔT) input to or extracted from the fluid by the recirculation device. In this case, T_{supply} is computed by

$$T_{supply} = T_{extract} + \frac{\Delta H}{c_p \dot{m}}$$

where c_p is the fluid specific heat of the default fluid material and $\dot{m}$ is the mass flow rate through the device.

The third method requires the specification of the conductance ($h_1 A$) and the external temperature ($T_{external}$). In this case, T_{supply} is computed by

$$T_{supply} = T_{extract} - \frac{h_1 A(T_{extract} - T_{external})}{c_p \dot{m}}$$

where c_p is the specific heat of the default fluid material and $\dot{m}$ is the mass flow rate through the device.

Assign a Recirculating Opening and Defining Thermal Properties

To assign a recirculating opening boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select two geometry faces to represent the supply and extract sections and right-click in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Opening** > **Recirculating**.
4. In the **Recirc Opening Thermal Model** dialog box, enter a **Name** for the opening.
5. Under **Thermal Specification**, select a **Thermal Condition** of **Temperature**, **Heat Input**, or **Conductance** and enter the associated value. If you selected **Conductance**, also enter a value for **External Radiation Temperature**.
6. Under **Flow Specification**, select an **Inlet Type** of **Mass Flow**, **Mass Flux**, or **Volume Flow** and enter the associated value.
7. Specify a **Supply Flow Direction** and, if needed, direction vector values.
8. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

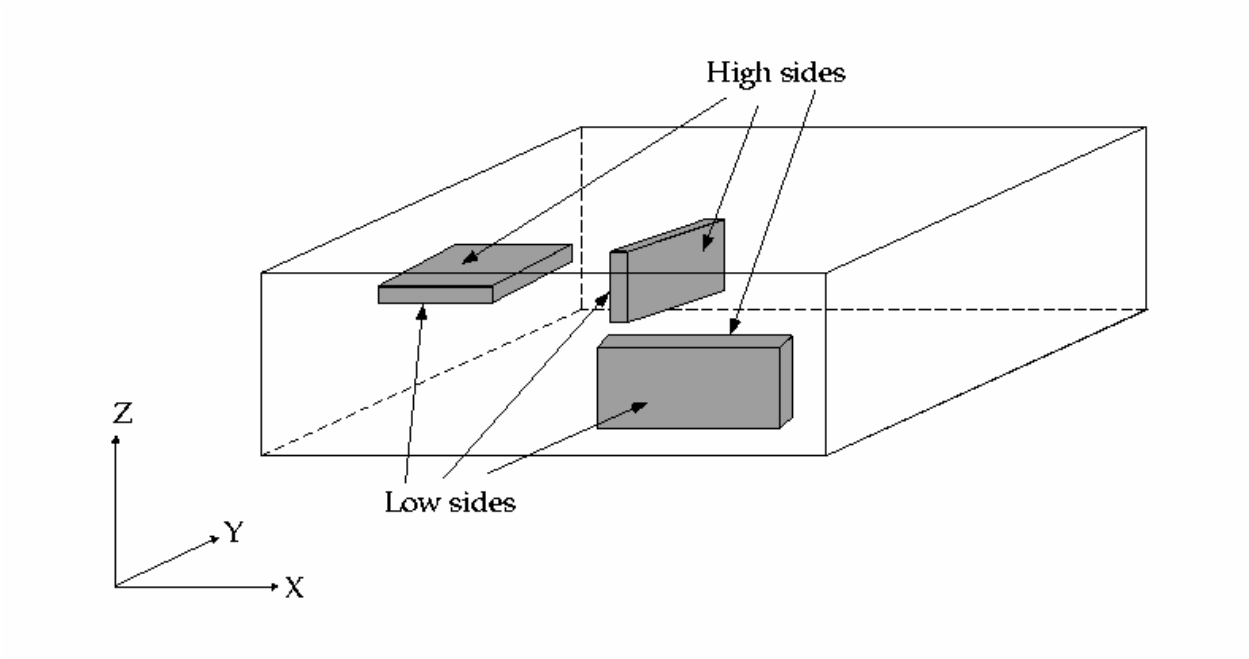
Assigning Plate Properties

A plate is a two-dimensional modeling boundary condition that is impervious to fluid flow. Plates can be used alone or in conjunction with other modeling objects to create complex objects in order to perform sophisticated thermal simulations. For example, plates and blocks can be used to build a complex PCB configuration that allows for a more detailed representation of a PCB. They can possess a thickness and are defined by both their geometry and their type. Plate geometries include rectangular, 2D polygon, circular, and 2D CAD. To assign a plate boundary condition, the selected geometry must be two-dimensional (for example, a rectangle or the face of 3D geometry).

Two plate types are available: adiabatic and conducting. Multiple types of conducting plates are available and are defined by their associated thermal specifications, including thickness, conductance, thermal resistance, and thermal impedance. Adiabatic thin plates do not conduct heat either across or in the plane of the plate. Conducting plates can conduct heat in either direction.

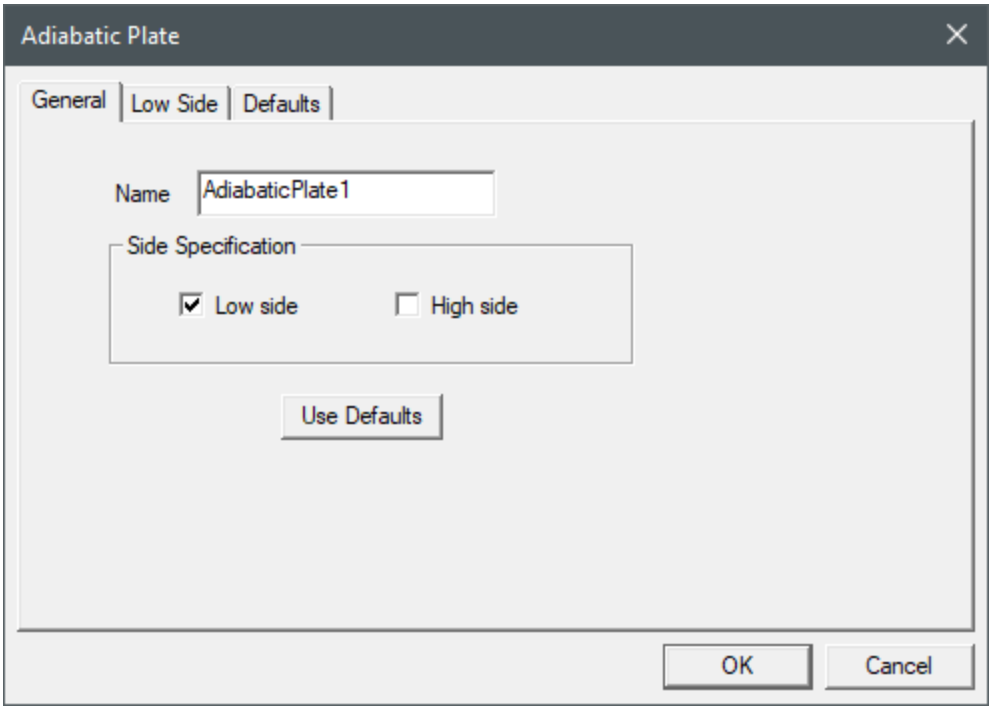
Plate sides are referred to as high and low, relative to the coordinates of the plane perpendicular to the plate. See figure below. The sides of a plate can exchange radiation with other objects in the model.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adiabatic Properties

Adiabatic plates have zero thickness and do not conduct heat in any direction, either normal to the plate or along the plane of the plate.

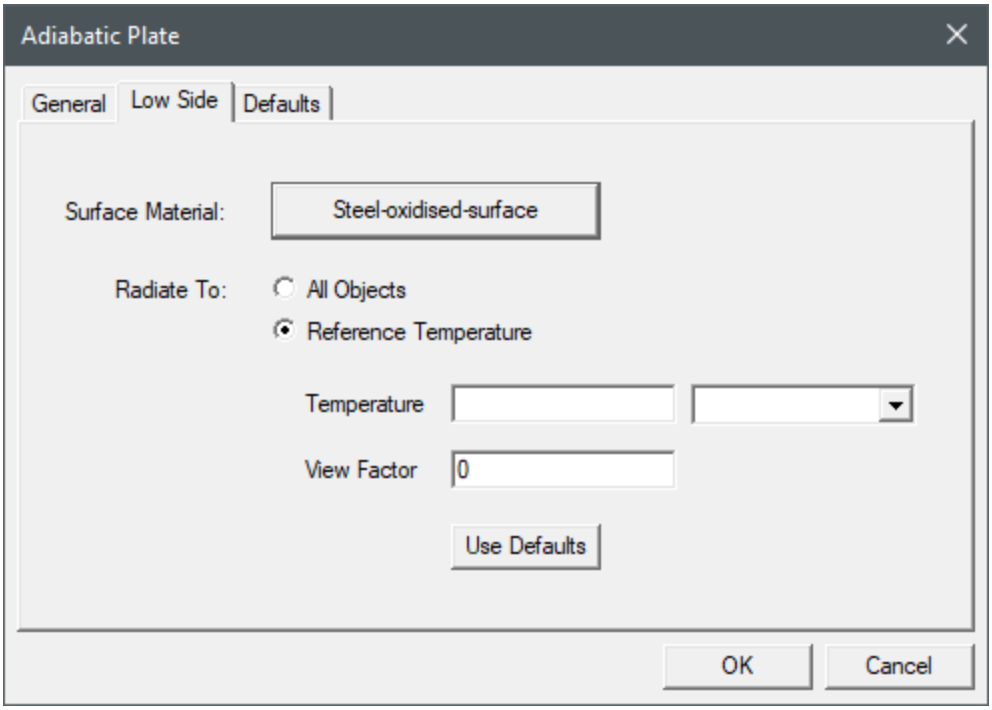


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Adiabatic Plate** dialog box **General** tab contains the following general information for the selected boundary condition.

Name	The **Name** of the plate appears in the **Project Manager** window and **3D Modeler** window.
Side Specification	**Low side** and **High side** enables the specification of different physical characteristics for each side of the plate.

If you select **Low side** and/or **High side**, the **Low Side**and **High Side** tabs contain the following thermal information.



Surface Material	**Surface Material** specifies the material type for the side of the plate using the **Select Definition** dialog box.
Radiate To	Select **All Objects** or **Reference Temperature**. **All Objects** specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. **Reference temperature** specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Temperature** and **View factor**, which is the proportion of radiation that leaves the plate and contacts surrounding objects. The default view factor is 1.0 and the value of the ambient temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning an Adiabatic Plate and Defining Thermal Properties

To assign an adiabatic plate boundary condition and define its properties:

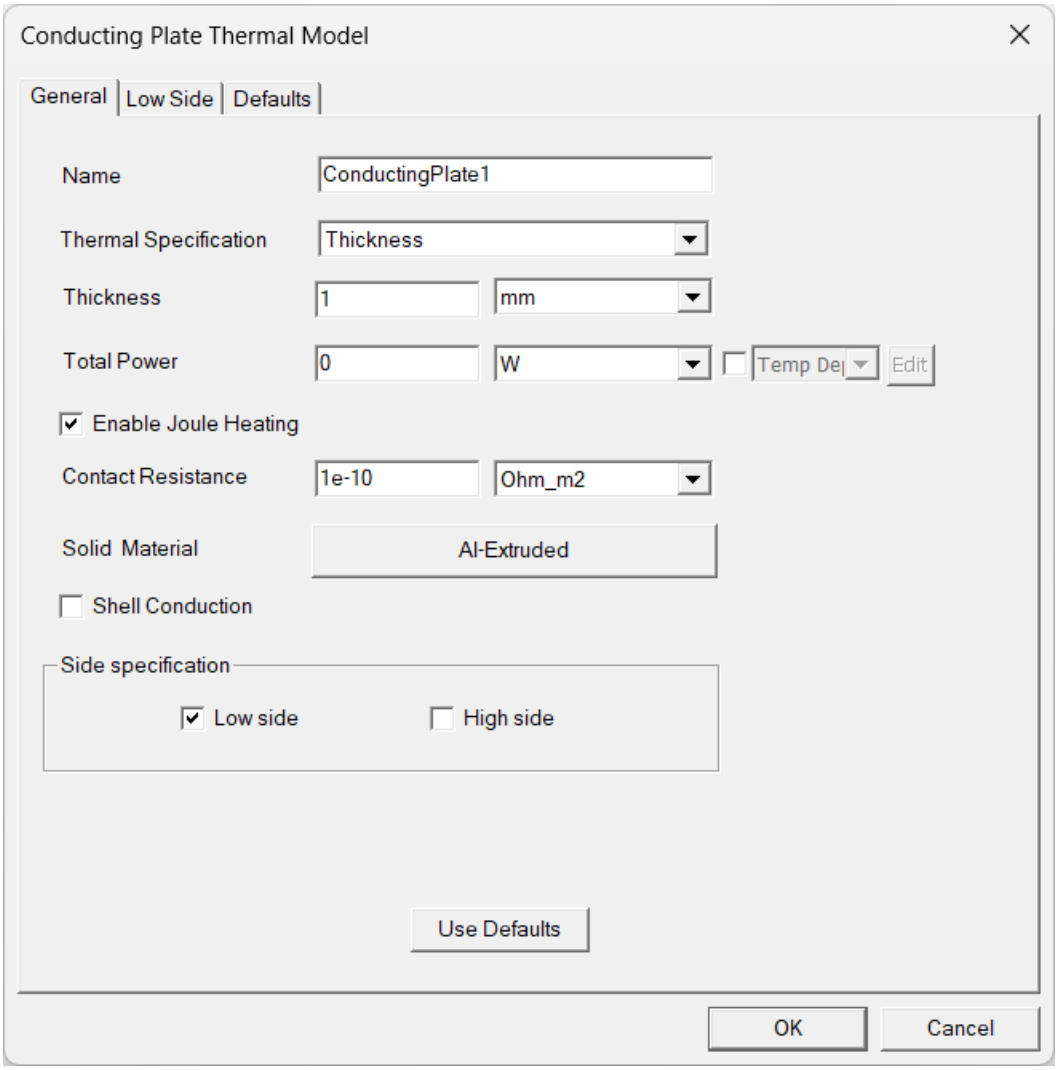
1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Plate** > **Adiabatic**.
4. On the **General** tab, enter a **Name** for the object.
5. If needed, under **Side Specification**, select **Low side** and/or **High side** to enable thermal specification of the low and/or high side of the plate.
6. Do one of the following:
   - If you did not select a **Side Specification** check box, click **OK**.
   - If you selected **Low side** and/or **High side**, click the **Low Side**and/or **High Side** tab.
7. In the next dialog box next to **Surface Material**, click **vacuum** to open the **Select Definition** dialog box. On the **Materials** tab, select the material for the side of the plate and click **OK**.
8. For **Radiate To**, click **All Objects** or **Reference Temperature**.
9. If you selected **Reference Temperature**, enter a **Temperature** and **View Factor**.
10. If you selected both **Low side** and **High side**, repeat steps 6 through 9.
11. Click **OK**.

> **Note:**
>
> The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:
>
> - **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
> - **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Conducting Plate Properties

Conducting plates can conduct heat in either direction and have no physical thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Conducting Plate Thermal Model** dialog box **General** tab contains the following general information for the selected boundary condition.

Note:

If needed, you can enter a temperature-dependent expression for **Total Power**. See Temperature Dependent Thermal Properties for more information.

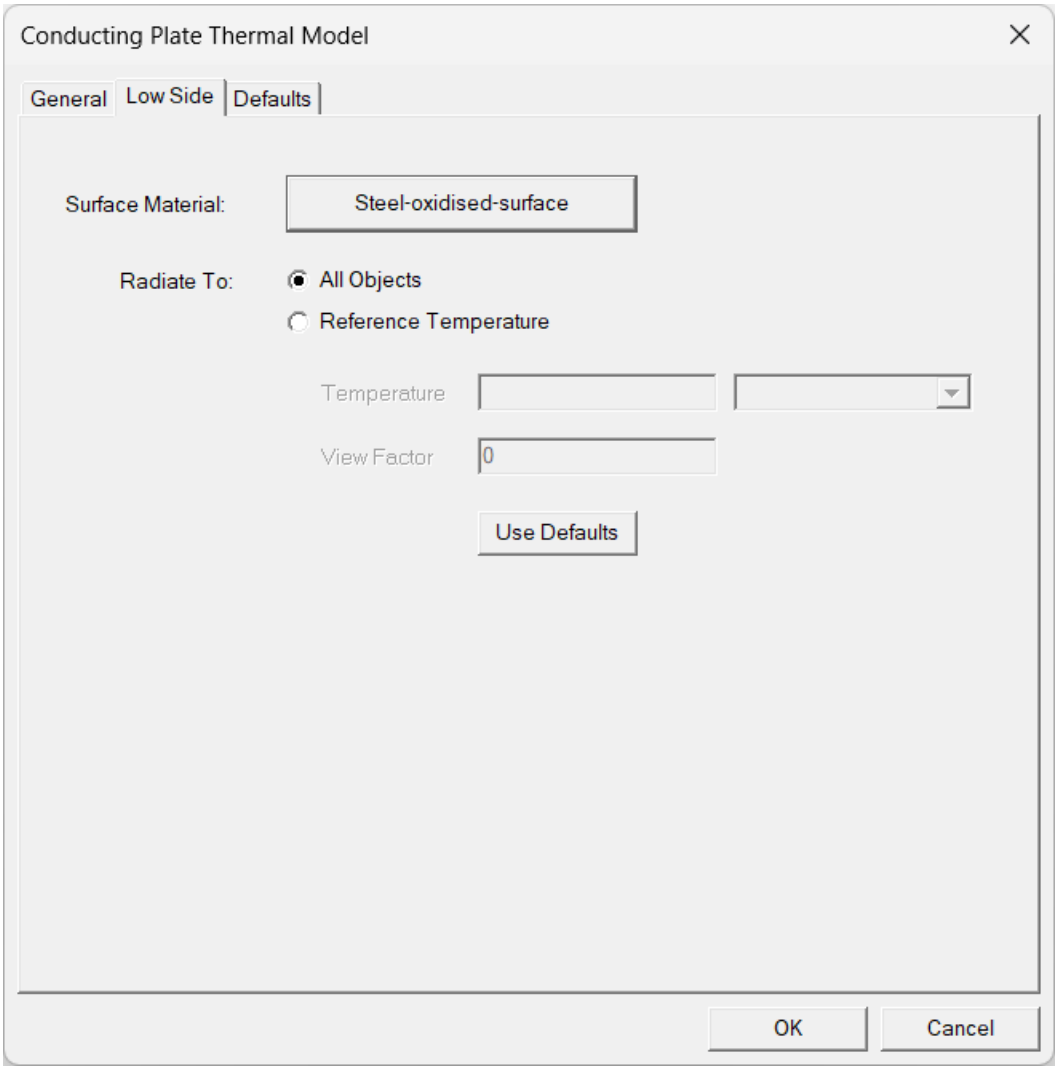
Name	The **Name** of the plate appears in the **Project Manager** window and **3D Modeler** window.
Thermal Specification: Thickness	**Thickness**specifies the effective thickness of the specified **Solid Material**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Specification: Conductance	**Conductance** specifies the value for the conductance. The inverse of this value is used to calculate the contact resistance of the plate.
Thermal Specification: Thermal Resistance	**Thermal Resistance** specifies the value for the thermal resistance to heat transfer.
Thermal Specification:Thermal Impedance	**Thermal Impedance** specifies the value for the thermal impedance. Icepak computes the thermal resistance of the plate as Z/A where Z is the thermal impedance of the plate and A is the area of the plate.
Total Power	**Total power** specifies the total power dissipated by the plate.
Solid Material (Thermal Specification: Thickness)	**Solid Material** specifies the material type of the plate using the **Select Definition** dialog box.
Enable Joule Heating	**Enable Joule Heating** allows you to model a contact resistance plate between conductor-conductor interfaces. Specify a **Contact Resistance** value and unit for heat transfer. Contact resistance represents barriers to current flow between two electrically conducting objects. Contact resistance is caused due to factors like surface imperfections or low contact pressures. The contact resistance is calculated by multiplying the specified value by its area. For example, a contact resistance of 0.7 mOhm specified on 4.0mm^2 is calculated as follows: $0.7e\text{-}3 * 4.0e\text{-}6 = 2.8e\text{-}9\ Ohm\text{-}m$.
Shell Conduction	**Shell Conduction** can be used to model thin sheets to switch on and off conjugate heat transfer on any plate.
Note: When **Native** Joule heating is enabled, shell conduction is disabled by default unless you deselect **Disable Shell Conduction for Joule Heating** on the Icepak Design Settings **Advanced** tab.	
Side Specification	**Low side** and **High side** allows you to specify different physical characteristics for each side of the plate.

If you select **Low side** and/or **High side**, the **Low Side**and **High Side** tabs contain the following thermal information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Surface Material	**Surface Material** specifies the material type for the side of the plate using the **Select Definition** dialog box.
Radiate To	Select **All Objects** or **Reference Temperature**. **All Objects** specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. **Reference temperature** specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Temperature** and **View factor**, which is the proportion of radiation that leaves the plate and contacts surrounding objects. The default view factor is 1.0 and the value of the ambient temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Conducting Plate and Defining Thermal Properties

To assign a conducting plate boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Plate** > **Conducting**.
4. On the **General** tab, enter a **Name** for the object.
5. Select a **Thermal Specification**. Do one of the following:
    - If you selected **Thickness**, enter the effective **Thickness**.
    - If you selected **Conductance**, enter the **Conductance**.
    - If you selected **Thermal Resistance**, enter the **Thermal Resistance**.
    - If you selected **Thermal Impedance**, enter the **Thermal Impedance**.
6. Enter the **Total Power**.
7. Click the **Solid Material** button to open the **Select Definition** dialog box. Select a material and click **OK**.
8. If needed, select **Enable Joule Heating** and enter a **Contact Resistance**.
9. If needed, select **Shell Conduction**.
10. If needed, under **Side Specification**, select **Low side** and/or **High side** to enable thermal specification of the low and/or high side of the plate.
11. Do one of the following:
    - If you did not select a **Side Specification** check box, click **OK**.
    - If you selected **Low side** and/or **High side**, click the **Low Side**and/or **High Side** tab.
12. Click the button next to **Surface Material** to open the **Select Definition** dialog box. On the **Materials** tab, select the material for the side of the plate and click **OK**.
13. For **Radiate To**, click **All Objects** or **Reference Temperature**.
14. If you selected **Reference Temperature**, enter a **Temperature** and **View Factor**.
15. If you selected both **Low side** and **High side**, repeat steps 9 through 12.
16. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Resistance Properties

A resistance is a three-dimensional modeling boundary condition that represents a partial obstruction to flow within the mesh region. To assign a resistance boundary condition, the selected geometry must be three-dimensional (for example, a box). Resistances can represent flow resistance due to components such as wires, cables, and insulation materials packed within a portion of the mesh region. The effect of any resistance is modeled as a pressure drop through its area or volume.

Pressure Drop Calculation for a 3D Resistance

There are three methods for modeling the pressure drop: Device/approach, power law, and loss curve. In each of the methods, Icepak calculates the velocity based on the thermal properties of the surrounding environment.

Device/Approach

$$\Delta p = \rho \left(\frac{l_c}{2} \right) \left(\frac{v}{A_{freearearatio}} \right)^n$$, where

- l_c is the user-specified device loss coefficient (for each coordinate direction in 3D)
- ρ is the fluid density
- v is the approach velocity
- $A_{freearearatio}$ is the ratio of the area through which the fluid can flow unobstructed to the total planar area of the obstruction
- n is the velocity dependence relationship where n = 1 denotes a linear relationship and n = 2 represents a quadratic relationship; pressure drop can be:
    - Linear - for a viscous flow regime (e.g., laminar flow, slow flow, very dense packing), where

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta p = \frac{\rho}{2} l_{c1} v$$

- Quadratic - for an inertial flow regime (e.g., turbulent flow), where

$$\Delta p = \frac{\rho}{2} l_{c2} v^2$$

- both linear and quadratic, where

$$\Delta p = \Delta p_{linear} + \Delta p_{quadratic}$$

Because the resistance to the fluid flow due to a volumetric resistance may be different in each of the three coordinate directions, you must provide the loss coefficient and free area ratio to calculate the pressure drop in each direction.

> **Note:**
>
> When using the **Device/Approach** pressure loss type, enter **Loss Coefficient** and **Free Area Ratio** values for **Linear** and/or **Quadratic** to specify linear, quadratic, or both. When specifying **Linear**, you must leave the **Quadratic** values as zero and vice versa.

Power Law

$\Delta p = C v^n$, where

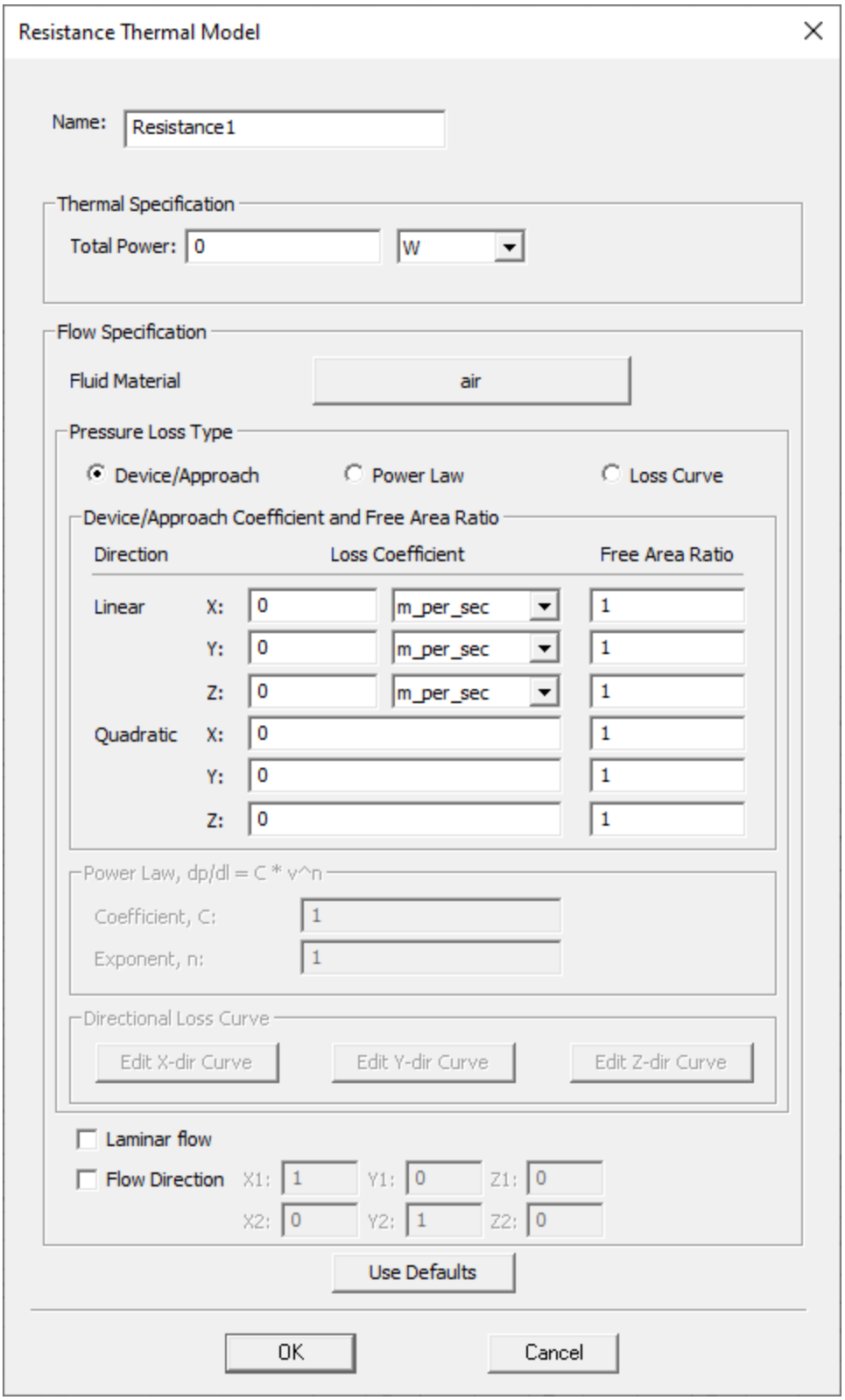
- Δp is the pressure drop across the 3D resistance
- v is the velocity
- C and n are constants

Pressure Loss Curve

Pressure loss curve is defined as a piecewise-linear profile for the pressure drop as a function of the speed of the fluid through the resistance in the three coordinate directions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Resistance Properties



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Resistance Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

> **Note:**
>
> If needed, you can define **Total Power** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

Name	The **Name** of the resistance appears in the **Project Manager** window and **3D Modeler** window.
Thermal Specification	
Total Power	The **Total Power** specifies the total power dissipated by the resistance.
Flow Specification	
Fluid Material	The **Fluid Material** specifies the fluid for the resistance. By default, it is set to water.
Pressure Loss Type: Device/Approach	
Loss Coefficient	The **Loss Coefficient**is used to calculate the pressure loss across the resistance for each coordinate direction in the case of a linear, quadratic, or linear+quadratic velocity dependence relationship.
Free Area Ratio	The **Free Area Ratio** specifies the ratio of the open area to the total area of the resistance.
Pressure Loss Type: Power Law ($dp/dl = C*v^n$)	
Coefficient, C	**Coefficent, C** is the coefficient used in the pressure drop calculation using the Power Law method. It is specified in SI units.
Exponent, n	**Exponent, n** is the exponent used in the pressure drop calculation using the Power Law method. It is specified in SI units.
Pressure Loss Type: Directional Loss Curve	
Edit X-, Y-, Z-dir Curve	Clicking these buttons displays the **Edit Dataset** dialog box for the associated coordinate direction. In the **Edit Dataset** dialog box, specify coordinates for velocity and pressure to create the pressure loss curve. See Adding Datasets for more information.
Laminar Flow	Enable **Laminar Flow** if the interior of the boundary condition is to be modeled as a laminar zone. This option is only enforced when a turbulent flow regime is selected in the **Icepak Solve Setup** dialog box.
Flow Direction	For some problems in which the principal axes of the resistance are not aligned with the coordinate axes of the domain, the flow direction can be specified by enabling **Flow Direction** and specifying two direction vectors. The third direction, which is not explicitly defined, is normal to the plan defined by the two specified direction vectors. The second

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	direction must be normal to the first. If you fail to specify two normal directions, the solver will ensure that they are normal by ignoring any component of the second direction that is in the first direction. You should therefore be certain that the first direction is correctly specified. **Note:** The **Flow Direction** is specified relative to the Global coordinate system.

Note:

When using the **Device/Approach** pressure loss type, enter **Loss Coefficient** and **Free Area Ratio** values for **Linear** and/or **Quadratic** to specify linear, quadratic, or both. When specifying **Linear**, you must leave the **Quadratic** values as zero and vice versa.

Assigning a Resistance and Defining Thermal Properties

To assign a resistance boundary condition and define its properties:

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Resistance**.
4. In the **Resistance Thermal Model** dialog box, enter a **Name** for the resistance.
5. Under **Thermal Specification**, enter a value for **Total Power**.
6. Under **Flow Specification**:
    - Click **Fluid Material** to open the **Select Definition** dialog box and select a material.
    - Select a **Pressure Loss Type** :
        - **Device/Approach**: Enter values for **Loss Coefficient** and **Free Area Ratio** for the linear and quadratic directions.
        - **Power Law**: Enter values for **Coefficient, C** and **Exponent, n**.
        - **Loss Curve**: Create pressure loss curves for the required coordinate directions.
7. If needed, enable **Laminar Flow**.
8. If needed, enable **Flow Direction** and enter values for each coordinate direction.
9. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

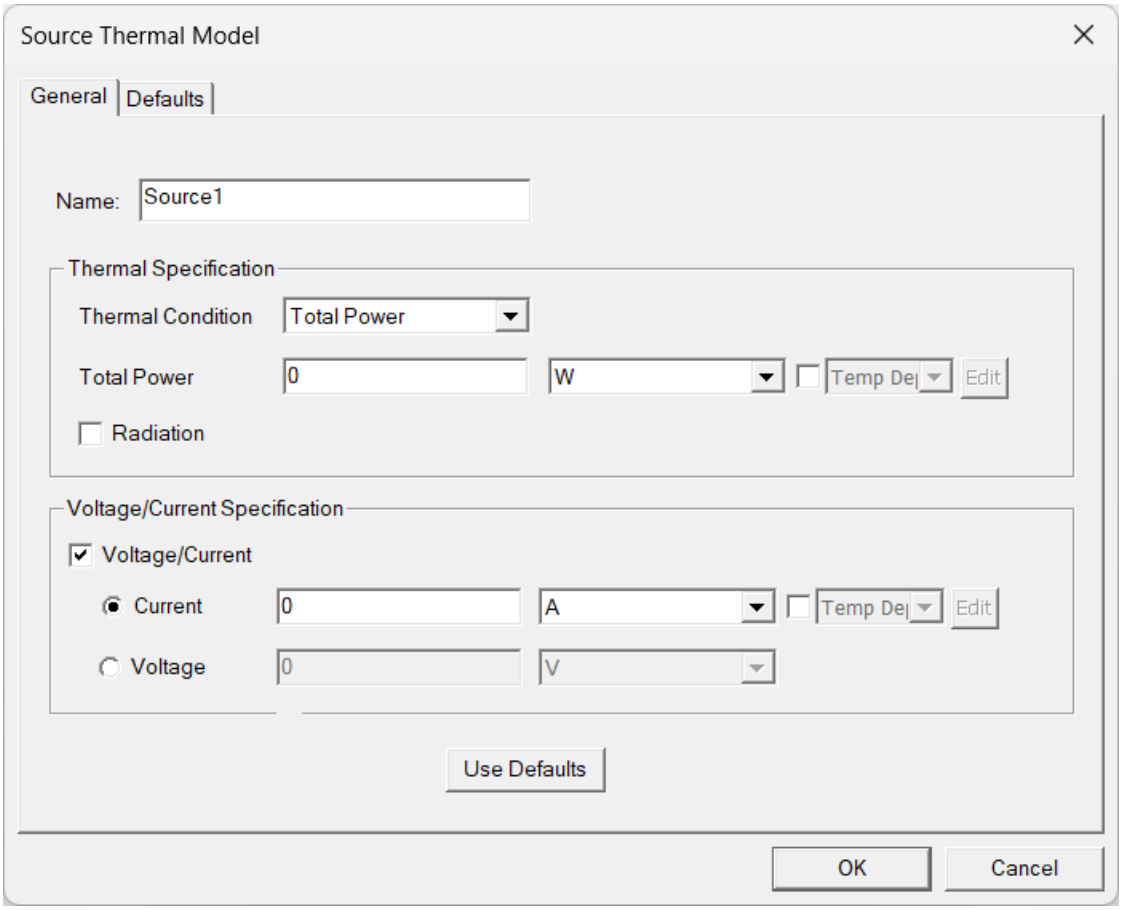
Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Source Properties

A source is a modeling boundary condition that represents a region in the model within which heat flux originates. Source geometries include rectangular, circular, 2D polygon, cylinder, and CAD. Sources can be used to specify the primary field variable of temperature. Sources can exchange radiation with other objects in the model.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Source Thermal Model** dialog box **General** tab contains the following general information for the selected boundary condition.

Note:

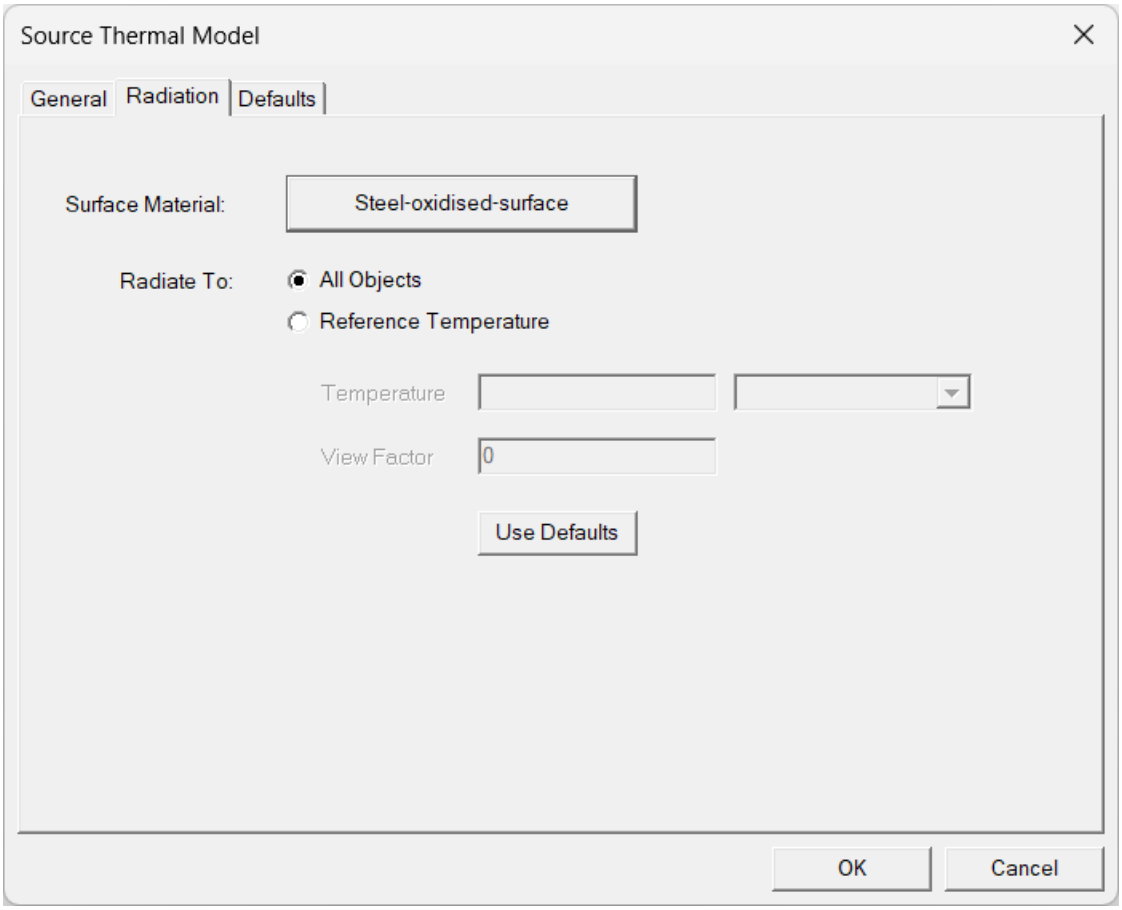
- If needed, you can enter a temperature dependent expression for **Total Power** and **Surface Flux**. See Temperature Dependent Thermal Properties for more information. For more information about specifying temperature-dependent power, see Using Datasets to Define Temperature Dependent Parameters.
- For more information on specifying transient parameters, see Specifying Parameters as a Function of Time.
- For more information on dynamic thermal modeling, see DTM - Thermostat for Source Control.
- If needed, you can define **Surface Flux** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

General	
Name	The **Name** of the source appears in the **Project Manager** window and **3D Modeler** window.
General: Thermal Specification	
Thermal Condition: Total Power	**Total Power** specifies the total power output over the plane of the source. If needed, select the check box and select **Temp Dep** to specify the temperature dependence of power. For transient simulations, select **Transient**, to specify power as a function of time.
Thermal Condition: Surface Flux	**Surface Flux** specifies the flux of heat per unit area of the source for a 2D source. If needed, select the check box and select **Temp Dep** to specify the temperature dependence of surface flux. For transient simulations, select **Transient**, to specify surface flux as a function of time.
Thermal Condition: Fixed Temperature	**Fixed Temperature** allows you to specify the temperature of the source. By default, the temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box. For transient simulations, select **Transient**, to specify temperature as a function of time.
Thermal Condition: DTM	**DTM** allows you to use the Dynamic Thermal Modeling feature for transient simulations. You can assign DTM to 2D source geometry. Dynamic Thermal Modeling allows you to specify a temperature-dependent power after each time step of a transient simulation.
Radiation	**Radiation** enables the specification of radiation as an active mode of heat transfer to and from the source.
General: Voltage/Current Specification	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Voltage/Current	**Voltage/Current** allows you to specify a voltage or current for the source. Select **Current** or **Voltage** and enter the appropriate value. For steady-state simulations, if needed, select the check box and select **Temp Dep**. For **Temp Dep**, click **Edit** to specify the temperature dependence of voltage or current. For transient simulations, select **Temp Dep**, **Transient**, or **DTM**. **Transient** allows you to specify voltage or current as a function of time. **DTM** (dynamic thermal modeling) allows you to specify a temperature-dependent voltage or current after each time step of a transient simulation.

If you select **Radiation**, the **Radiation** tab contains the following thermal information.



Surface Material	**Surface Material** specifies the material type for the side of the plate using the **Select Definition** dialog box.
Radiate To	Select **All Objects** or **Reference Temperature**. **All Objects** specifies that the object or surface exchanges radiation with all other objects and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	surfaces also specified as subject to radiation with heat transfer. **Reference temperature** specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Temperature** and **View factor**, which is the proportion of radiation that leaves the plate and contacts surrounding objects. The default view factor is 1.0 and the value of the ambient temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box.

Assigning a Source and Defining Thermal Properties

To assign a source boundary condition and define its properties:

> **Note:**
>
> When selecting geometry in the **3D Modeler**, ensure the appropriate selection mode is enabled.
>
> - For two-dimensional geometry, press F to enable face-selection mode.
> - For three-dimensional geometry, press O to enable object-selection mode.

1. Select and right-click a geometry component in the **3D Modeler** window or the history tree.
2. From the right-click menu, select **Assign Thermal**>**Source**.
3. On the **General** tab, enter a **Name** for the source.
4. Under **Thermal Specification**, select a **Thermal Condition**. Do one of the following:
   - If you selected **Total Power**,enter the **Total Power**.
   - If you selected **Surface Flux**, enter the **Surface Heat**.
   - If you selected **Fixed Temperature**, enter the **Temperature**.
   - If you selected **DTM**, ensure you specify the dynamic thermal modeling control program script in the Icepak Solve Setup dialog. To create the script file, use the Thermostat for Source Control toolkit.
5. If needed, select **Radiation**.
6. If needed, enable **Voltage/Current**, select **Current** or **Voltage**, and enter a value. For steady-state simulations, if needed, enable the check box and select **Temp Dep**. For transient simulations, if needed, enable the check box and select **Transient** or **DTM**. Define a variation function for **Temp Dep** or **Transient**.
7. Do one of the following:
   - If you did not select the**Radiation** check box, click **OK**.
   - If you selected **Radiation**, click the **Radiation** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the next dialog box next to **Surface Material**, click **vacuum** to open the **Select Definition** dialog box. On the **Materials** tab, select the material for the side of the plate and click **OK**.
9. For **Radiate To**, click **All Objects** or **Reference Temperature**.
10. If you selected **Reference Temperature**, enter a **Temperature** and **View Factor**.
11. Click **OK**.

> **Note:**
>
> The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:
>
> - **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
> - **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Wall Properties

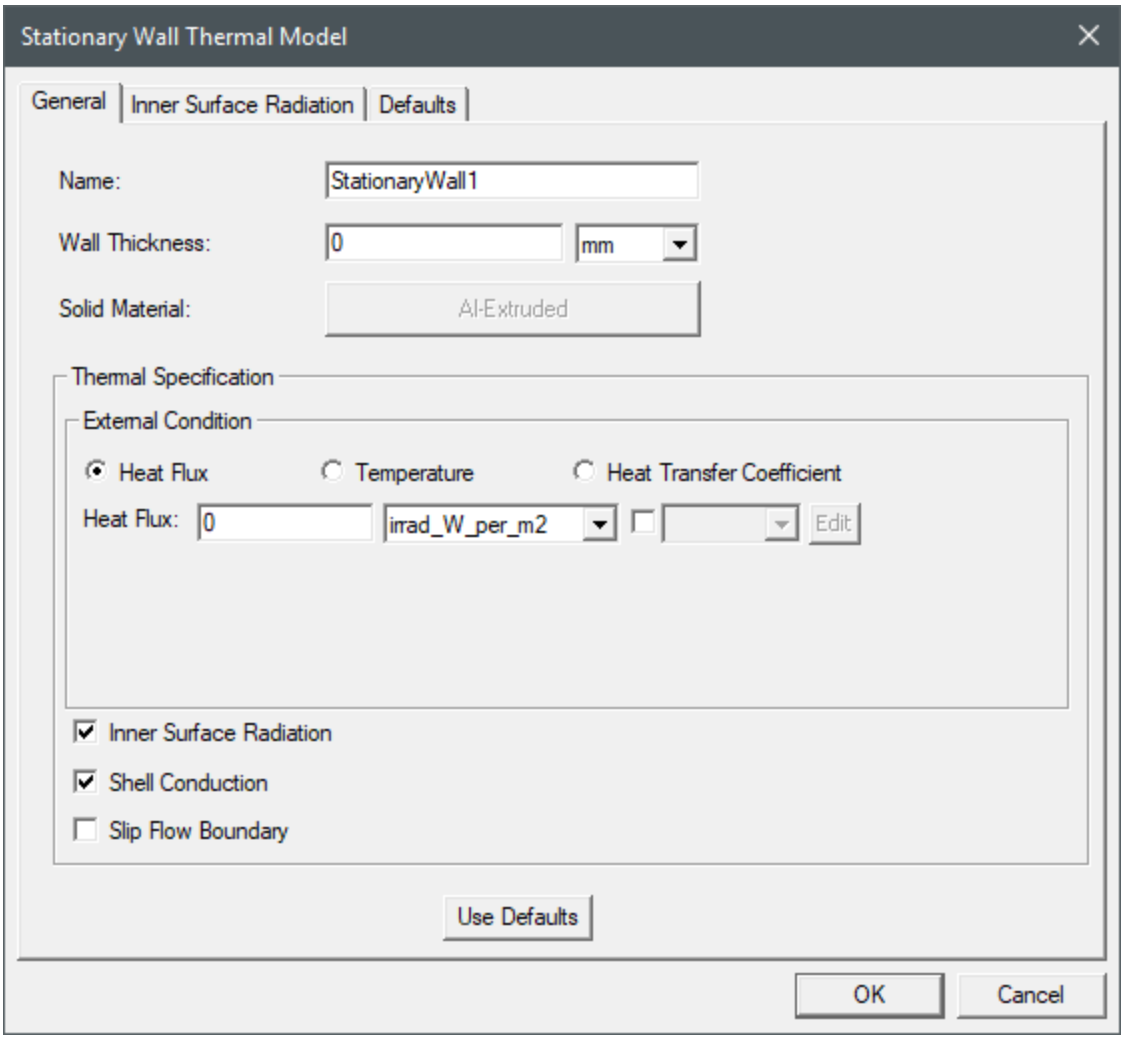
A wall is a two-dimensional modeling boundary condition that represents all or part of the mesh region boundary. Walls can be specified with respect to heat flux, temperature, or heat transfer coefficient. To assign a wall boundary condition, the selected geometry must be two-dimensional (for example, a face of the mesh region).

Throughout this section, reference will be made to the inside and outside of a wall. The inside of the wall is the side in contact with the fluid in the mesh region; the outside of the wall is the side exposed to the conditions external to the mesh region. For a wall with non-zero effective thickness, the inner and outer sides of the wall coincide; however, the inner and outer surface materials can be different.

Stationary Wall Properties

A stationary wall is a non-moving wall. For stationary walls, the flow velocity is zero at the wall.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Stationary Wall Thermal Model** dialog box **General** tab contains the following general and thermal information for the selected boundary condition.

Note:

- If needed, you can enter a temperature dependent expression for **Heat Transfer Coefficient**. See Temperature Dependent Thermal Properties for more information.
- If needed, you can define **Temperature**, **Heat Transfer Coefficient**, and **Heat Flux** as a spatial, or volumetric, profile using an expression and a 3D dataset. See Adding 3D Datasets for more information.

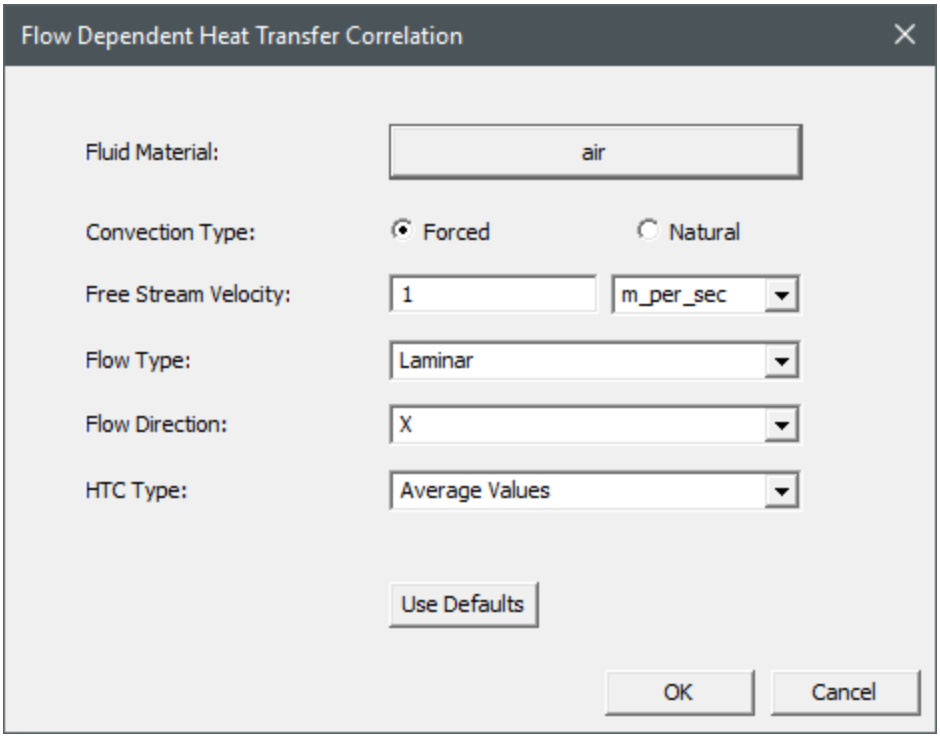
Name	The **Name** of the wall appears in the **Project Manager** window

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	and **3D Modeler** window.
Wall Thickness	**Wall Thickness** specifies the effective thickness of the wall.
Solid Material	**Solid Material** specifies the material type for the wall using the **Select Definition** dialog box.
Thermal Specification	
External Condition: Heat Flux	**Heat Flux** specifies a uniform fixed rate of heat transfer through the wall.
External Condition: Temperature	**Temperature** specifies a uniform fixed temperature on the outer surface of the wall.
External Condition: Heat Transfer Coefficient	**Heat Transfer Coefficient** specifies convective heat transfer at the outer surface of the wall. The heat transfer coefficient can be set automatically using existing empirical Nusselt correlations for forced convection and natural convection flows. These correlations depend on the geometry of the wall, fluid properties, Reynolds number (for forced convection flows), and Rayleigh number (for natural convection flows). To specify the heat transfer coefficient using correlations select **Use Correlation** and click **Edit Correlation** to open the **Flow Dependent Heat Transfer Correlation** dialog box. To specify external surface radiation, select **External Surface Radiation** and click **Edit Parameters** to open the **External Surface Radiation** dialog box.
Inner Surface Radiation	**Inner Surface Radiation** allows you to specify the default radiation characteristics of the inner surface of the wall.
Slip Flow Boundary	**Slip Flow Boundary** allows you to set the only the normal component of the velocity to zero at the wall. The flow velocity at the wall is always set to zero (that is, a no-slip boundary condition). For models where the no-slip condition is not appropriate (for example, in flows where the static pressure is very low), you can enable a slip flow boundary

Note: If radiation is enabled but **Inner Surface Radiation** is disabled, then the emissivity default is used in the calculation even if **Inner Surface Radiation** is disabled.

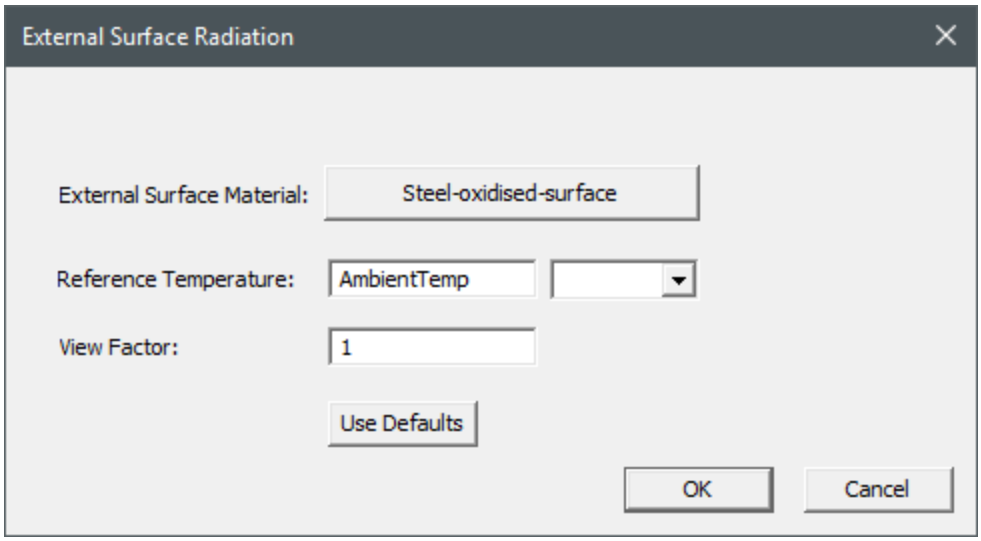
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you select **Use Correlation** for **Heat Transfer Coefficient** and click **Edit Correlation**, the **Flow Dependent Heat Transfer Correlation** dialog box contains the following thermal information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

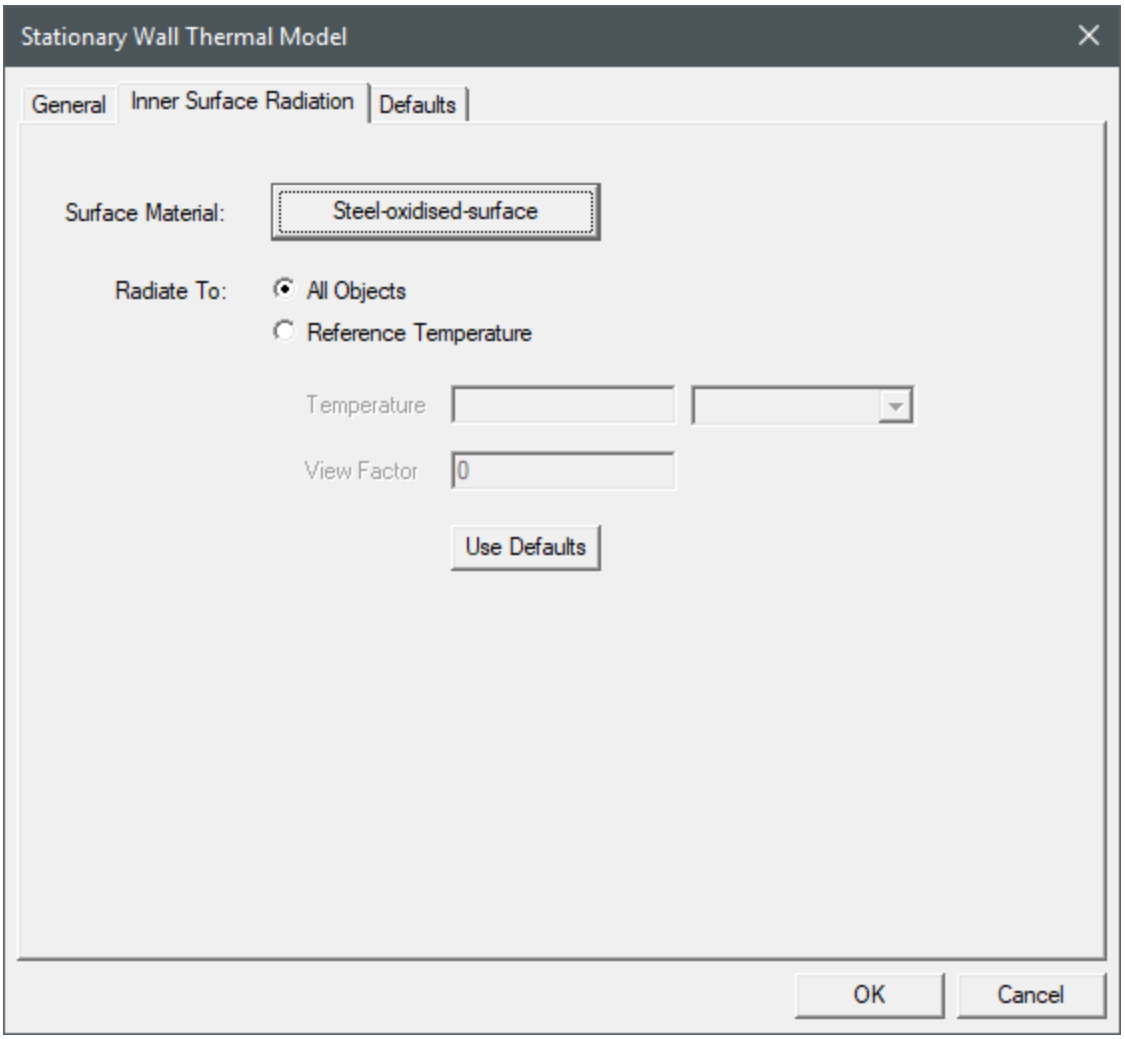
Fluid Material	**Surface Material** specifies the fluid material type for the external flow using the **Select Definition** dialog box.
Convection Type	Select a **Convection Type** of **Forced** or **Natural** to define the heat transfer mode.
Forced Convection	
Free Stream Velocity	**Free Stream Velocity** specifies the external flow velocity.
Flow Type	**Flow Type** specifies a **Laminar** or **Turbulent** external flow.
Flow Direction	**Flow Direction** specifies the direction of the external flow.
HTC Type	**HTC Type** specifies the type of the heat transfer coefficient used. For forced convection situations, there are empirical correlations to define a local value of the heat transfer coefficient. To specify an average heat transfer coefficient for the entire wall surface, select **Average Values**. To specify a locally varying heat transfer coefficient, select **Local Values**.
Natural Convection	
Ambient Temperature	**Ambient Temperature** specifies the ambient external temperature for the fluid surrounding the wall.
Surface	**Surface** specifies the orientation of the surface with respect to gravity. The heat transfer coefficients for natural convection flow depend on the orientation of the wall surface relative to the direction of gravity. To specify the orientation of the surface, select one of the following three options: **Vertical**, **Top**, and **Bottom**.



If you select **External Surface Radiation** for **Heat Transfer Coefficient** and click **Edit Parameters**, the **External Surface Radiation** dialog box contains the following thermal information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Surface Material	**External Surface Material** specifies the material type for the external sides of the wall using the **Select Definition** dialog box.
Reference Temperature	**Reference Temperature** specifies a fixed temperature to which the selected sides of the object radiate heat.
View Factor	**View Factor** is the proportion of radiation that leaves the plate and contacts surrounding objects.



If you select **Inner Surface Radiation**, the **Inner Surface Radiation** tab contains the following thermal information.

Surface Material	**Surface Material** specifies the material type for the side of the wall using the **Select Definition** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiate To	Select **All Objects** or **Reference Temperature**. **All Objects** specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. **Reference Temperature** specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Temperature** and **View Factor**, which is the proportion of radiation that leaves the plate and contacts surrounding objects. The default view factor is 1.0 and the value of the ambient temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box.

Assigning a Stationary Wall and Defining Thermal Properties

To assign a stationary wall object type and define its properties:

1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal**>**Wall** > **Stationary**.
4. On the **General** tab, enter a **Name** for the wall.
5. Enter the **Wall Thickness**.
6. Select a **Solid Material**.
7. Select a **Thermal Specification**.
   - If you selected **Heat Flux**, enter a value for **Heat Flux**.
   - If you selected **Temperature**, enter a value for **Temperature**.
   - If you selected **Heat Transfer Coefficient**, enter a value for **Heat Transfer Coefficient**. If needed, select **Use Correlation**, click **Edit Correlation**, and specify the correlation settings in the **Flow Dependent Heat Transfer Coefficient** dialog box. If needed, select **External Surface Radiation**, click **Edit Parameters**, and specify the radiation settings in the **External Surface Radiation** dialog box.
10. If needed, select **Inner Surface Radiation**.
11. If needed, select **Shell Conduction**.
12. If needed, select **Slip Flow Boundary**.
13. Do one of the following:
    - If you did not select **Inner Surface Radiation**, click **OK**.
    - If you selected **Inner Surface Radiation**, click the **Inner Surface Radiation** tab.

13. In the **Inner Surface Radiation** tab, click **Surface Material** to open the **Select Definition** dialog box. On the **Materials** tab, select the material for the side of the plate and click **OK**.
14. For **Radiate To**, click **All Objects** or **Reference Temperature**.
15. If you selected **Reference Temperature**, enter a **Temperature** and **View Factor**.
16. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Wall Heat Transfer Coefficient Correlation Formulas

The following are heat transfer coefficient correlation formulas for forced convection and natural convection scenarios.

Forced Convection

Laminar Flow

$$h_x = \left[\frac{0.332k\mathrm{Re}_x^{0.5}\mathrm{Pr}^{0.3333}}{x}\right] h_{avg} = \left[\frac{0.664k\mathrm{Re}_L^{0.5}\mathrm{Pr}^{0.3333}}{L}\right]$$

where:

k is the thermal conductivity of fluid

Re is the Reynolds number of the flow

Pr is the Prandtl number of the fluid

L is the length of the surface in the direction of the flow

Turbulent Flow

$$h_x = \left[\frac{0.0296k\mathrm{Re}_x^{0.8}\mathrm{Pr}^{0.3333}}{x}\right] h_{avg} = \left[\frac{0.037k\mathrm{Re}_L^{0.8}\mathrm{Pr}^{0.3333}}{L}\right]$$

where:

k is the thermal conductivity of fluid

Re is the Reynolds number of the flow

Pr is the Prandtl number of the fluid

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

L is the length of the surface in the direction of the flow

Natural Convection

Vertical Wall (Side Wall)

$$h_{avg} = \frac{k}{L}\left[0.825 + \frac{0.387Ra_L^{(1/6)}}{\left[1+(0.492/\mathrm{Pr})^{(9/16)}\right]^{(8/27)}}\right]^2$$

where:

k is the thermal conductivity of fluid

L is the length of the surface in the direction of the flow

Ra is the Raleigh number of the flow

Pr is the Prandtl number of the fluid

Top Wall

For $Ra_L < 10^4$: $h_{avg} = 0.425\left[\frac{\Delta T}{L}\right]^{0.33}$

For $10^4 \le Ra_L \le 10^7$: $h_{avg} = \left[\frac{0.54kRa_L^{0.25}}{L}\right]$

For $10^7 \le Ra_L \le 10^{11}$: $h_{avg} = \left[\frac{0.15kRa_L^{0.3333}}{L}\right]$

where:

Ra is the Raleigh number of the flow

L is the length of the surface in the direction of the flow

T is the surface temperature

k is the thermal conductivity of fluid

Bottom Wall

For $Ra_L < 10^5$: $h_{avg} = 0.83(\Delta T)^{0.33}$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where:

Ra is the Raleigh number of the flow

L is the length of the surface in the direction of the flow

T is the surface temperature

k is the thermal conductivity of fluid

Symmetry Wall Properties

The symmetry wall boundary condition allows you to model half of a mesh region in the case where the mesh region is geometrically symmetric about a center plane. The velocity component normal to a symmetry wall is set to zero. When you assign a symmetry wall, you cannot specify any thermal data and Icepak automatically specifies a zero heat flux condition at the symmetry plane.

There are no additional inputs for a symmetry wall.

Assigning EM Loss Properties

Data can be linked from an HFSS, Maxwell, or Q3D design to an EM Loss thermal boundary condition in an Icepak design. You can assign multiple EM Loss boundary conditions that are linked from the same source design.

- In HFSS, surface loss exists on non-solve-inside conducting objects, and volume loss exists on solve-inside lossy objects.
- In Maxwell, surface loss exists on Impedance Boundaries, and volume loss usually exists on solve-inside conducting objects.
- In Q3D, volume loss exists in conductive objects for DC RL field type, and surface loss exists on conductive objects for the AC RL field type. Volume loss can be a DC loss or a harmonic loss, which is a post-processing field for the DC RL analysis.

> **Note:**
>
> Lossy objects are those whose material definition includes a loss tangent greater than zero.

You can couple volume or surface loss into an Icepak design by assigning EM Loss boundary conditions to geometry objects. During the Icepak simulation process, EM loss data is mapped onto the mesh and consumed into the solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

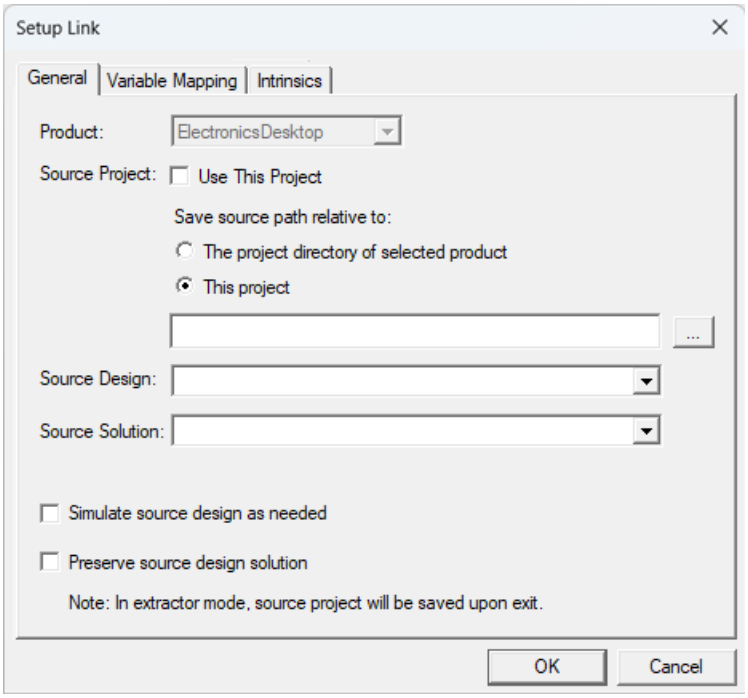
Note:

A summary of coupled EM Loss data can be viewed on the **Solutions** dialog **Profile** tab during the solution process. In the **Project Manager**, expand **Analysis**, right-click the solution setup, and select **Profile**.

Assign EM Loss Properties

1. Select one or more geometry in the **3D Modeler** window or history tree and select **EM Loss**.

   The **Setup Link** dialog box is displayed.

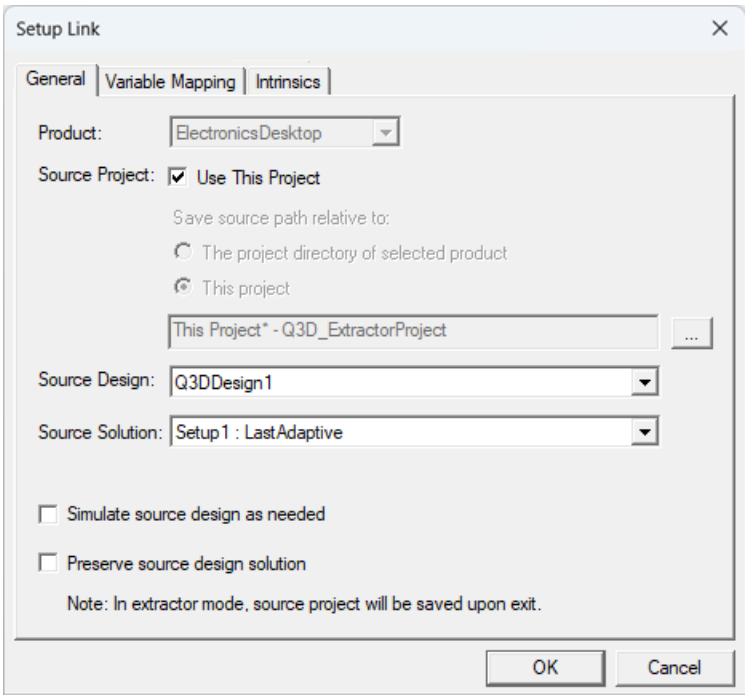


2. Specify the project that has the source design in one of the following ways:
   a. Check **Use This Project** to pick a design from a current working project.
   b. Click the ellipsis and browse your computer to pick the design.

Note:

The option **Use this project** points to the directory relative to where your current working project is located.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

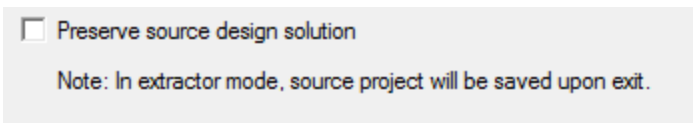


3. Pick the source design from the **Source Design** drop-down menu which lists all the designs in the specified project.
4. Pick the setup type from the **Source Solution** pull-down menu (e.g., Last Adaptive, Sweep).
5. Select the last two check boxes as needed and click **OK**.

> **Note:**
>
> Go to Simulate source design as needed, for more information about that option.

Preserve source 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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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.

Simulate source design as needed

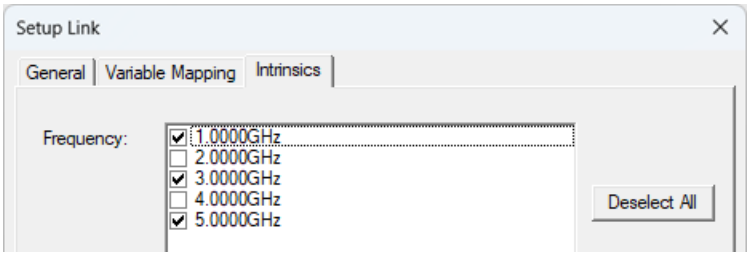
Select this option if you modify the source design and you want an updated source design solution to be linked to the target design. If you select this option, make sure that you also invalidate the cached link data in the target. To do this, right-click **Analysis** and select the option **Clear Linked Data** from the short-cut menu.



6. Click the **Variable Mapping** tab and ensure the **Variable Mapped to Source Design** values are as desired for each variable.

> **Note:**
>
> You can click **Map Variable by Name** to compare the source design variables to the Icepak design variables. Any variables with the same name are mapped so that the source design conforms to the Icepak design.

7. For HFSS designs (**Modal** and **Terminal** solution types only) and Maxwell designs (**AC Magnetic** solutions), click the **Intrinsics** tab and select a **Frequency**, which defines the frequency used in the source design. 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 Icepak design. (See *Important* note below.)

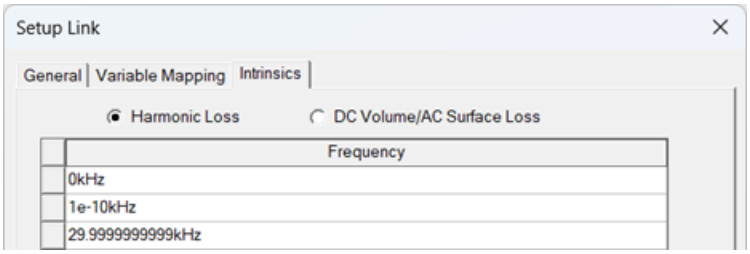


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

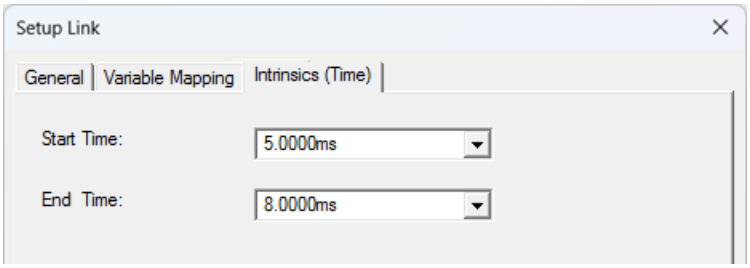
Multiple frequencies are available from HFSS designs (**Modal** and **Terminal** solution types only) and Maxwell 3D designs (**AC Magnetic** solutions), providing that you include a frequency sweep and choose to save the fields for all frequencies.

> **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 Icepak design with only one frequency at a time selected under the *Intrinsics* tab.

8. For Q3D designs, click the **Intrinsics** tab and select **Harmonic Loss** or **DC Volume/AC Surface Loss**.



9. For Maxwell transient solutions (**Magnetic** or **Electric**), click the **Intrinsics (Time)** tab and select the **Start Time** and **End Time** from the drop-down menus.

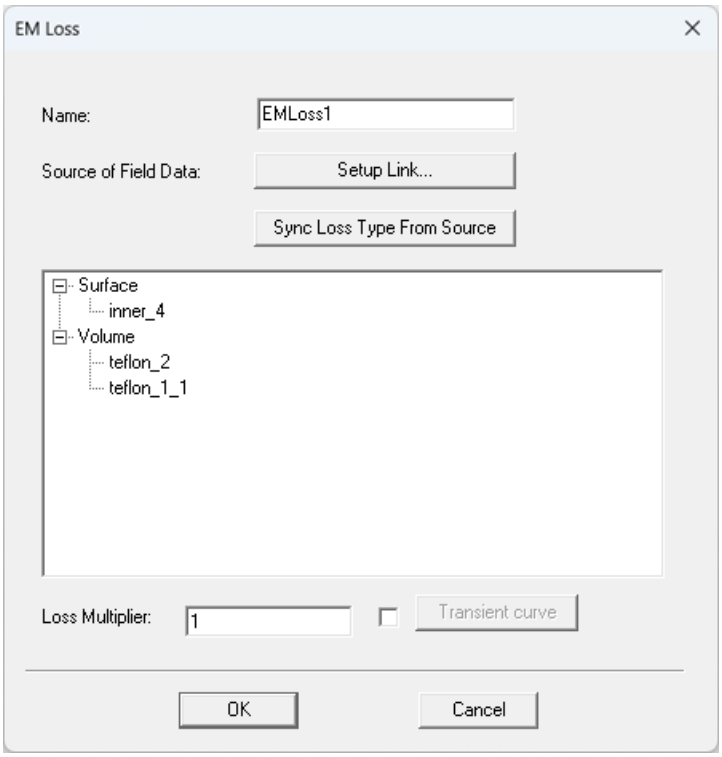


> **Note:**
>
> The Icepak solution is based on the total energy loss in the source design during the period you specify in the **Intrinsics (Time)** tab divided by the same specified time period. This quotient is the average power loss that will be applied to the target solution as a constant EM Loss.

10. Click **OK** to close the **Setup Link** dialog. The **EM Loss** dialog box appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Enter a **Name** for the EM loss boundary.



12. Drag and drop the geometry under **Surface** or **Volume** to specify it as either surface or volume loss.
13. If needed, enter a **Loss Multiplier** value. A loss multiplier can be applied to increase or decrease the EM losses applied during the Icepak simulation. For example, a loss multiplier value of 1.2 could be used to build in a safety factor of 20% into the thermal solution.
14. If needed, enable **Transient curve** to define the loss multiplier as function of time. The **Transient Multiplier - Transient Variation** dialog box opens. See Using Datasets to Define Transient Parameters for more information.
15. Click **OK** to close the **EM Loss** dialog box. The EM loss assignments appear under **Thermal** in the **Project Manager**.

> **Note:**
>
> To edit the EM loss link, click **Setup Link** to open the **Setup Link** dialog box.
>
> If the source HFSS, Maxwell, or Q3D design is updated, you can click **Sync Loss Type From Source** to update the link.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

To determine the total volume and surface EM loss values in the source design, use the Fields Calculator in the source design by choosing the appropriate objects. EM losses from the source design listed in the solution profile may differ from those assigned in Icepak due to differences in meshing or geometry translations. Source and target values match when scaling factors are close to one.

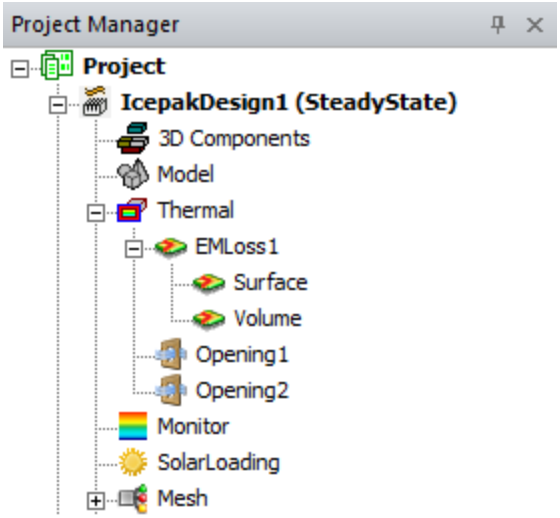
Note:

To produce accurate results, mesh in HFSS and Icepak should be similar. To evaluate mesh similarity between the source design and Icepak design, check the scaling factor in the Icepak solution profile and ensure the value is close to 1.0.

Task	Real Time	CPU Time	Memory	Information
Get Thermal Loss	00:01:10	00:00:00	0 K	Source: This Project*, Maxwell3DDesign1 - Setup1 : LastAdaptive
				Freq='50kHz'
				Total Loss on Source Object 'plate': 1.44024W, Scaling Factor: 1.02732
				Total Loss on Source Object 'coil': 2.41865W, Scaling Factor: 1.00034.

Modifying an EM Loss Boundary Condition

EM Loss boundary conditions can contain assignments of surface loss, volume loss, or both. The **Project Manager** displays the EM Loss boundary condition and, when expanded as in the following image, the volume and/or surface loss assignments.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting the boundary condition (e.g., EMLoss1 in the image above) selects both surface and volume assignments. Selecting either Surface or Volume selects the surface or volume assignments, respectively.

Reassign a Surface or Volume Assignment

You can reassign surface or volume assignments to other geometry.

1. In the **3D Modeler** window or history tree, select the geometry to which you want to reassign the surface or volume assignment.
2. In the **Project Manager** under **Thermal**, right-click the surface or volume assignment.
3. From the right-click menu, select **Reassign**. The surface or volume assignment is now assigned to the selected geometry.

Add to a Surface or Volume Assignment

You can add geometry to a surface or volume assignment.

1. In the **3D Modeler** window or history tree, select the geometry you want to add to the EM loss boundary condition.
2. In the **Project Manager** under **Thermal**, right-click the EM loss boundary condition.
3. From the right-click menu, select **Add to Surface** or **Add to Volume**. The selected geometry is added to the EM loss boundary condition.

Remove from an EM Loss Assignment

You can remove geometry from an EM loss assignment.

1. In the **3D Modeler** window or history tree, select the geometry you want to remove from the EM loss boundary condition.
2. In the **Project Manager** right-click **Thermal**, and select **Remove from Assignment**.

Delete an EM Loss Boundary Condition or Assignment

Right-click either the boundary condition or one of its assignments and select **Delete**.

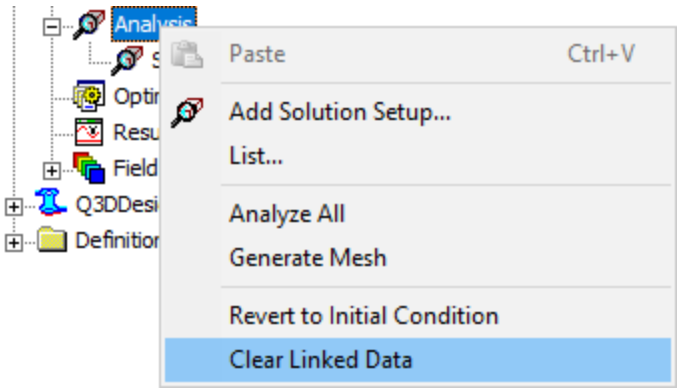
- Selecting **Delete** from the boundary condition right-click menu deletes boundary condition and all its assignments.
- Selecting **Delete** from a surface or volume assignment deletes only that assignment if more than one assignment exists.

Clearing Linked Data

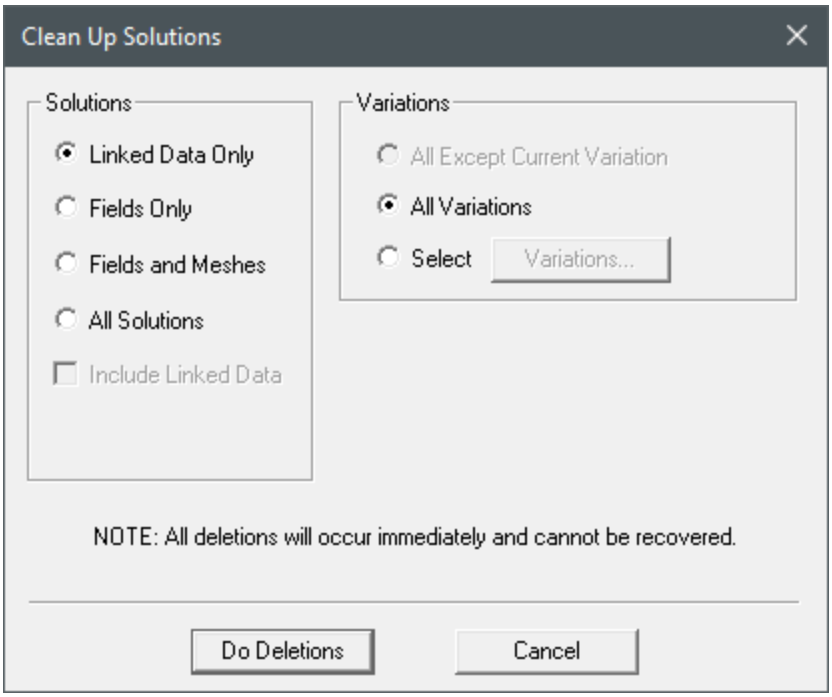
Data linked from other design types (e.g., EM Loss data, metal fraction data) can be removed from the design containing the data. There are two ways to clear the linked data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Project Manager**, right-click **Analysis** and select **Clear Linked Data**.



- From the **Icepak** menu, select **Results** > **Clean Up Solutions**. In the **Clean Up Solutions** dialog box, select **Linked Data Only** or another option and **Include Linked Data**. Then click **Do Deletions**.



Assigning Blower Properties

A blower is a three-dimensional modeling boundary condition. To assign a blower boundary condition, select two or three pieces of two-dimensional geometry (for example, a sheet). A blower boundary condition can be used to represent a type 1 blower (an impeller) or a type 2

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

blower (centrifugal blower). To assign a type 1 blower, one of the selected pieces of geometry must be a curved face.

Often used in conjunction with heat sinks, blowers are similar to 3D fans, but instead allow air to be expelled in a direction perpendicular to the incoming flow. They may have single or dual inlets, and are composed of at least two openings and either a hollow block or a pair of walls.

Impellers - Type 1 Blowers

Impellers are designed to model motorized impellers. They consist of two aligned circular wall objects that are separated by a ring-shaped air-outlet opening and either one or two air-inlet openings. Air is drawn into the impeller though the opening(s) on the top and/or bottom of the device and is then redirected 90° before being expelled circumferentially into the cabinet.

Centrifugal Blowers - Type 2 Blowers

Centrifugal blowers consist of an air-outlet opening and either one or two air-inlet openings. Air is drawn into the blower though the inlet opening(s) and is then redirected 90° C. The flow is then concentrated in one direction though a single circular or rectangular outlet opening.

Specifying Swirl

To specify the swirl for an impeller, you must specify the impeller blade angle and the rotational speed (RPM) of the impeller blades. The tangential air velocity, u_{tan} , changes as the blower operating point changes on the blower curve:

$$u_{tan} = \omega r - \frac{u_{rad}}{\tan \beta}$$

where ω is the angular velocity of the blades,

$$\omega = N \times \frac{2\pi}{60}$$

and

$$u_{rad} = \frac{Q}{2\pi r h}$$

where

h = height of impeller blades (case height)

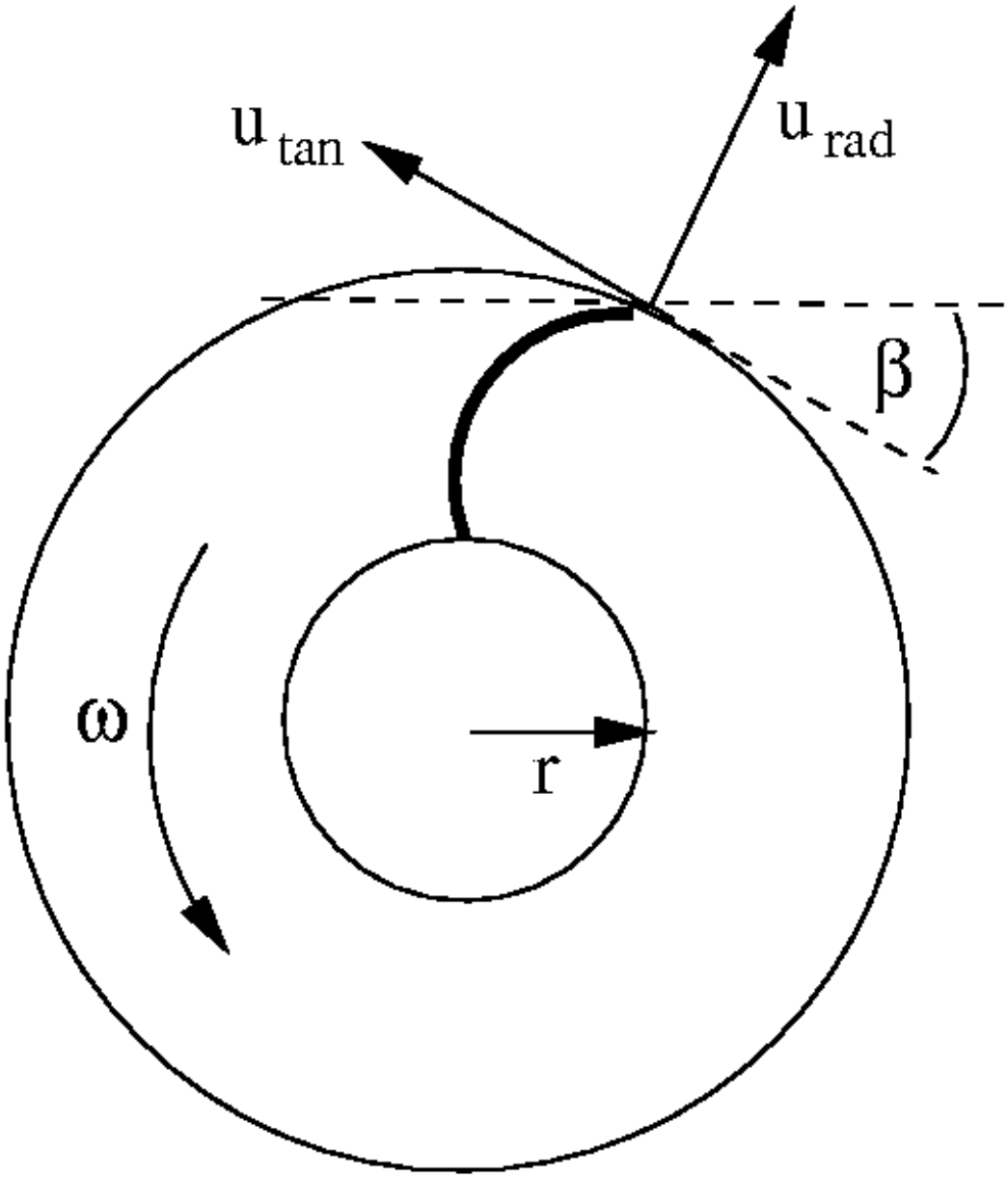
r = radius of inlet opening

N = RPM of impeller blades

Q = Volumetric flow rate (from characteristic curve)

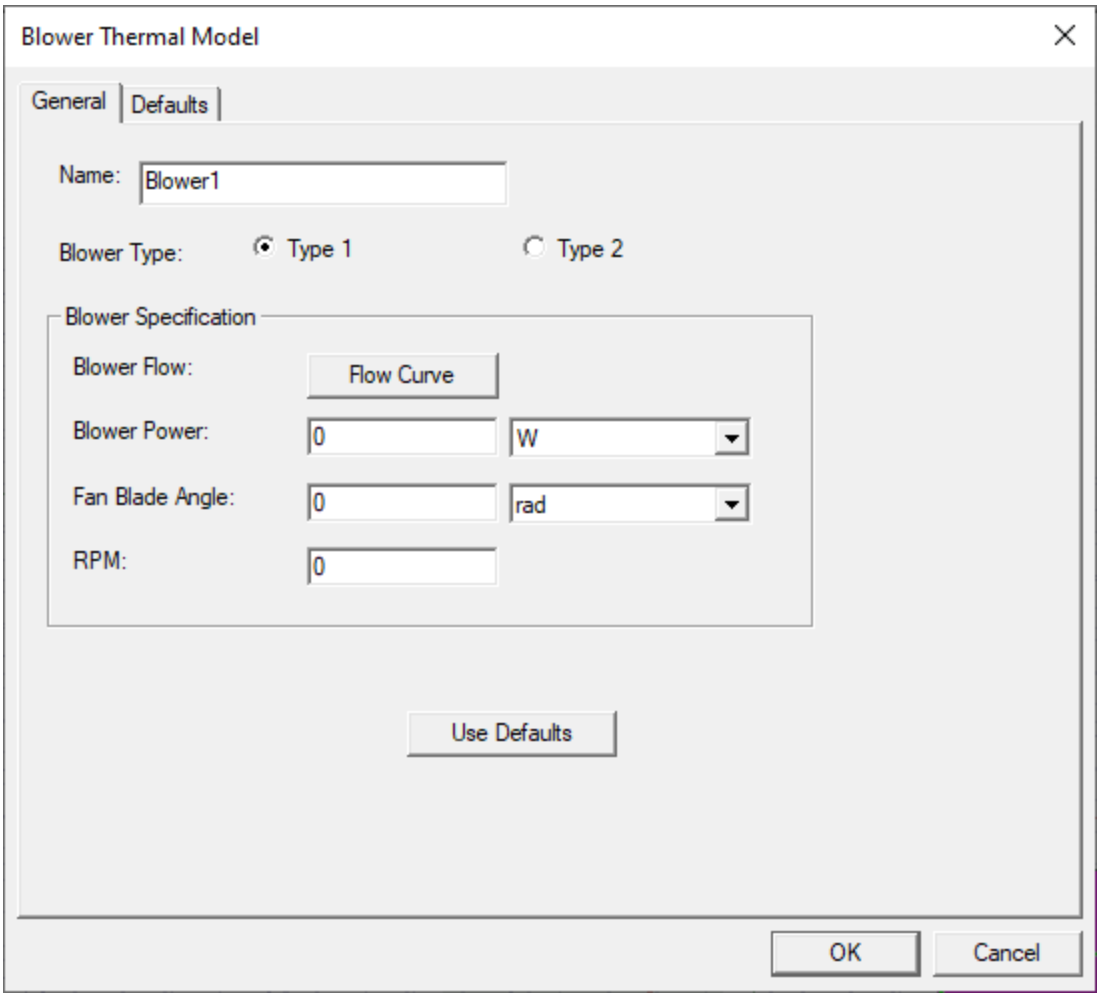
β = impeller blade angle measured with respect to ω

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Blower Properties



The **Block Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Blower Type	The **Blower Type** specifies whether the blower is **Type 1** (impeller) or **Type 2** (centrifugal blower).
Blower Specification	
Blower Flow	**Blower Flow** specifies the flow based on a specified flow curve. Click **Flow Curve** to open the **Edit Dataset** dialog box and enter **Velocity** and **Pressure** data to create the flow curve. See Adding Datasets for more information.
Blower Power	**Blower Power** specifies the total power dissipated by the blower.
Fan Blade Angle (Type 1)	**Fan Blade Angle** specifies the angle of the blower's fan blade for a type 1 blower.
RPM (Type 1)	**RPM** specifies the revolutions per minute of the blower fan for a type 1 blower.
Exit Exhaust Angle (Type 2)	**Exit Exhaust Angle** specifies the angle of the exhaust flow for a type 2 blower.

Assigning a Blower and Defining Thermal Properties

To assign a blower boundary condition and define its properties:

1. Press F to enable face-selection mode.
2. Select two or more pieces of two-dimensional geometry and right-click in the **3D Modeler** window or the history tree.

> **Note:**
>
> To assign a Type 1 blower, ensure that one of the selected pieces of geometry is curved. The curved face is automatically assigned as the blower outlet. The planar faces are assigned as the inlet(s).
>
> For Type 2 blowers, the face selected first is assigned as the outlet, and the subsequently selected faces are assigned as the inlet(s).

3. From the right-click menu, select **Assign Thermal**>**Blower**.
4. In the **Blower Thermal Model** dialog box, enter a **Name** for the block.
5. For **Blower Type**, select **Type 1** (impeller) or **Type 2** (centrifugal blower).
6. Under **Blower Specification**, do the following:
   - Click **Flow Curve** and enter **Velocity** and **Pressure** values in the **Edit Dataset** dialog to specify a flow curve.
   - Enter a **Blower Power** for the blower and select the desired thermal unit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For type 1 blowers, specify a **Fan Blade Angle**.
- For type 1 blowers, specify the **RPM**.
- For type 2 blowers, specify the **Exhaust Exit Angle**.

7. Click **OK**.

> **Note:**
>
> The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:
>
> - **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
> - **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Assigning Heat Exchanger Properties

A heat exchanger is a two-dimensional modeling boundary condition used to represent the heat exchange with the surrounding air. For a planar heat exchanger in Ansys Icepak, a lumped-parameter model is used to model a heat exchange element. The heat exchanger enables you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator. To assign a heat exchanger boundary condition, select a piece of two-dimensional geometry (for example, a sheet).

Modeling a Planar Heat Exchanger

In Icepak, you can create a lumped-parameter model for a heat exchange element (for example, a radiator or condenser). The heat exchanger allows you to specify both the pressure drop and the heat transfer coefficient as functions of the velocity normal to the heat exchanger.

Modeling the Pressure Loss through a Heat Exchanger

In the heat exchanger model in Ansys Icepak, the heat exchanger is considered to be infinitely thin, and the pressure drop through the heat exchanger is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the radiator, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient, which can be specified as a constant or as a polynomial function.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^{N} r_n v^{n-1}$$

where r_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the resistance.

Modeling the Heat Transfer through a Heat Exchanger

The heat flux from the heat exchanger to the surrounding fluid is given as

$$q = h\left(T_{air,d} - T_{ext}\right)$$

where q is the heat flux, $T_{air,d}$ is the temperature downstream of the heat exchanger, and T_{ext} is the reference temperature for the liquid. The convective heat transfer coefficient, h, can be specified as a constant or as a polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^{N} h_n \nu^n; 0 \le N \le 7$$

where h_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the resistance in m/s.

Either the actual heat flux (q) or the heat transfer coefficient and heat exchanger temperature (h, $T_{air,d}$) may be specified. q (either the entered value or the value calculated using $q = h\left(T_{air,d} - T_{ext}\right)$) is integrated over the heat exchanger surface area.

Calculating the Heat Transfer Coefficient

To model the thermal behavior of the heat exchanger, you must supply an expression for the heat transfer coefficient, h, as a function of the fluid velocity through the resistance, v. To obtain this expression, consider the heat balance equation:

$$q = \frac{\dot{m} c_p \Delta T}{A} = h\left(T_{air,d} - T_{ext}\right)$$

where

q = heat flux

$\dot{m}$ = fluid mass flow rate

c_p = specific heat capacity

h = empirical heat transfer coefficient

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T_{ext} = external temperature (reference temperature for the liquid)

$T_{air,d}$ = temperature downstream from the heat exchanger

A = heat exchanger frontal area

Note:

The Heat Balance Equation can be rewritten as:

$$q = \frac{\dot{m}c_p(T_{air,u} - T_{air,d})}{A} = h\left(T_{air,d} - T_{ext}\right)$$

where $T_{air,u}$, is the upstream air temperature. The heat transfer coefficient, h, can therefore be computed as

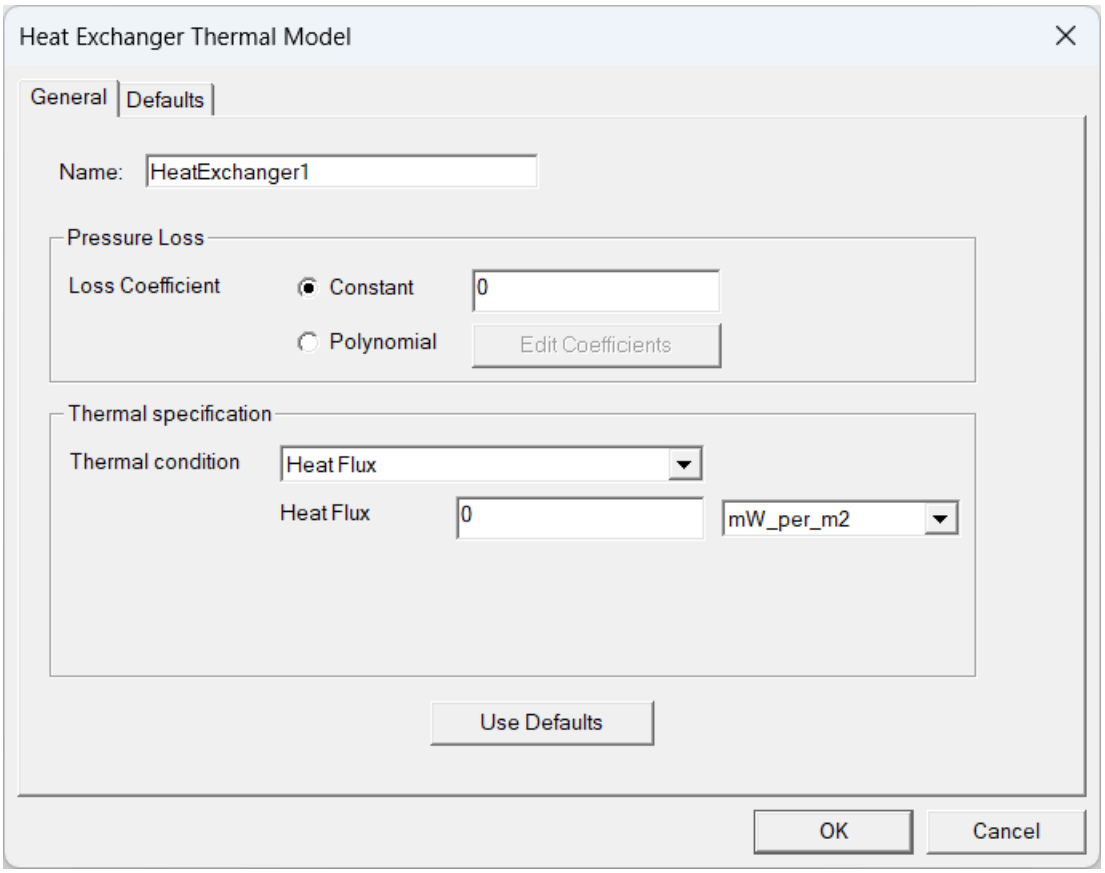
$$h = \frac{\dot{m}c_p(T_{air,u} - T_{air,d})}{A(T_{air,d} - T_{ext})}$$

or, in terms of the fluid velocity,

$$h = \frac{\rho v c_p\left(T_{air,u} - T_{air,d}\right)}{T_{air,d} - T_{ext}}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assign Heat Exchanger Properties



The **Heat Exchanger Thermal Model** dialog box contains the following general and thermal information for the selected boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

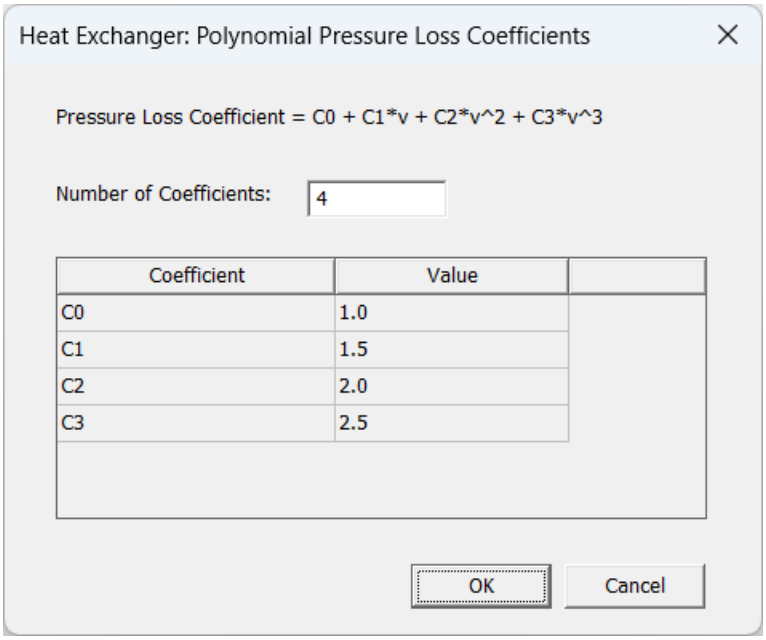
Name	The **Name** of the boundary condition appears in the **Project Manager** window and **3D Modeler** window.
Pressure Loss	
Loss Coefficient	For **Loss Coefficient**, select **Constant** or **Polynomial**. **Constant** specifies a constant loss coefficient. For **Polynomial**, click **Edit Coefficients** and specify the **Number of Coefficients** and their values in the **Heat Exchanger: Polynomial Pressure Loss Coefficients** dialog box.
Thermal Specification	
Thermal Condition	For **Thermal Condition**, select **Heat Flux** or **Heat Transfer Coefficient**.
Heat Flux	**Heat Flux** specifies a fixed rate of heat transfer from the resistance to the surrounding fluid.
Heat Transfer Coefficient	**Heat Transfer Coefficient** specifies a heat transfer coefficient to model the heat input/output of the resistance. You can specify a **Constant** value for the heat transfer coefficient (h in $q = h\left(T_{air,d} - T_{ext}\right)$. See Modeling a Planar Heat Exchanger). Alternatively, you can specify a **Polynomial** heat transfer coefficient by entering the coefficients for the polynomial equation in the **Heat Exchanger: Polynomial HTC** dialog box. Also, specify the **External Temp**.

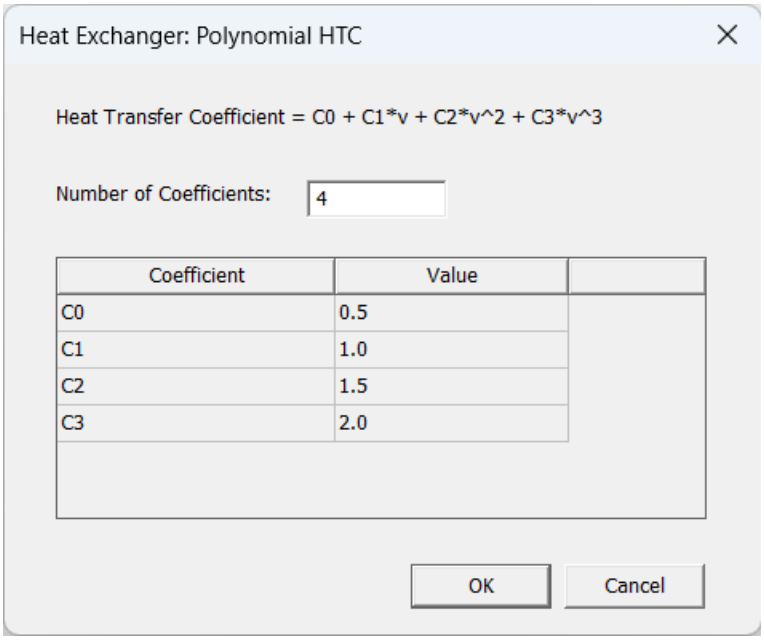
Polynomial Pressure Loss and Heat Transfer Coefficient

The polynomial pressure loss coefficient and heat transfer coefficient equations are as follows:

$C0 + (C1 * v) + \left(C2 * v^2\right) + \left(C3 * v^3\right)$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Assigning a Heat Exchanger and Defining Thermal Properties

To assign a heat exchanger boundary condition and define its properties:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Press F to enable face-selection mode.
2. Select and right-click a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Thermal** > **Heat Exchanger**.
4. In the **Heat Exchanger Thermal Model** dialog box, enter a **Name** for the heat exchanger.
5. Under **Pressure Loss**, select **Constant** or **Polynomial**.
   a. For **Constant**, enter a constant pressure loss coefficient value.
   b. For **Polynomial**, click **Edit Coefficients** and define coefficient values in the **Heat Exchanger: Polynomial Pressure Loss Coefficients** dialog box.
6. Under **Thermal Specification**, select **Heat Flux** or **Heat Transfer Coefficient** for the **Thermal condition**.
   a. For **Heat Flux**, enter a **Heat Flux** value and select a unit.
   b. For **Heat Transfer Coefficient**, select **Constant** or **Polynomial**. For **Constant**, enter a constant heat transfer coefficient and select a unit. For **Polynomial**, click **Edit Coefficients** and define coefficient values in the **Heat Exchanger: Polynomial HTC** dialog box. Also, specify the **External Temp**.
7. Click **OK**.

Note:

The boundary condition dialog has a **Defaults** tab, which contains information related to default settings for the boundary condition properties and the following buttons:

- **Save Defaults** saves the currently defined properties as the default for newly created boundary conditions.
- **Revert to Standard Defaults** sets the boundary condition properties to the saved default values.

Identifying Objects

When working with an imported CAD model, **Object Identification** allows you to create an Icepak Grille and Opening(s) with corresponding thermal boundary conditions from selected CAD geometry.

- Identifying Grilles
- Identifying Openings
- Identifying Fans

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Identifying Grilles

You can create a grille (geometry and thermal boundary condition) by selecting a CAD face or previously created openings. The boundary condition generated is automatically assigned a calculated free area ratio based on the openings within the selection.

1. Press **F** to enter face selection mode and select either one CAD face or one or more opening surfaces in the **3D Modeler** window.

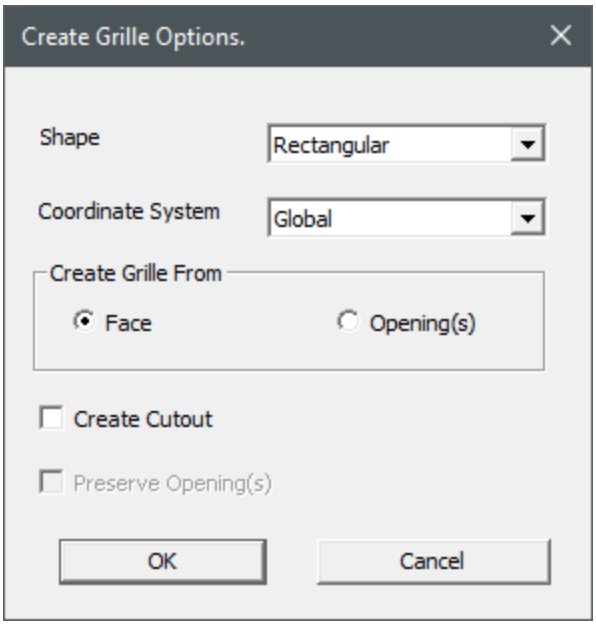
   Note:

   - Only one CAD face may be selected to create a grille.
   - When creating a grille with previously created openings, you can select the opening geometry in the history tree if they were created as two-dimensional sheets. If you selected **Create Cutout** when creating the openings, Icepak will not create the grille.

2. Right-click and select **Object Identification** > **Grille**.

   Note:

   Alternatively, you can access this option from the **Icepak** menu.



3. Select a **Shape** for the grille to be created. See "Grille Shapes" on the next page for more information.
4. Select the **Coordinate System** on which the grille will be created.

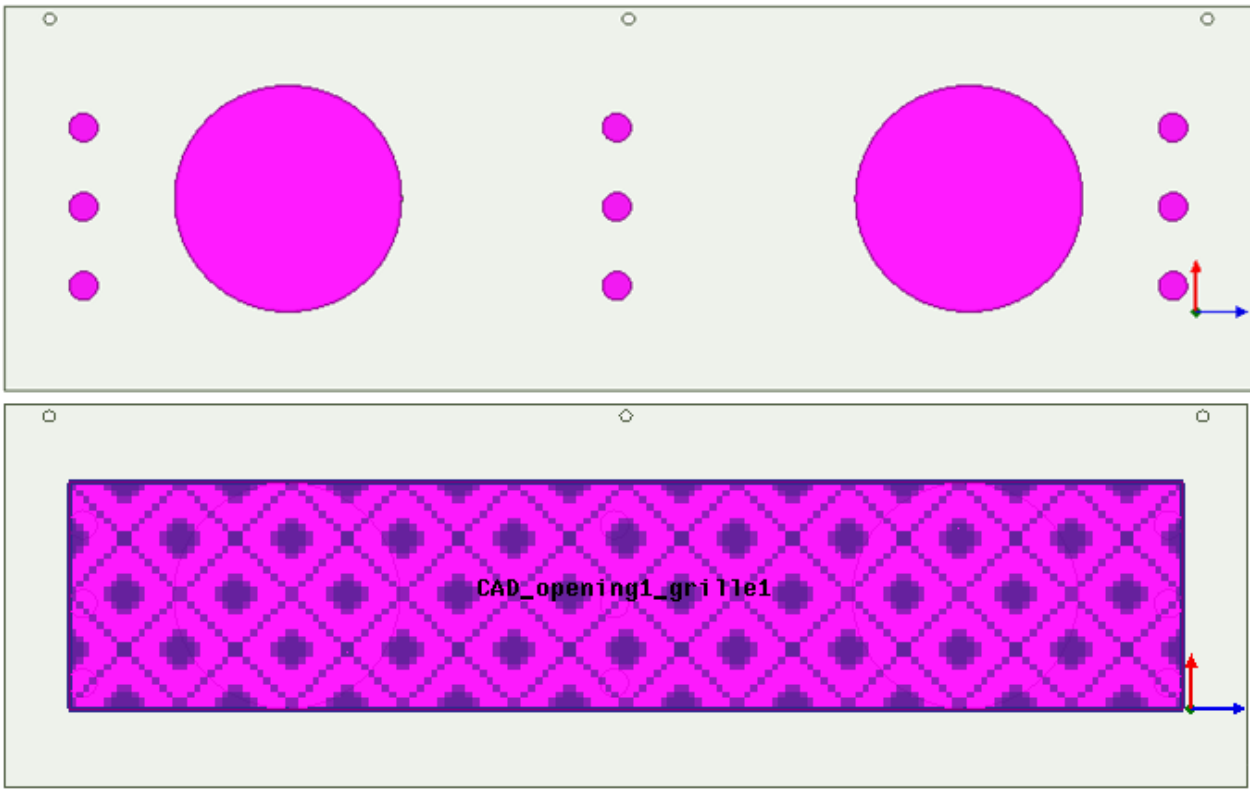
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Under **Create Grille From**, select one of the following:
   - Select **Face** when using a CAD surface.
   - Select **Opening(s)** when using one or more openings.
6. If needed, select the following check boxes:
   - **Create Cutout** creates a three-dimensional representation of the opening(s).
   - When creating a grille from openings, **Preserve Opening(s)** retains the opening geometry in the history tree and **3D Modeler** window.
7. Click **OK**.

Grille Shapes

Rectangular

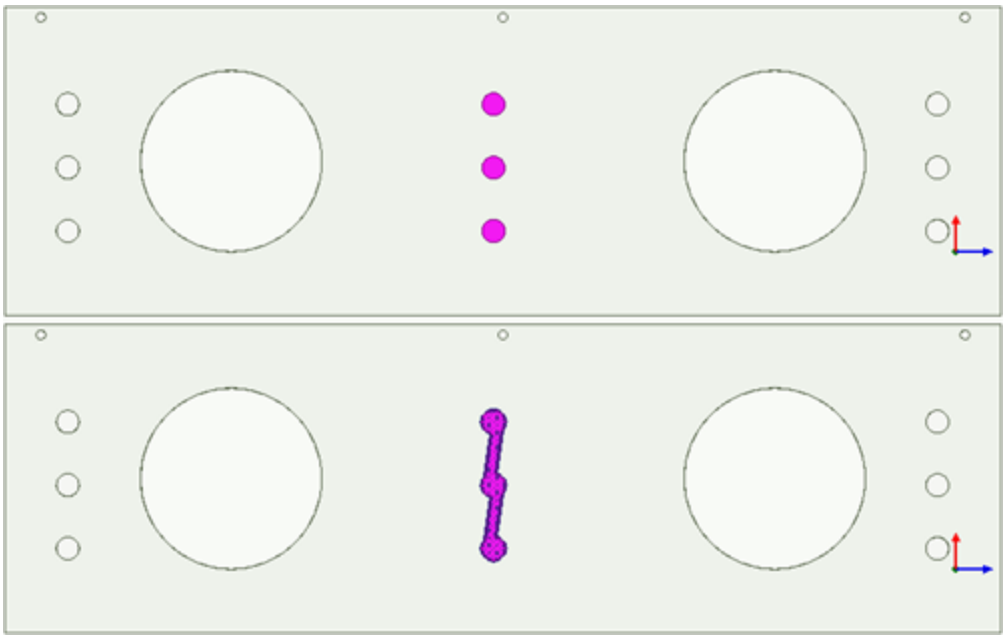
When **Rectangular** is used to create the grille, the resulting grille and corresponding boundary condition creates a rectangular fit covering only the area containing the openings on the surface.



Automatic

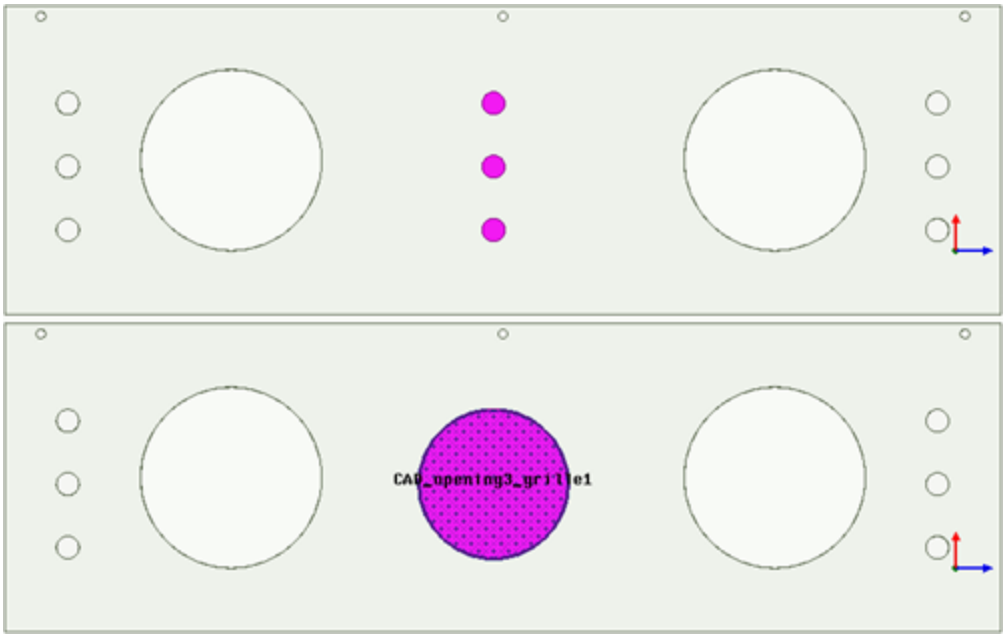
Automatic creates a grille with a best-fit polygon surrounding the openings

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Circular

Circular creates a circle surrounding the selected CAD face or openings. In the following image, three openings are selected to create the circular grille.



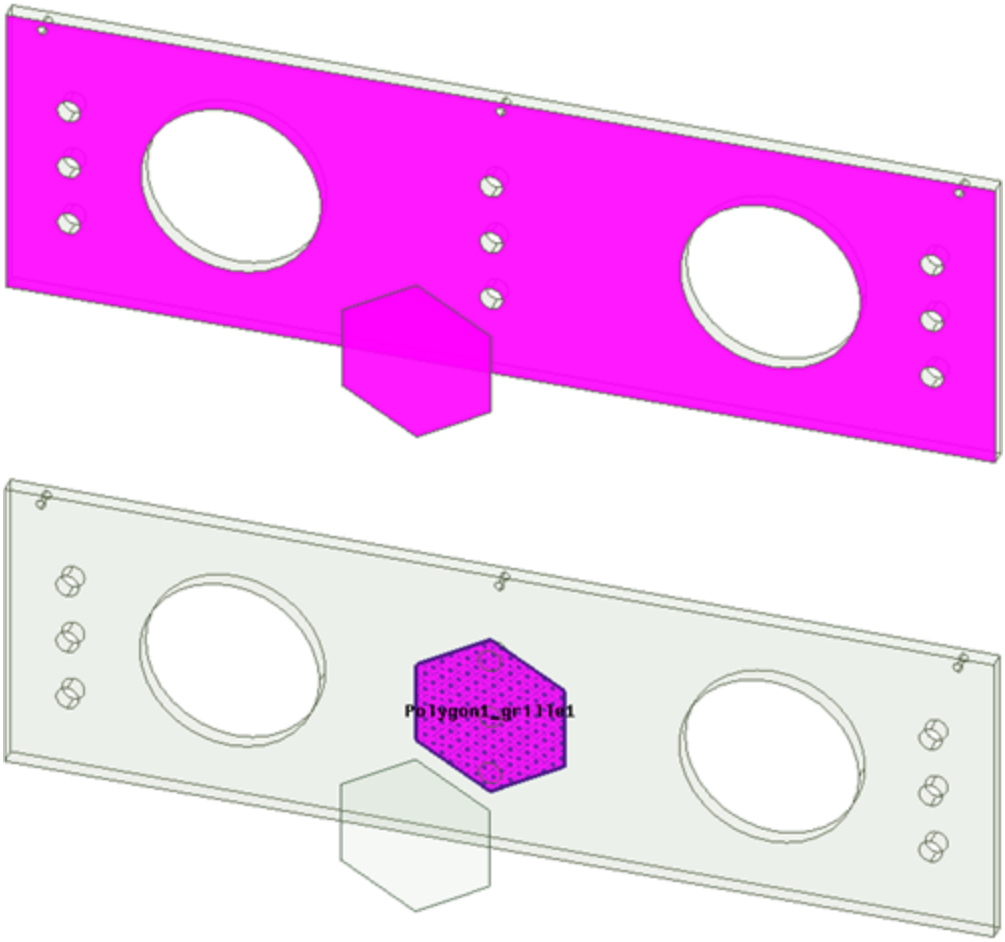
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inclined Rectangular

Use **Inclined Rectangular** to create the grille on a non-planar face that is not aligned with the Global axes. Like **Rectangular**, the size of the resulting geometry and boundary condition encompasses only the area containing the openings on the surface.

Polygonal

Polygonal requires the selection of a polygonal sheet in object selection mode (press O for object selection mode) and a CAD face in face selection mode (press F for face selection mode). The shape of the selected polygonal sheet is normally projected onto the CAD face to create the grille geometry and corresponding boundary condition.

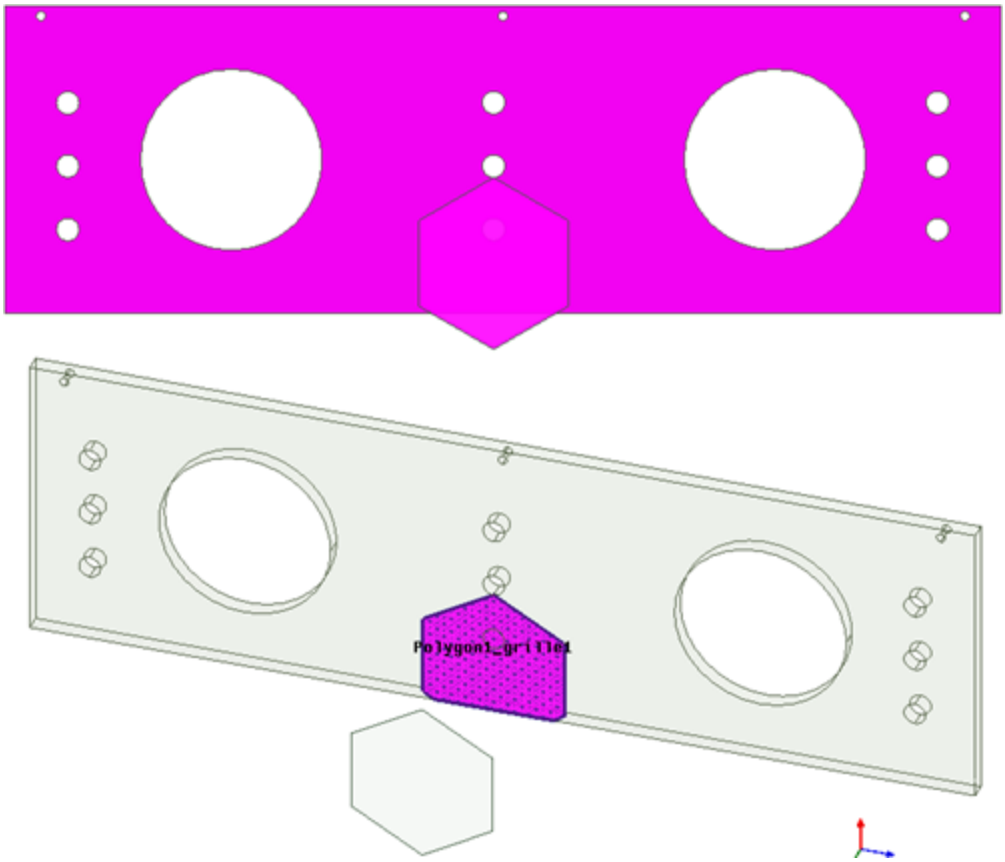


The polygonal sheet used to project onto the CAD face can be slightly offset as seen in the following image.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

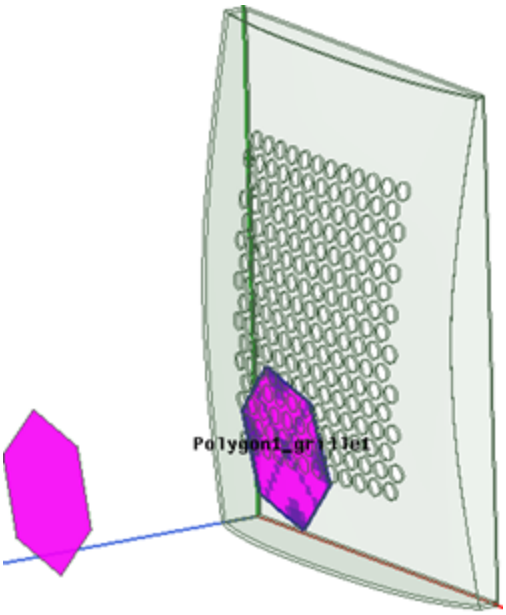
If the polygonal sheet is completely misaligned with the CAD face, or if it is not parallel to the CAD face, the grille is not created.



The polygonal shape can also be projected onto non-planar surfaces as seen in the following image.

Note: Performing grille identification on curved geometry may not complete as expected if the selected geometry has cutouts.

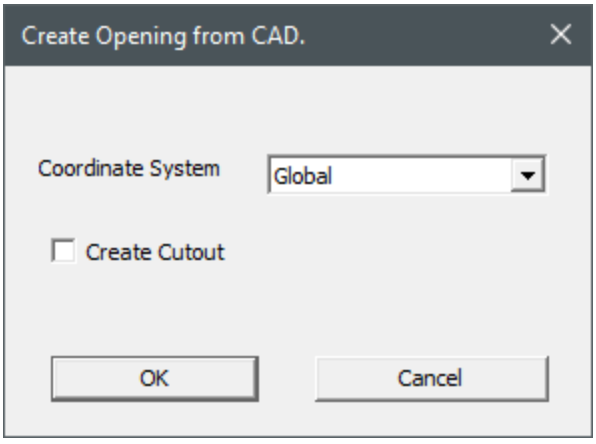
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Identifying Openings

You can create openings by selecting a CAD face. Geometry and thermal boundary conditions are created for all openings on the selected CAD face.

1. Press F to enter face selection mode.
2. Select a CAD face in the **3D Modeler** window.
3. Right-click in the **3D Modeler** window and select **Object Identification** > **Opening**.



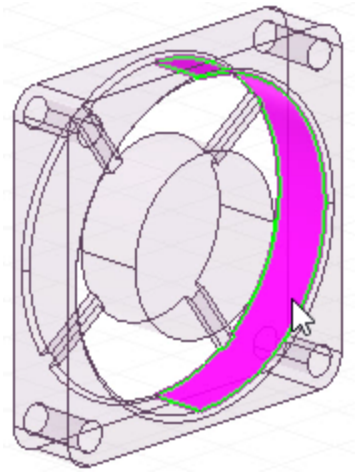
4. Select the **Coordinate System** on which the opening(s) will be created.
5. If needed, select **Create Cutout**, which creates a three-dimensional representation of the opening(s).
6. Click **OK**.

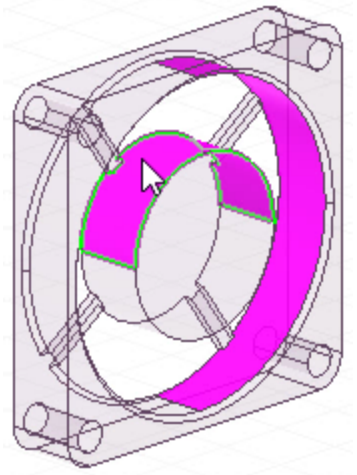
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Identifying Fans

You can create a 3D circular fan (geometry and native component instance) by selecting two curved sheets of two concentric CAD cylinders.

1. Press F to enter face selection mode and select the face of one curved sheet of a CAD cylinder.
2. Use the Ctrl key to select two curved faces, each belonging to a concentric CAD cylinder as illustrated in the images below.

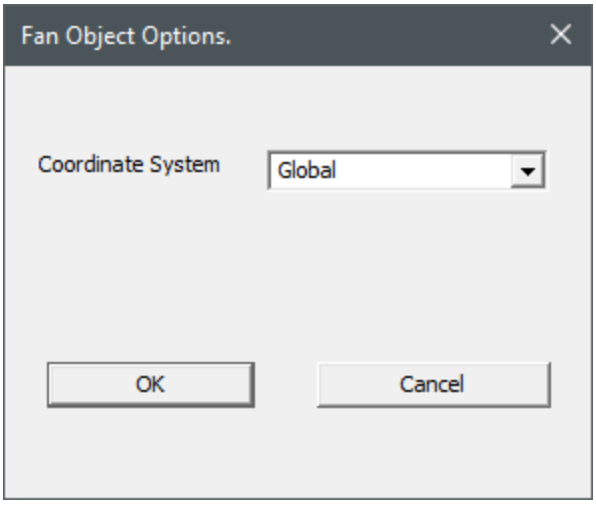




3. Right-click and select **Object Identification** > **Fan**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Alternatively, you can access this option from the **Icepak** menu.



4. Select the **Coordinate System** on which the fan will be created.
5. Click **OK**. The native component instance is generated under **3D Components** in the **Project Manager** and is automatically assigned geometric properties.
6. If needed, right-click on the native component instance and select **Edit Definition** to edit the fan's flow and thermal properties.

Reassigning Thermal Boundary Conditions

1. Select the object or object face to which you want to assign an existing object.
2. Click **Icepak>Thermal>Reassign**.

The **Reassign Boundary** window appears.

3. Select an existing boundary condition from the list, and then click **OK**.

The boundary condition is reassigned to the object or object face.

Alternatively, select the object or object face to which you want to assign an existing boundary condition. Right-click the existing boundary condition in the project tree, and then click **Reassign** on the shortcut menu.

Adding to Thermal Boundary Conditions

You can add geometry to a thermal boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object or object face you want to add to an existing boundary condition.
2. In the **Project Manager** under **Thermal**, right-click the boundary condition to which you want to add the selection.
3. From the right-click menu, select **Add to Assignment**.

The boundary condition is updated to include the selected object or object face.

Removing from Thermal Boundary Conditions

You can remove geometry from a thermal boundary condition.

1. Select the object or object face you want to remove from an existing boundary condition.
2. In the **Project Manager** under **Thermal**, right-click the boundary condition from which you want to remove the selection.
3. From the right-click menu, select **Remove from Assignment**.

The boundary condition is updated to remove the selected object or object face.

1. Select the boundary condition you want to delete by selecting its icon in the project tree.
2. On the **Edit** menu, click **Delete** .

To delete *all boundary conditions*:

- Click **Icepak**>**Thermal**>**Delete All**.

You can also *delete one or more boundary conditions* in the **Design List** dialog box:

1. Click **Icepak** menu, click **List**.

The **Design List** dialog box appears.

2. Under the **Thermal** tab, click the row of the boundary condition you want to delete.
3. Click **Delete**.

Showing and Hiding Thermal Boundary Conditions

1. Click **Icepak>Thermal>Visualization** if you want to show or hide thermal boundary conditions.
2. Clear the **View Geometry**, **View Name**, or **View Vector** selection of boundary conditions that you want to hide from view. Select the options you want to show.

The options affect all view windows.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Hiding thermal boundary conditions also turns off a check for boundary condition overlaps during boundary condition assignment. In the case of very large models with many objects, hiding boundary conditions can prevent delays during boundary condition assignment. Full model validation will subsequently check for boundary condition overlaps.

Reprioritizing Thermal Boundary Conditions

Each thermal boundary condition you assign overwrites any existing thermal boundary condition that it overlaps. You can change the priority of a previously assigned thermal boundary condition to be greater than a more recently assigned thermal boundary condition.

The order of thermal boundary conditions is important because, for any given triangle of the mesh, only one thermal boundary condition can be visible to the solvers. When two thermal boundary condition definitions overlap, the one with the higher priority is visible to the solvers.

1. Click **Icepak>Thermal>Reprioritize** to reprioritize thermal boundary conditions.

   The **Reprioritize Boundaries** window appears. The order the thermal boundary conditions appear in the list indicates the order in which they were defined. The lowest priority assignment appears at the top of the list.

2. Drag the thermal boundary condition you want to change to the desired order of priority.

   > **Note:**
   >
   > The order of thermal boundary conditions in the project tree is alphabetical. The order does not correspond to the order of thermal boundary conditions visible to the solvers.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10 - Native 3D Components in Icepak

Native 3D components in Icepak are detailed compound objects with complex geometry, such as a fan or a heatsink, and differ from other components, which can be created from existing geometry and boundary conditions (3D components) or imported from HFSS 3D Layout designs (layout components).

Native 3D components can be copied and pasted within a single design and also between two designs. In the **Project Manager**, right-click the component under **3D Components** and select **Copy**. Then right-click **3D Components** in the design to which you want to copy the component and select **Paste**.

Native 3D components available in Icepak include:

- Fan
- Heatsink
- PCB
- CTM
- 3D IC
- Packages (beta)

> **Note:** The maximum number of native 3D components displayed in the **Project Manager** and history tree is three hundred. All native 3D components are displayed in the **3D Modeler** window.

Fans

Fans are two- or three-dimensional modeling components. A fan is used to move fluid into, out of, or within the computational domain. Fan geometries include circular, rectangular, and 2D polygon. Flow types include curved and fixed flow (mass and volumetric). Fixed flow fans are always located on a domain boundary; the flow direction of the fan determines whether it is an intake fan (drawing fluid into the domain) or an exhaust fan (expelling fluid from the domain). Fans with characteristic flow curves can be located anywhere within the domain or on the domain boundary.

Fans are always associated with a magnitude and direction of flow. Circular fans can possess a central hub of non-zero radius and rectangular fans can have a rectangular hub. The hub is impervious to flow but can transfer heat to and from the fluid. The magnitude of the flow can be specified either as a fixed value or as a function of pressure drop across the fan.

In real-world applications, the performance of a fan is described by its characteristic curve. In Icepak, you can also specify a constant total mass flow or volume flow rate.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fan Characteristic Curve

The relationship between volumetric flow rate and the pressure drop across the fan (static pressure) is described by the fan characteristic curve, which is usually supplied by the fan manufacturer. Figure 1 shows a characteristic curve for a common tube-axial fan. The total volumetric flow rate, *Q*, is plotted against fan static pressure, p_{fs}.

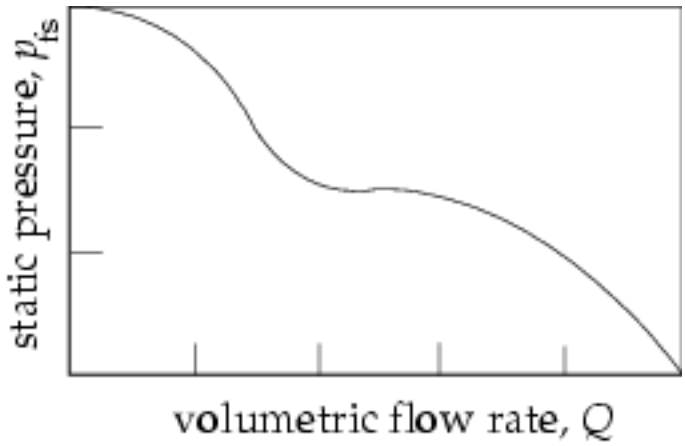


Fig. 1 Tube-axial Fan Curve

Fan static pressure is computed by:

$$p_{fs} = p_{discharge} - p_{intake}$$

$$Q = Q_0(p_0 - p_{fs})/p_0 \qquad (2)$$

where p_{intake} is the pressure averaged over the face of the intake side of the fan, and $p_{discharge}$ is the pressure averaged over the face of the discharge side of the fan. Q_0 is the volume flow rate at zero static pressure, p_0.

For internal fans, both p_{intake} and $p_{discharge}$ are computed by Icepak. For an intake fan, $p_{discharge}$ is computed by Icepak and p_{intake} is the ambient pressure. The value of the ambient pressure is specified in the **Icepak Design Settings** dialog box. For an exhaust fan, $p_{discharge}$ is the ambient pressure, and p_{intake} is computed by Icepak. The default ambient pressure is zero (gauge pressure), which should be satisfactory in almost all situations.

The accuracy of the fan flow rate used by Icepak is directly related to the accuracy with which the fan static pressure is computed. This, in turn, depends on how accurately pressure losses in the entire system are modeled. Therefore, care should be taken to model all features of the system that contribute to the overall nature of the pressure distribution in the system.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fans in Series

Fans can be arranged in series (see Figure 1). There are two major advantages of placing several small fans in series rather than using a single large fan to achieve a given flow rate:

- Fans placed in series can be made active and inactive individually, so you can create different air flow patterns.
- Fans placed in series can direct air flows into localized streams.

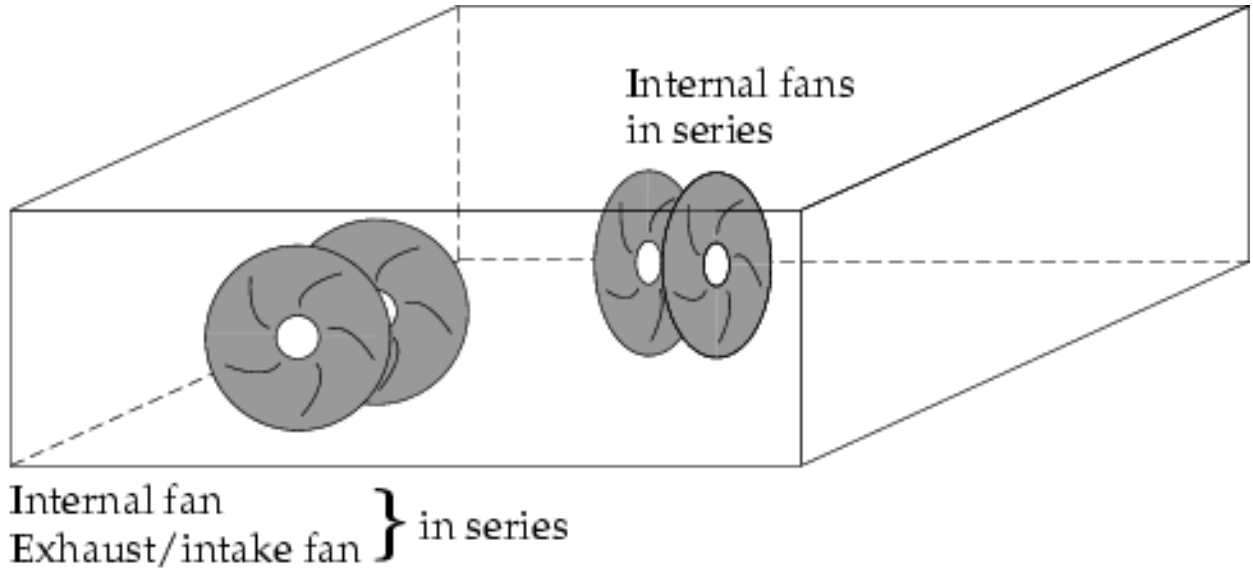
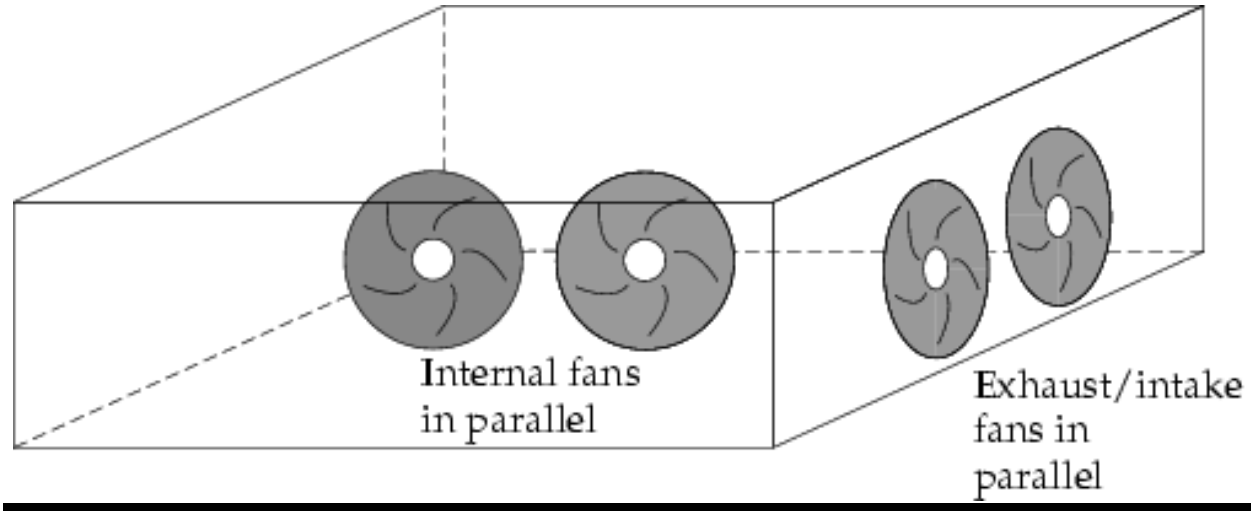


Fig. 1 Fans in Series

Fans in Parallel

Fans can also be positioned in parallel (see Figure 1). Two or more fans placed in parallel can increase flow with respect to a single fan of equivalent power.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fig. 1 Fans in Parallel

Geometry, Location, and Dimensions

Fan geometries include circular, rectangular, and 2D polygon. When creating a fan component, it is initially positioned at the center of the working coordinate system. You can change the location of the fan geometry by using the **Arrange** operations (such as **Move**) from the right-click menu in the history tree.

Simple Fans

A simple fan is a two-dimensional fan that can have circular, rectangular ,or polygonal geometry. Circular fans can include hubs. You must specify the size of the **Hub Radius** (inner radius), and its overall **Radius** (outer radius), as shown in Figure 1.

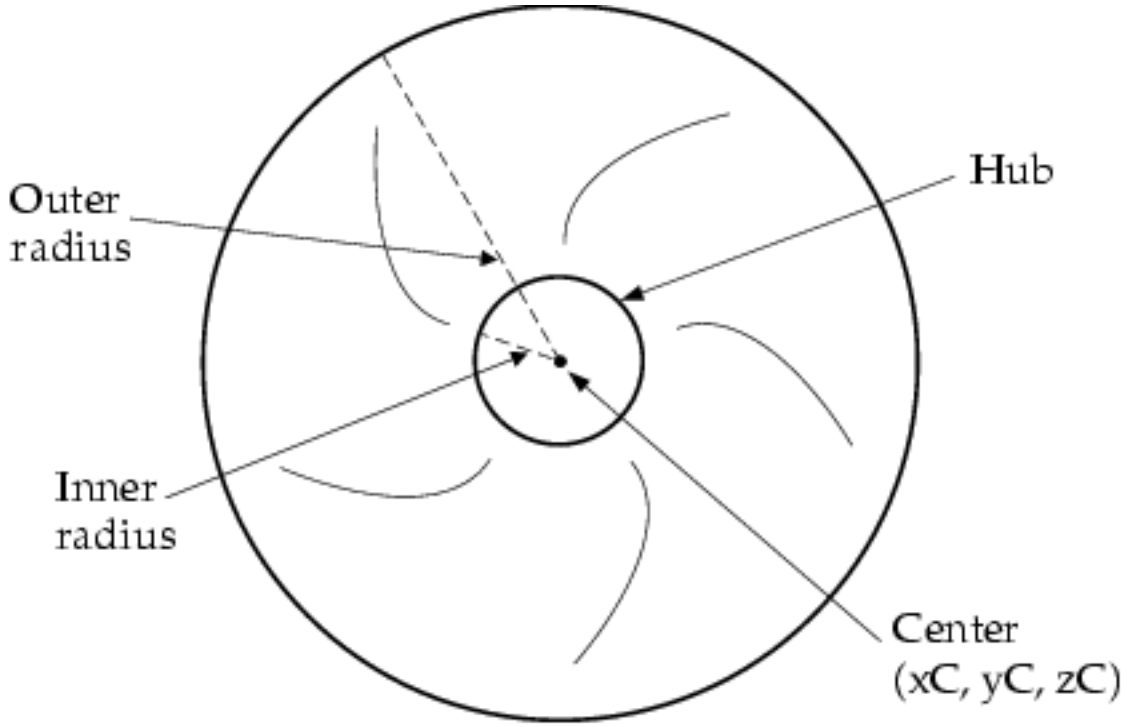


Fig. 1 Circular Fan Definition

Rectangular fans can also include a hub. To create a hub for a rectangular fan, you must specify the equivalent radius for the hub if it were to be created as a circular hub for a circular fan; that is, you specify *r* in Figure 2.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

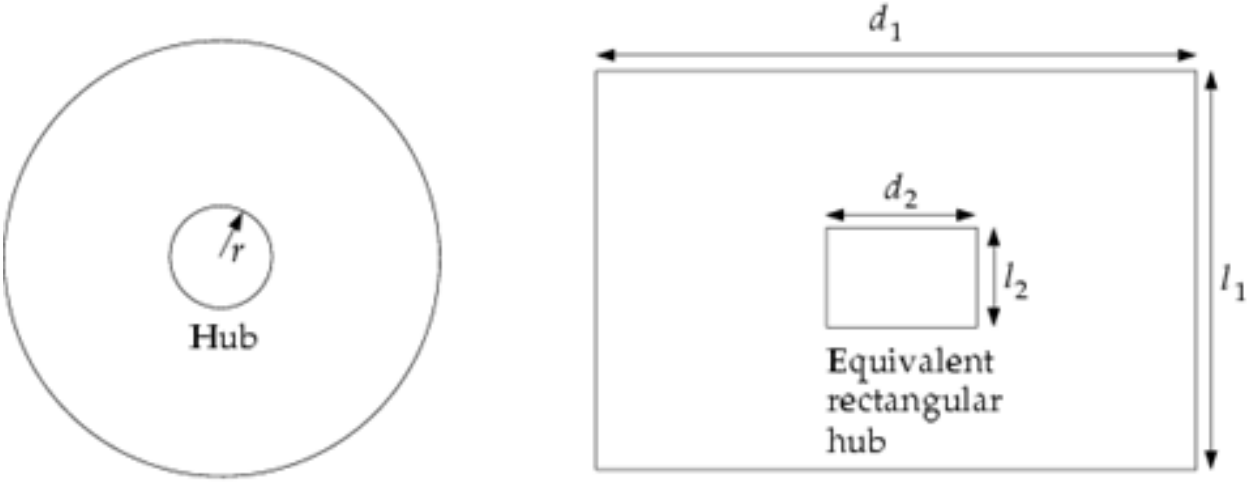


Fig. 2 Rectangular Fan Hub Definition

Icepak creates a rectangular hub with the area πr^2. Icepak uses the area of the rectangular hub and the ratio of the lengths of the sides of the rectangular fan (d_1 and l_1 in Figure 2) to calculate the lengths of the sides of the rectangular hub (d_2 and l_2 in Figure 2):

$$d_2 l_2 = \pi r^2 \text{ and } \frac{d_2}{l_2} = \frac{d_1}{l_1} \qquad (1)$$

If the fan is square, then $d_2 = l_2 = x$, and

$$x^2 = \pi r^2 \text{ that is, } x = \sqrt{\pi r} \qquad (2)$$

Flow Direction

Exhaust fans expel fluid from the mesh region. The flow exits the mesh region in a direction perpendicular to the fan.

By default, the direction of flow through an intake fan is normal to the plane of the fan (see Figure 1).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

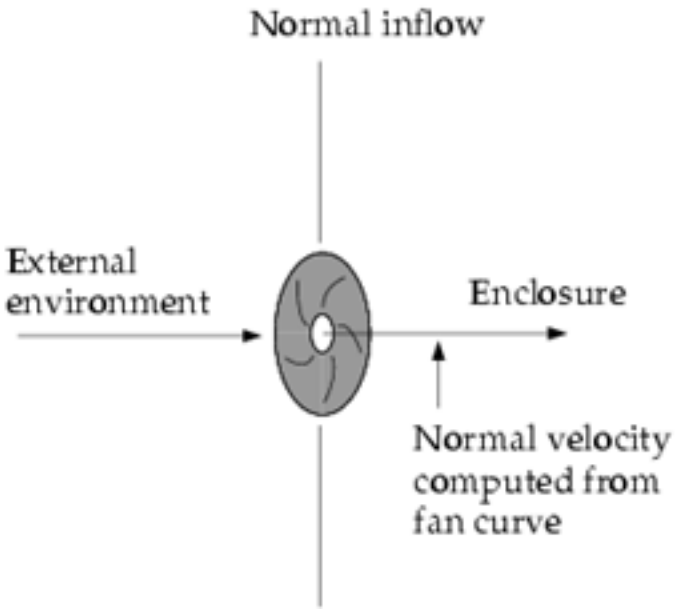


Fig. 1 Intake Fan Flow Direction

Internal fans are located anywhere inside the mesh region. Fluid flows through the fan in a direction perpendicular to it. You must specify the direction of flow for an internal fan using the **Swap Direction** button and viewing the direction of the fan flow vector in the 3D Modeler. For a rectangular, 2D polygon, or circular fan, if flow is in the direction of increasing axis coordinates, the inward direction is positive. If flow is in the direction of decreasing coordinates, the inward direction is negative.

Specifying Swirl

To specify the swirl for a fan, you must specify the swirl magnitude or the rotational speed of the fan.

Swirl Magnitude

By default, Icepak assumes that fluid exits the fan in the direction normal to the plane of the fan. Alternatively, you can specify a swirl magnitude. This skews the flow direction in the θ direction, that is, the direction of blade revolution. Swirl magnitude is defined by

$$u_\theta(r) = u_z(r)\left(\frac{r}{R}\right)S \quad (1)$$

where $u_\theta(r)$ is the velocity in the direction of revolution, $u_z(r)$ is the velocity in the direction normal to the fan, r is the radial coordinate, R is the outer radius of the fan, and S is the swirl magnitude ($S = 0$ by default).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fan RPM

Instead of using a specified swirl magnitude, Icepak can also enable the swirl factor to change as a function of the operating point on the fan curve. This is achieved by specifying the RPM (revolutions per minute) of the fan. The swirl magnitude (the ratio of the tangential velocity to the axial velocity) can then change as the fan operating point changes on the fan curve. For example, if the flow rate through the fan decreases, the swirl magnitude will increase, and if the flow rate through the fan increases, the swirl magnitude will decrease. Fan RPM is defined by

$$u_\theta(r) = \left[(RPM) \times \frac{2\pi}{60} \times r\right] \frac{1}{20} \quad (2)$$

where *r* is the radial coordinate, and it is assumed that only 5% of the maximum tangential velocity of the fan is transferred to the fluid.

Fan Efficiency

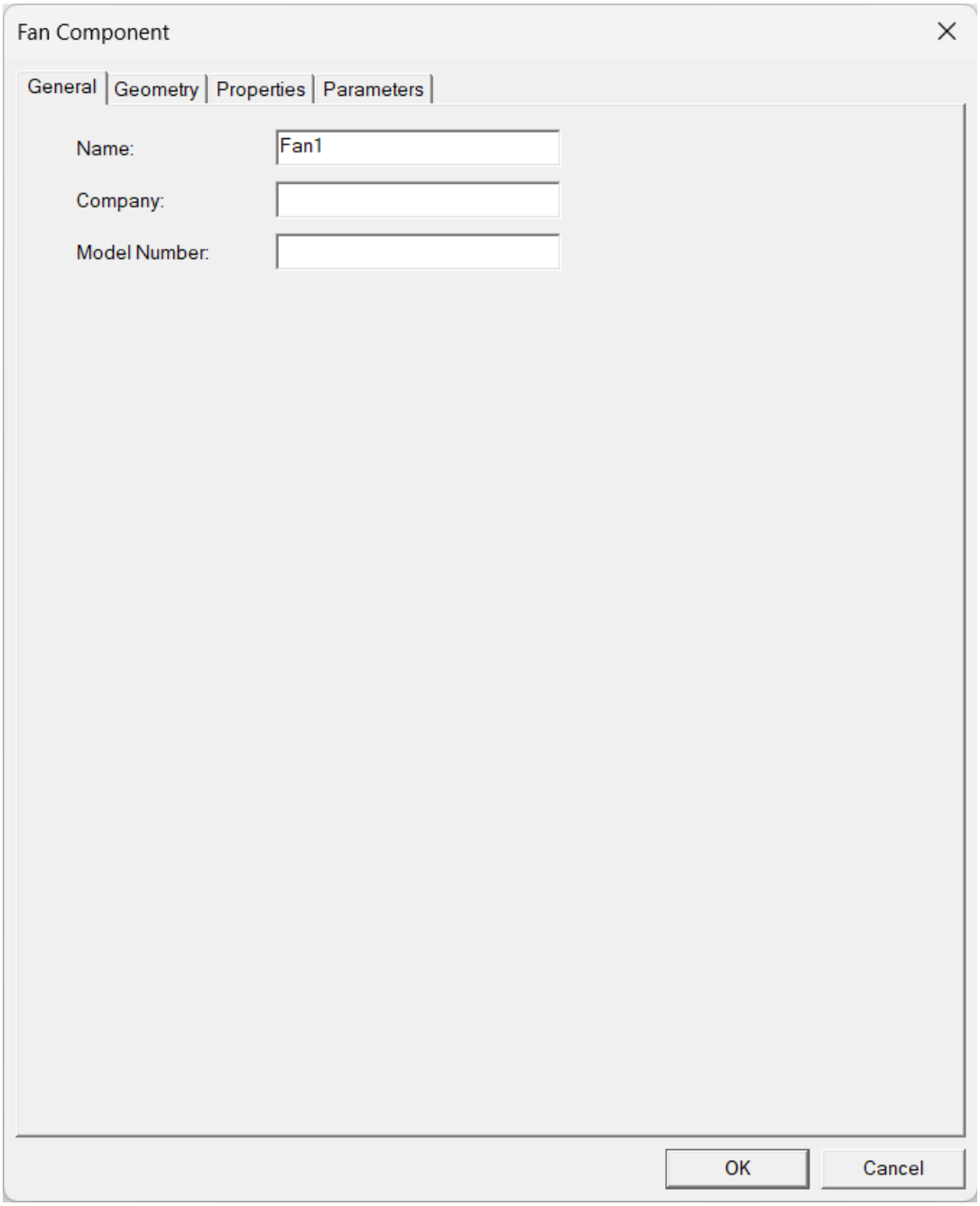
For a 3D fan, you can specify the efficiency by inputting a volumetric heat source on the hub. This power source is the percentage of the total work done by the fan's motor (*W*) that is lost as thermal energy to the hub block. The equation for calculating the power, *P*, of a fan is:

$$P = (1 - e)\, W \quad (1)$$

Creating Fans

The Ansys Electronics Desktop uses a tabbed for the creation of a fan component, including tabs to input **General**, **Geometry**, **Properties**, and **Parameters** information. To create components using a wizard interface, access the **Icepak**>**Thermal** options in the **Options** dialog (**Tools**>**Options**>**General Options**) and select **Use Wizards for data input when creating new boundaries**.

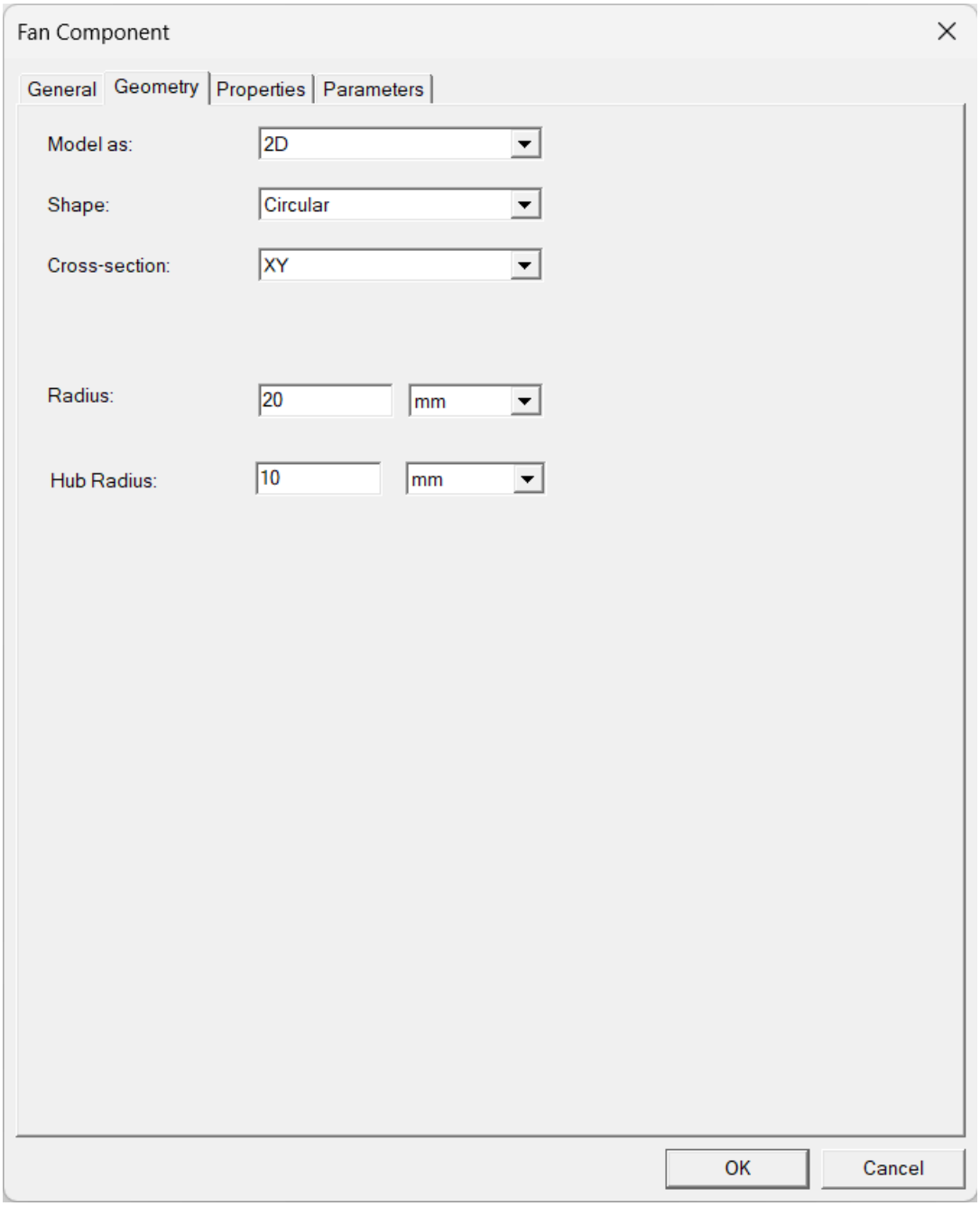
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Fan Component** dialog box **General** tab contains the following general information for the selected object.

Name	The **Name** of the object appears in the **Project Manager** window and **3D Modeler** window.
Company	**Company** specifies the manufacturer of the fan.
Model Number	**Model Number** specifies the model number of the fan.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

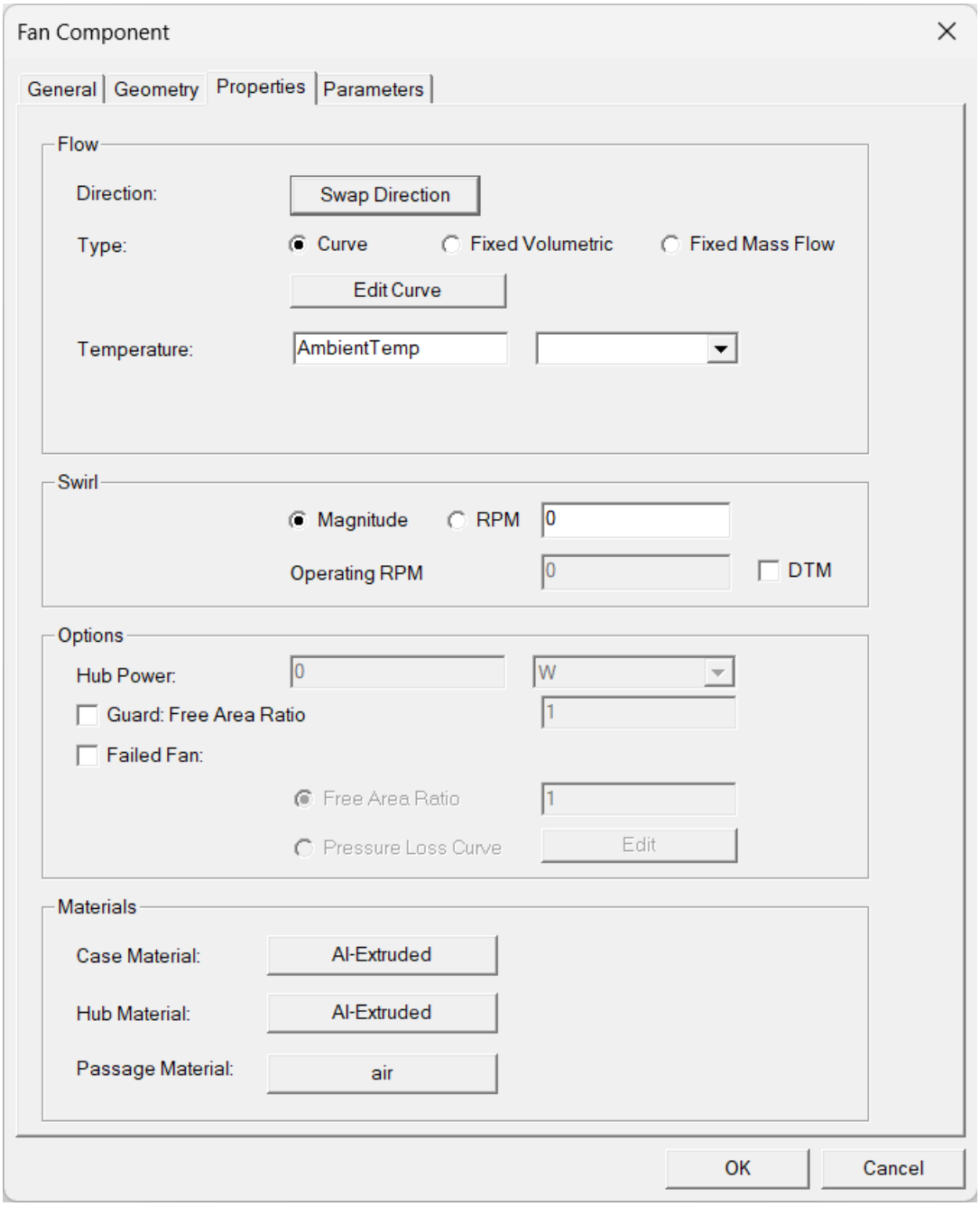


The **Fan Component** dialog box **Geometry** tab contains the following settings that define the shape and location of the fan:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model as	**Model as** specifies whether the fan is two-dimensional or three-dimensional.
Shape	**Shape** specifies the shape of the fan (circular, rectangular, or polygon). Polygon fans can only be modeled as 2D fans.
Cross-section	**Cross-section** specifies the plane on which the fan is placed (**XY**, **YZ**, or **ZX**).
Circular	
Radius	**Radius** specifies the length of the radius of the fan.
Hub Radius	**Hub radius** specifies the length of the radius of the fan hub.
Rectangular	
Fan Dimensions	**Fan Dimensions** specifies the length and width of the fan.
Hub Dimensions	**Hub Dimensions** specifies the length and width of the fan hub.
Polygon	
Radius	**Radius** specifies the length of the radius of the fan.
Num of Edges	**Num of Edges** specifies the number of sides of the fan.
Case (3D Fans)	
Side	**Side** specifies the direction the fan's case extends.
Height	**Height** specifies the height of the fan's case.
Size (Circular only)	**Size** specifies the size of the circular fan's case.
Length (Rectangular only)	**Length** specifies the length of the rectangular fan's case.
Width (Rectangular only)	**Width** specifies the width of the rectangular fan's case.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Fan Component** dialog box **Property** tab contains the following settings that define the thermal properties of the fan:

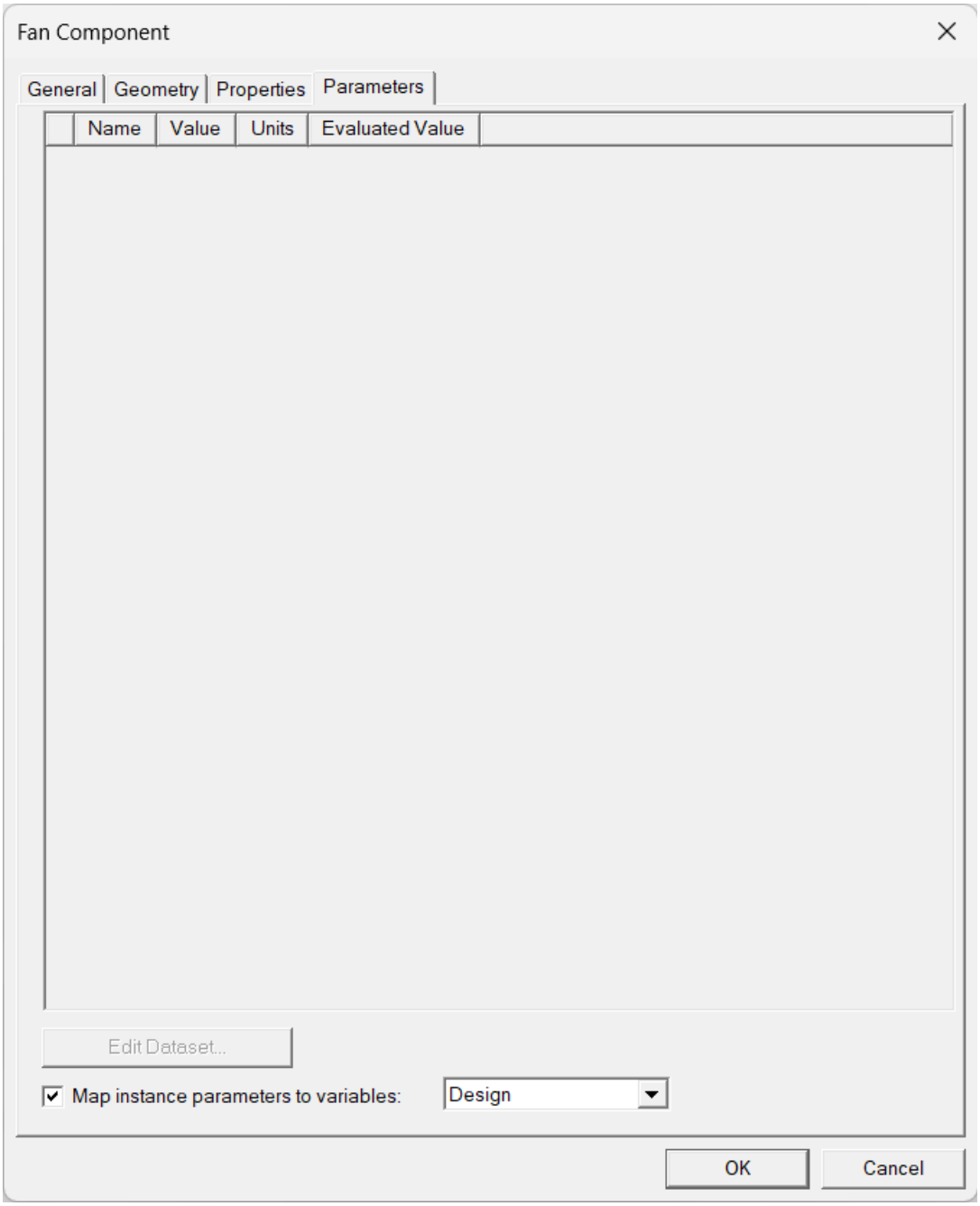
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow	
Direction	Click **Swap Direction** to change the direction of the flow created by the fan.
Type: Curve	**Curve** specifies that the fan creates a flow based on a specified flow curve. Click **Edit Curve** to open the **Edit Dataset** dialog box and enter **Volume Flow** and **Pressure** data to create the flow curve. See Adding Datasets for more information.
Note: Icepak supports a maximum of 50 volume flow and pressure pairs.	
Type: Fixed Volumetric	**Fixed Volumetric** specifies that the fan creates a fixed flow and requires a input for the volume flow rate.
Type: Fixed Mass Flow	**Fixed Mass Flow** specifies that the fan creates a fixed flow and requires a input for the mass flow rate.
Temperature	**Temperature** specifies the temperature of the fluid being drawn into the model.
Transient Strength	**Transient Strength** is available if the **Solution Type** is set to **Transient**. **Transient Strength** multiplier scales the total pressure of the fan. The total pressure of the fan is the sum of the static pressure (fan curve) and dynamic pressure: $$p_{tot} = p_{stat} + \frac{1}{2}\rho\,(U \cdot U)$$ **Transient Strength** can be defined using a square wave time variation. See Specifying Parameters as a Function of Time for more information.
Swirl	
Magnitude	**Magnitude** specifies the magnitude for the swirl of the fan.
RPM	**RPM** specifies the revolutions per minute of the fan.
Operating RPM	**Operating RPM** specifies a working RPM value that is used in conjunction with the nominal RPM from an existing fan curve to dynamically update the fan curve as follows: $$\frac{p_1}{p_2} = \frac{N_1^2}{N_2^2}, \frac{Q_1}{Q_2} = \frac{N_1}{N_2}$$ where N_1 is the nominal RPM of the fan curve, N_2 is the operating RPM, p_1 is the static pressure from the fan curve, p_2 is the updated static pressure, Q_1 is the volumetric flow rate from the fan curve, and Q_2 is the updated volumetric flow rate. Select **DTM** to specify temperature-dependent RPM after each time step of a transient simulation.
Note: For more information on dynamic thermal modeling, see DTM - Thermostat for Source	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Control.	
Options	
Hub Power (3D Fans)	**Hub Power** specifies the amount of energy, in the form of a volumetric heat source, that is absorbed by the hub from the motor. The hub power is an indirect specification of the fan's efficiency.
Guard: Free Area Ratio (3D Fans)	**Guard: Free Area Ratio** specifies whether the fan has a guard on its housing, creating a grille on the housing opposite the fan. Enter a free area ratio to specify the ratio of the area through which a fluid can flow unobstructed to the total planar area of the obstruction. Icepak calculates the loss coefficient based on the free area ratio.
Failed Fan	**Failed Fan** specifies whether the fan is operational during the simulation. Enabling this option will causes the fan (or, for a 3D fan, the side of the housing in the same plane as the fan) to act as a grille. To calculate the loss coefficient, select **Free Area Ratio** and enter a value. To define a piecewise-linear profile for the pressure drop as a function of the speed of the fluid through the failed fan, select **Pressure Loss Curve**. Then click **Edit** to open the **Edit Dataset** dialog box and specify a loss curve as you would for a grille. See Adding Datasets for more information.
Materials (3D Fans)	
Hub Material	**Hub Material** specifies the material type for the fan using the **Select Definition** dialog box.
Case Material	**Case Material** specifies the material type for the external sides of the fan using the **Select Definition** dialog box.
Passage Material	**Passage Material** specifies the fluid material passing through the fan. By default, the material assigned is that of the global Region. If no global Region exists, the default material assigned is air.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Fan Component** dialog box **Parameters** tab contains the following information about the component variables defined for the object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Creating a Fan

> **Note:** You can define component variables in Icepak native 3D components. See Defining Native Component Instance Parameters for more information.

1. In the **Project Manager**, right-click **3D Components** and select **Create** >**Fan**.
2. On the **Fan Component** dialog box **General** tab, enter following information:
- **Name**
- **Company** (if known)
- **Model Number** (if known)
3. Click the **Geometry** tab.
4. From the **Model as** drop-down menu, select **2D** or **3D** to model the fan as a two- or three-dimensional component.
5. From the **Shape** drop-down menu, select of the following:
- **Circular**
- **Rectangular**
- **Polygon** (2D only)
6. From the **Cross-section** drop-down menu, select the plane on which to place the fan.
7. Enter size information:
- For **2D** and **3D Circular** fans, enter **Radius** and **Hub Radius**.
- For **2D** and **3D Rectangular** fans, enter **Fan Dimensions** and **Hub Dimensions**.
- For **2D Polygon** fans, enter **Radius** and **Num of Edges**.
8. For **3D** fans, enter the following information under **Case**:
- **Side**
- **Height**
- **Size** (circular fans only)
- **Length** (rectangular fans only)
- **Width** (rectangular fans only)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

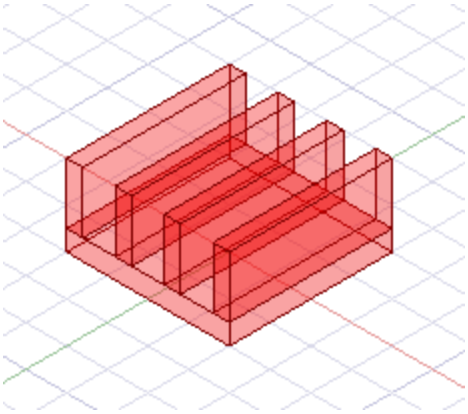
9. Click the **Property** tab.
10. Under **Flow** , do the following:
    - If needed, click **Swap Direction**.
    - Select a **Type** option and enter the flow curve data, volume flow rate, or mass flow rate.
    - Enter a **Temperature**.
11. Under **Swirl**, select **Magnitude** or **RPM** and enter the value.
12. Enter an **Operating RPM**. For transient simulations, if needed, enable **DTM**.
13. Under **Options**, do the following:
    - For 3D fans, specify a **Hub Power**.
    - For 3D fans, if needed, enable **Guard: Free Area Ratio** and specify a free area ratio.
    - If needed, enable **Failed Fan** and select **Free Area Ratio** or **Pressure Loss Curve**.
14. If you defined any component variables, click the **Parameters** tab to review the variable (s).
15. If needed, select **Map instance parameters to variables** and select an option from the drop-down list.
16. Click **OK**.

Heatsinks

Heatsinks are three-dimensional modeling components. Heatsinks consist of a base and fins and can exchange radiation with other objects in the model.

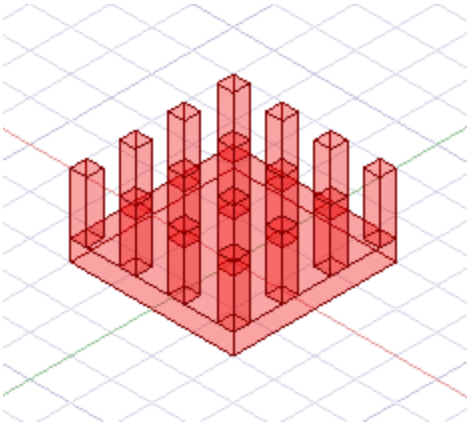
There are three types of heatsinks available:

- Extruded heatsinks consist of a base and fins that run parallel to each other across the length of the base. You specify the number of fins, their dimensions, and the direction in which the fluid will flow across the base.

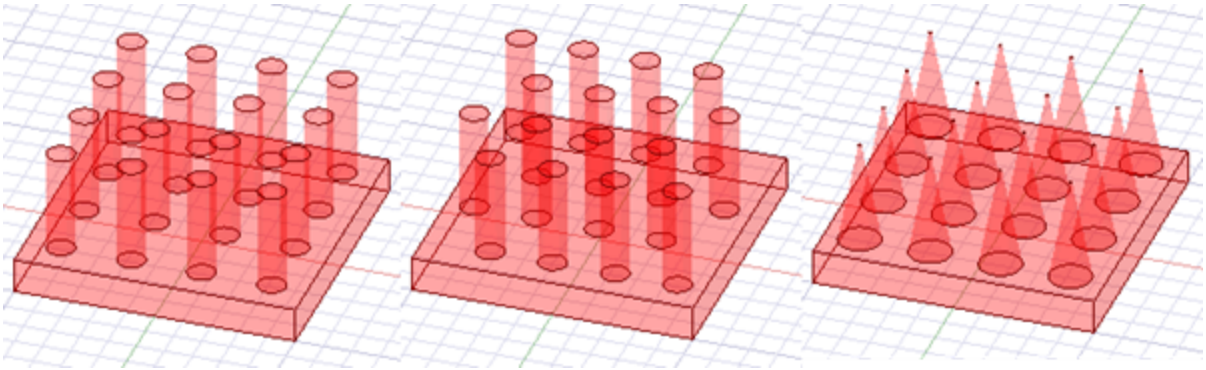


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Cross cut extrusion heatsinks have rectangular fins separated from each other in both directions parallel to the base. You specify the number of fins in the two directions parallel to the plane of the base, and the dimensions of the fins.



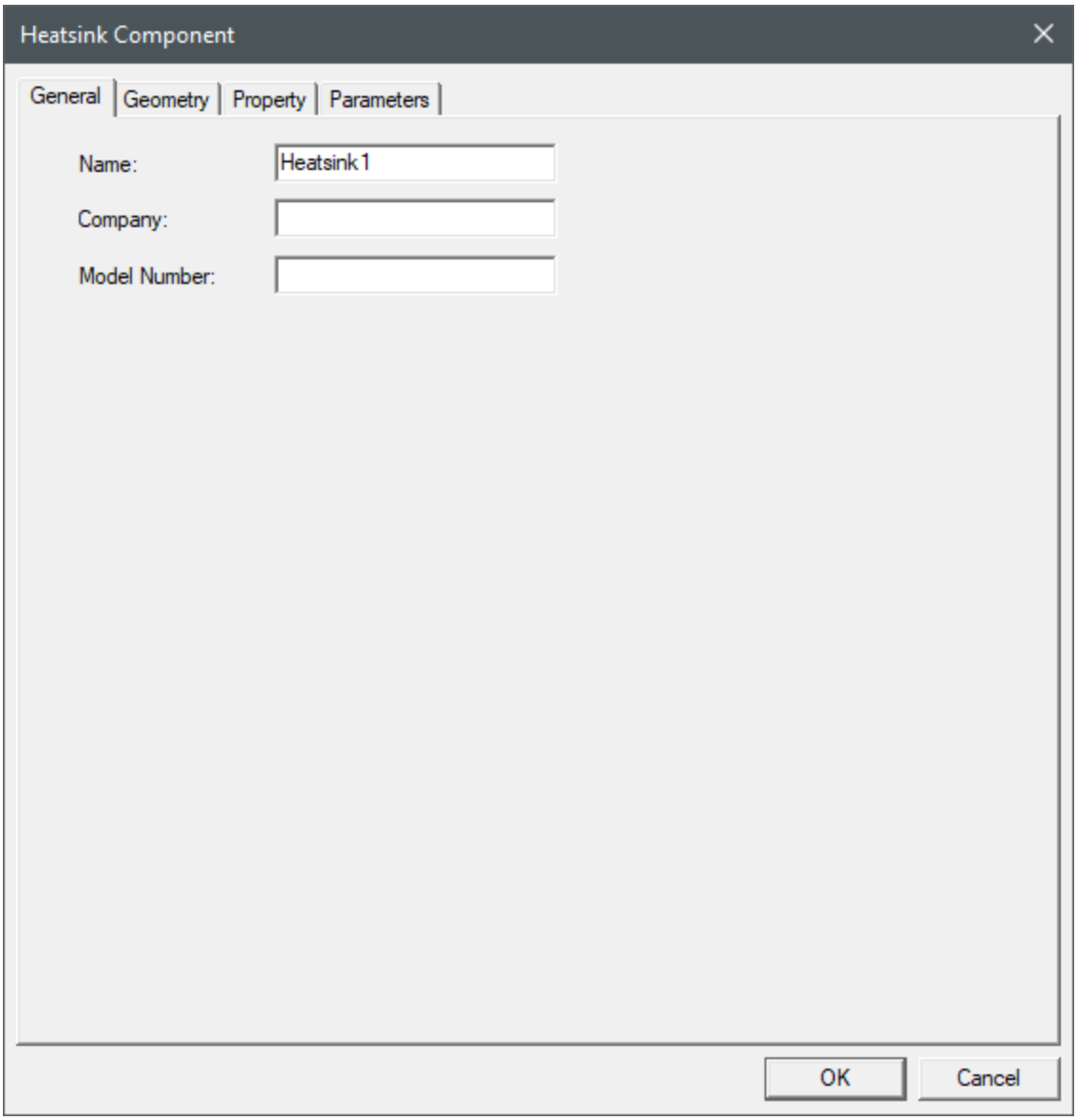
- Cylindrical pin heatsinks have cylindrical or tapered (cone-shaped) pins separated from each other in both directions parallel to the base. You specify the number of pins in the two directions parallel to the plane of the base, and the dimensions of the pins. The pins can be in-line or staggered. Examples of cylindrical pin heatsinks are shown in the following image (in-line cylinder, staggered cylinder, and in-line cone, respectively).



Creating Heatsinks

Heatsinks are three-dimensional modeling components. A heatsink can exchange radiation with other objects in the model.

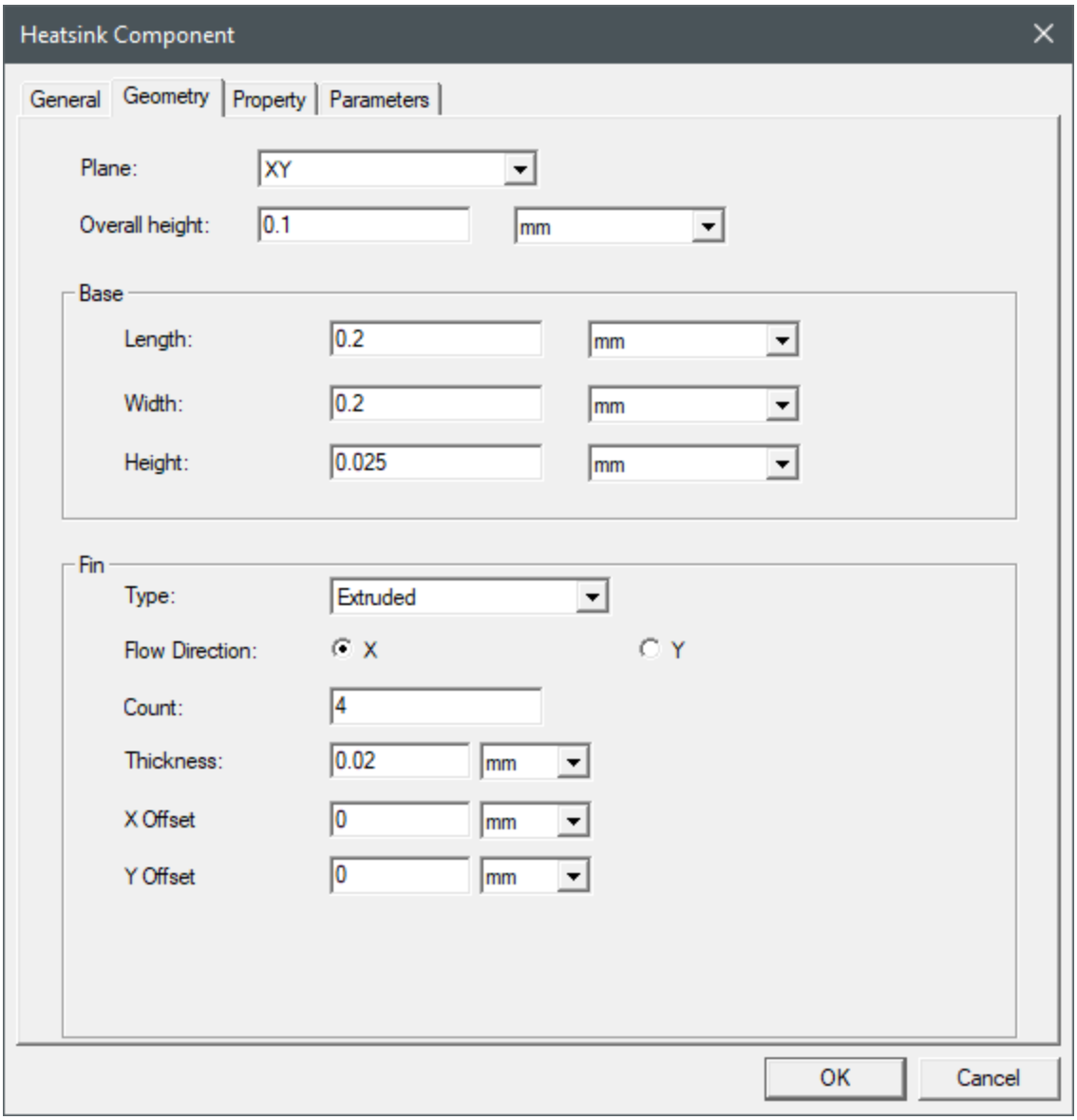
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Heatsink Component** dialog box **General** tab contains the following general information for the selected object.

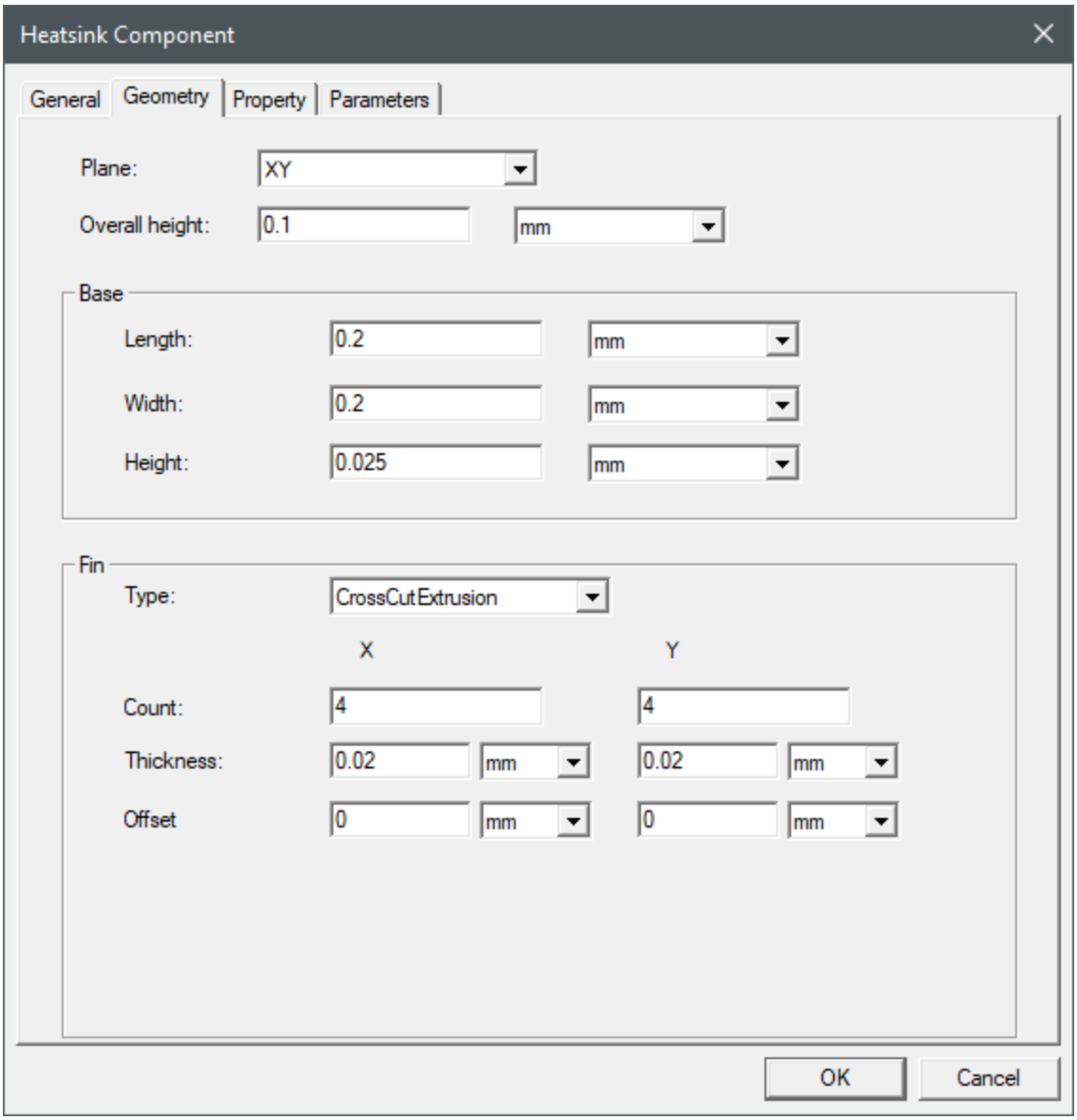
Name	The **Name** of the object appears in the **Project Manager** window and **3D Modeler** window.
Company	**Company** specifies the manufacturer of the heatsink.
Model Number	**Model Number** specifies the model number of the heatsink.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



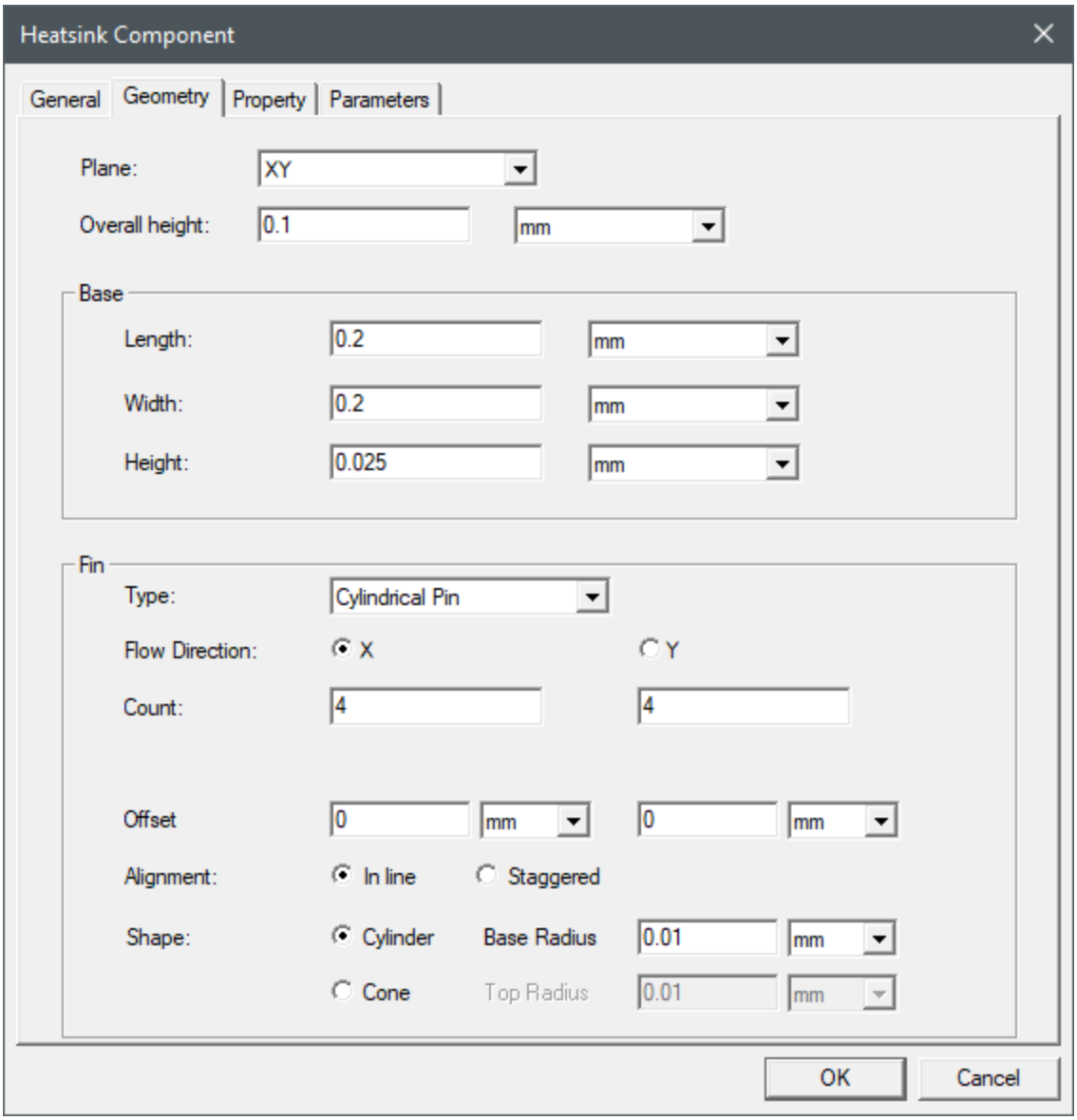
Heatsink Component: Geometry: Extruded Heatsink

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Heatsink Component: Geometry: CrossCut Extrusion Heatsink

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



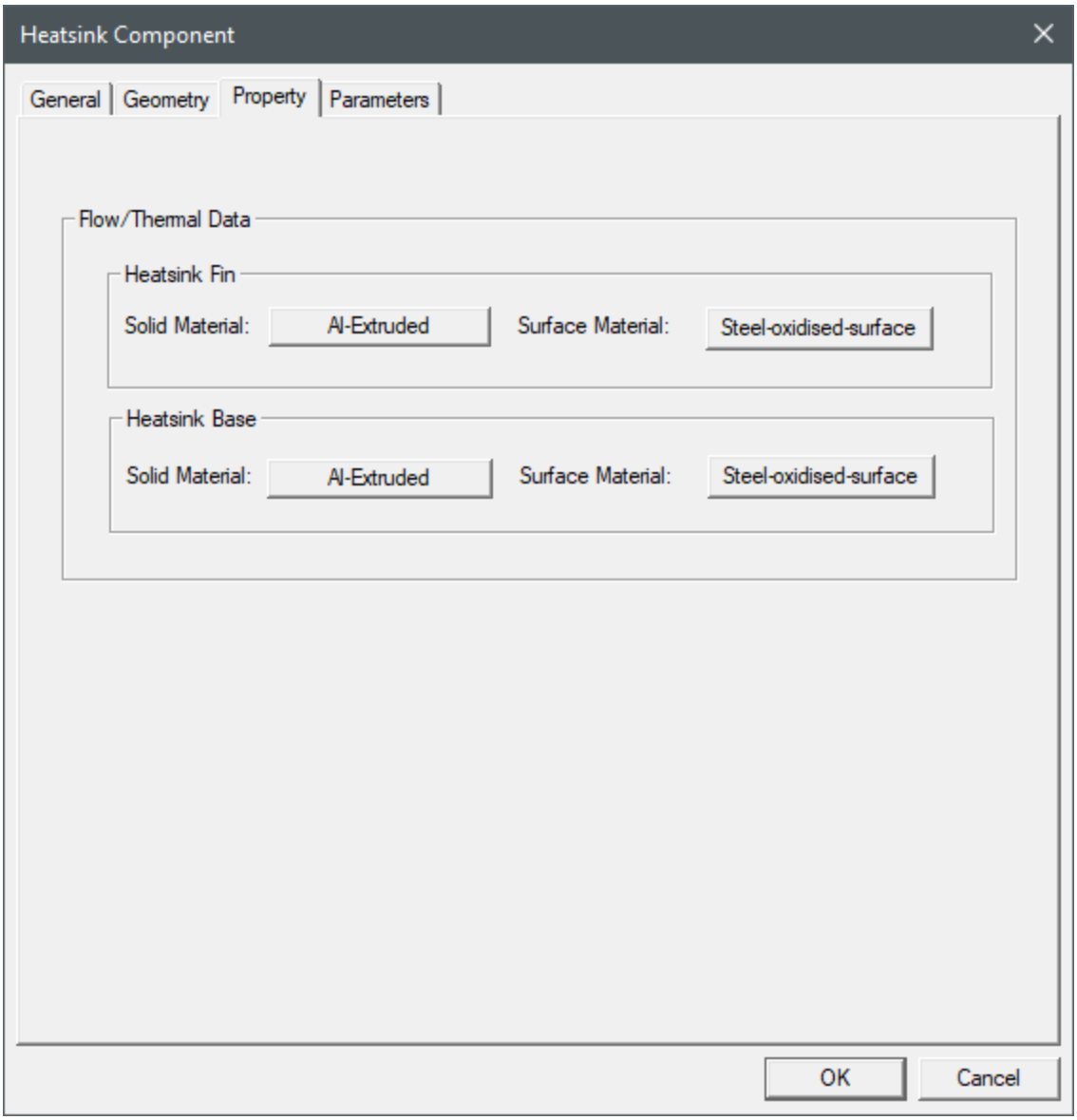
Heatsink Component: Geometry: Cylindrical Pin Heatsink

The **Heatsink Component** dialog box **Geometry** tab contains the following settings that define the shape and location of the heatsink:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane	**Plane** specifies the plane on which the fan is placed (**XY**, **YZ**, or **ZX**).
Overall height	**Overall height** specifies the height of the heatsink, including the base and pins.
Base	
Length	**Length** specifies the length of the heatsink's base.
Width	**Width** specifies the width of the heatsink's base.
Height	**Height**specifies the height of the heatsink's base.
Fin	
Type	**Type** specifies the type of heatsink.
Flow Direction	**Flow Direction** specifies the direction in which the length of the fins run along the base of the heatsink, or the direction of the flow through the domain.
Count	**Count** specifies the number of fins on the heatsink. For **CrossCut Extrusion** and **Cylindrical Pin** heatsinks, you must specify the number of fins along both directions of the base of the heatsink.
Thickness	**Thickness** specifies the thickness of the fins. For a **CrossCut Extrusion** heatsink, you can specify the thickness of fins along both directions of the base of the heatsink.
Pin Radius	**Pin Radius** specifies the radius of the pin fins. For **Cone** pin fins, specify the radius for the **Top** in addition to the **Base**.
Offset	**Offset** specifies the offset of the fins in the direction parallel to the base. For example, if you selected **XY** under **Plane**, the available directions will be X and Y. For **CrossCut Extrusion** and **Cylindrical Pin** heatsinks, you can specify the offset of fins along both directions of the base of the heatsink. For **Cylindrical Pin** heatsinks that have bases larger than the fin footprints, specify the **Offset** of the fins in the direction parallel to the base.
Pin Alignment	**Pin Alignment** specifies whether the fins on a **Cylindrical Pin** heatsink are arranged in **In line** or **Staggered**.
Pin Shape	**Pin Shape**specifies whether the fins on a **Cylindrical Pin** heatsink are a uniform **Cylinder** or a tapered **Cone**. For **Cone** pin fins, specify the **Pin Radius** for the **Base** and the **Top**.

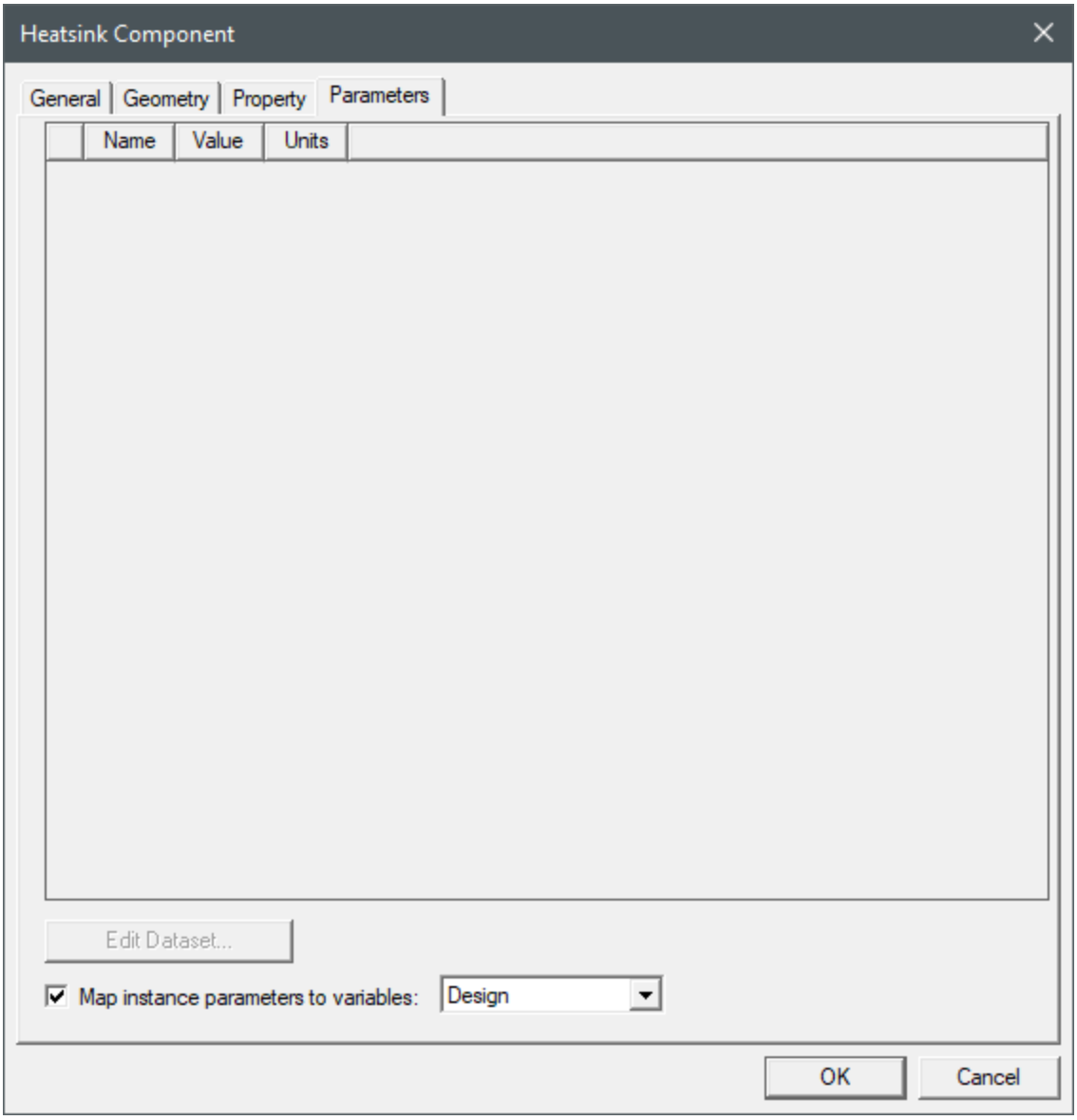
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Heatsink Component** dialog box **Properties** tab contains the following settings that define the **Flow/Thermal Data** properties of the heatsink:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heatsink Fin	
Solid Material	**Solid Material** specifies the material type of the heatsink fins using the **Select Definition** dialog box.
Surface Material	**Surface Material** specifies the material type of the external sides of the heatsink fins using the **Select Definition** dialog box.
Heatsink Base	
Solid Material	**Solid Material** specifies the material type of the heatsink base using the **Select Definition** dialog box.
Surface Material	**Surface Material** specifies the material type of the external sides of the heatsink base using the **Select Definition** dialog box.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Heatsink Component** dialog box **Parameters** tab contains the following information about the component variables defined for the object.

Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Creating a Heatsink

> **Note:** You can define component variables in Icepak native 3D components. See Defining Native Component Instance Parameters for more information.

1. In the **Project Manager**, right-click **3D Components** and select **Create** >**Heatsink**.
2. On the **Heatsink Component** dialog box **General** tab, enter following information:
   - **Name**
   - **Company** (if known)
   - **Model Number** (if known)
3. Click the **Geometry** tab.
4. From the **Plane** drop-down menu, select the plane on which to place the heatsink.
5. Enter the **Overall height** of the heatsink.
6. Under **Base**, enter the **Length**, **Width**, and **Height** of the base.
7. Under **Fins**, select the heatsink **Type**.
   - For **Extrusion**, select a **Flow Direction** and enter fin **Count**, **Thickness**, and **Offset**.
   - For **CrossCut Extrusion**, enter fin **Count**, **Thickness**, and **Offset** for both specified directions.
   - For **Cylindrical Pin**, select a **Flow Direction**, enter fin **Count**, **Pin Radius**, and **Offset**, and select the **Pin Alignment** and **Pin Shape**.
8. Click the **Property** tab.
9. Under **Flow/Thermal**, click **Base Material** and **Fin Material**s to open the **Select Definition** dialog box. On the **Materials** tab, select the material and click **OK**.
10. If you defined any component variables, click the **Parameters** tab to review the variable (s).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

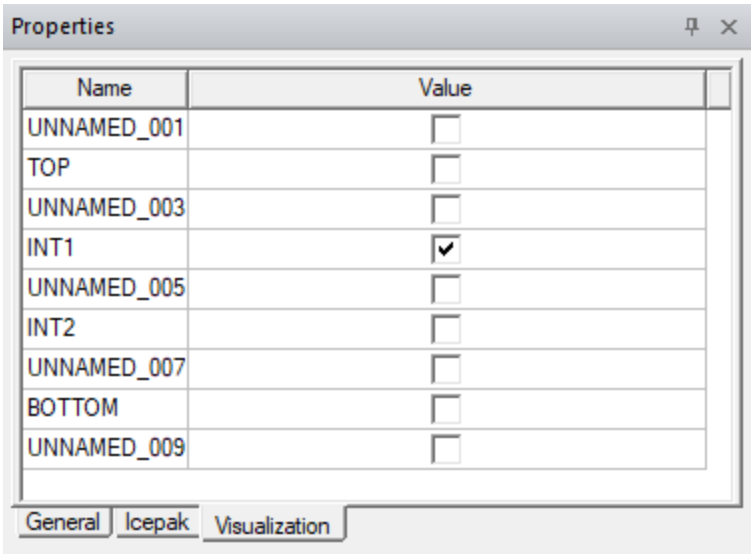
11. If needed, select **Map instance parameters to variables** and select an option from the drop-down list.
12. Click **OK**.

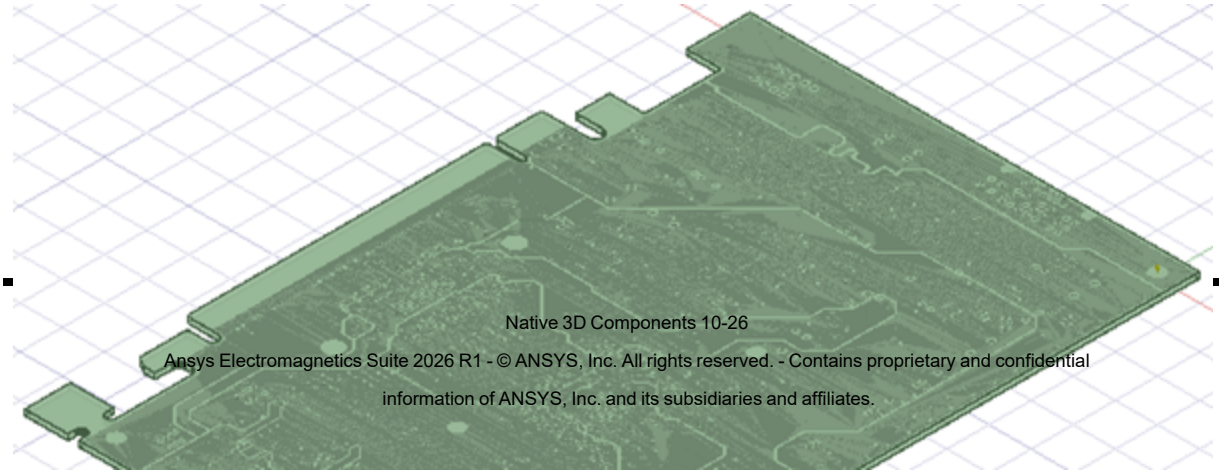
PCBs

PCBs are three-dimensional modeling components representing printed circuit boards. Almost all electronic enclosures contain PCBs. PCBs are the carriers of traces, metal planes and thermal vias that form the circuitry.

In Icepak, you can model a PCB by creating a PCB component and establishing a link with an existing HFSS 3D Layout design. The HFSS extents and stackup-layer thicknesses in the HFSS 3D Layout design are used to create the geometry in the Icepak design. Electrical and mechanical devices can be included in the imported geometry. Also, Icepak can import and model solderbumps above and solderballs below a package stackup.The data link also shares thermal losses and trace data for the PCB with the Icepak design. The 3D Layout trace data is used to determine the metal fraction content for the PCB component in Icepak.

When PCB geometry is created from an HFSS 3D Layout design, the layer names and stackup order are reflected in the Icepak design in the history tree and **Properties** window. On the **Properties** window **Visualization** tab, you can view the ECAD layers and select layer(s) to be displayed in the **3D Modeler** window.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

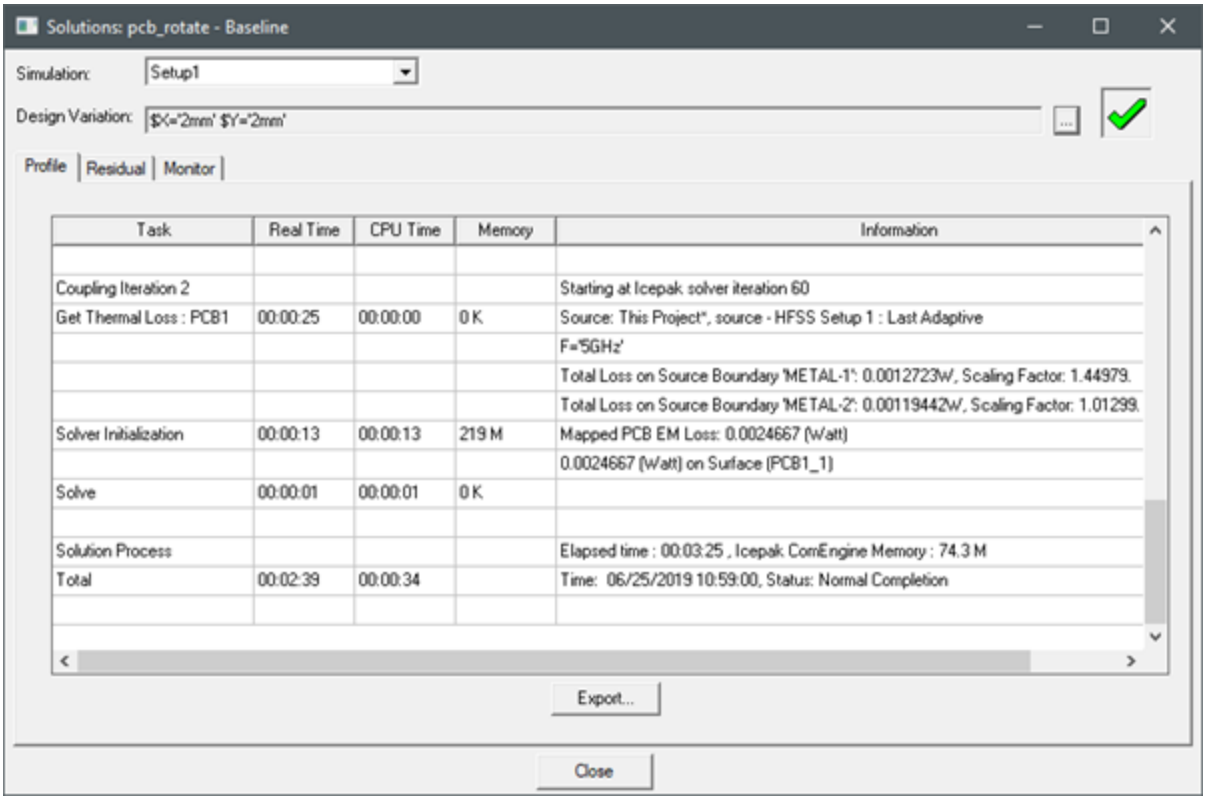
Note:

- Currently, only links between designs within the same project are supported.
- You can create multiple instances of a PCB component and move them using translation and rotation functions. Each instance uses the ECAD data link from the source design. See Inserting Component Instances and Moving and Rotating Objects for more information.

Coupling HFSS 3D Layout AC and DC Loss

In HFSS 3D Layout, you can add a solution setup to solve for alternating current (AC) and direct current (DC) loss (see SIwave Solution Setup for HFSS 3D Layout). In an Icepak design, you can link a PCB component to the HFSS 3D Layout solution setup to include the loss data in the Icepak simulation.

The losses applied to the PCB component are shown in the **Solutions** dialog box. For one-way coupling only, if the design includes multiple instances of the PCB component, the losses are applied to each instance.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

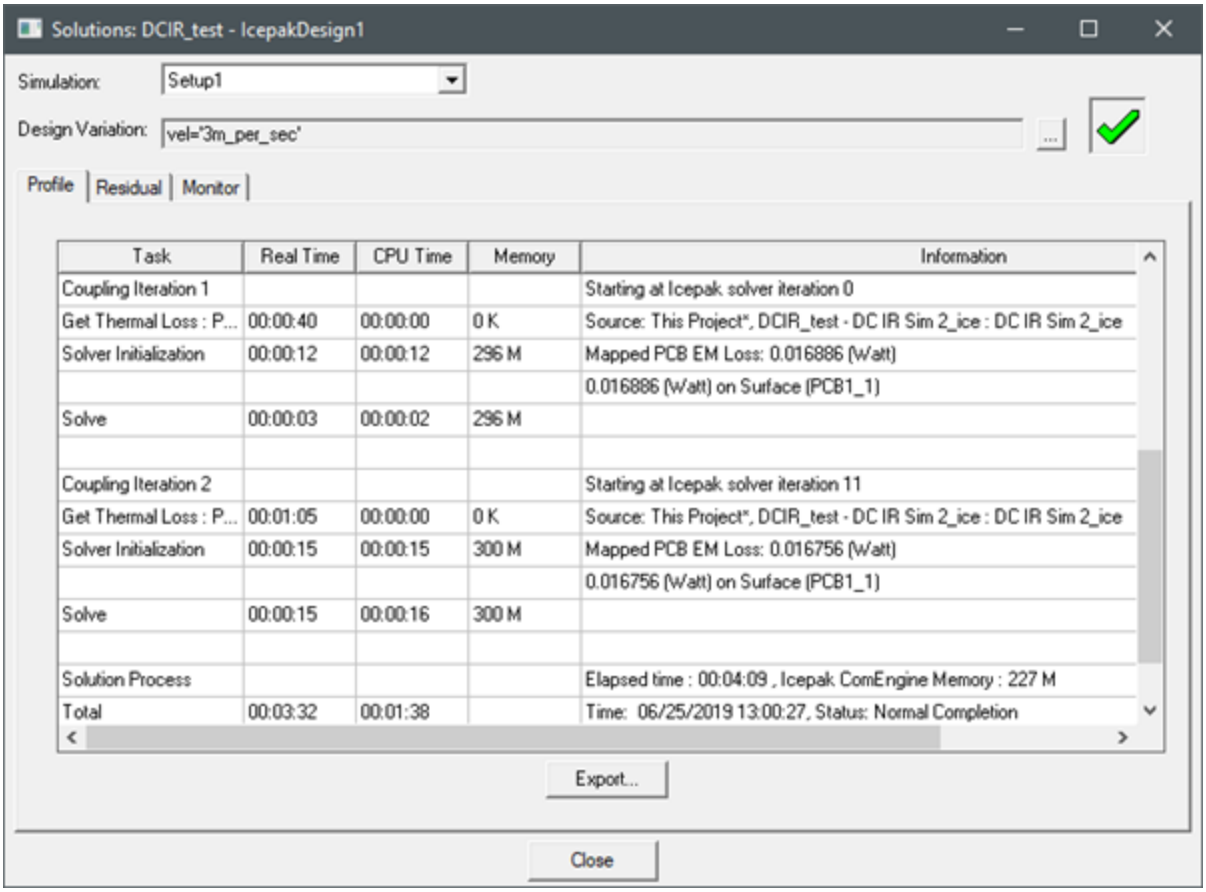
Coupling SIwave DC IR Loss

> **Note:** Use of Ansys Cloud Burst Computing services for models containing SIwave DC IR loss coupling is currently not supported.

In HFSS 3D Layout, you can add an SIwave solution setup to solve for voltage drop across a power plane (see SIwave Solution Setup for HFSS 3D Layout). In an Icepak design, you can link a PCB component to the SIwave solution setup to include the power dissipation data in the Icepak simulation.

The EM loss maps for a coupled solution setup can be displayed in the 3D Modeler window. The maps can be displayed prior to running the coupled analysis or after analyzing the solution setup. See Displaying DC IR Loss for more information.

The losses applied to the PCB component are shown in the **Solutions** dialog box. For one-way coupling only, if the design includes multiple instances of the PCB component, the losses are applied to each instance.

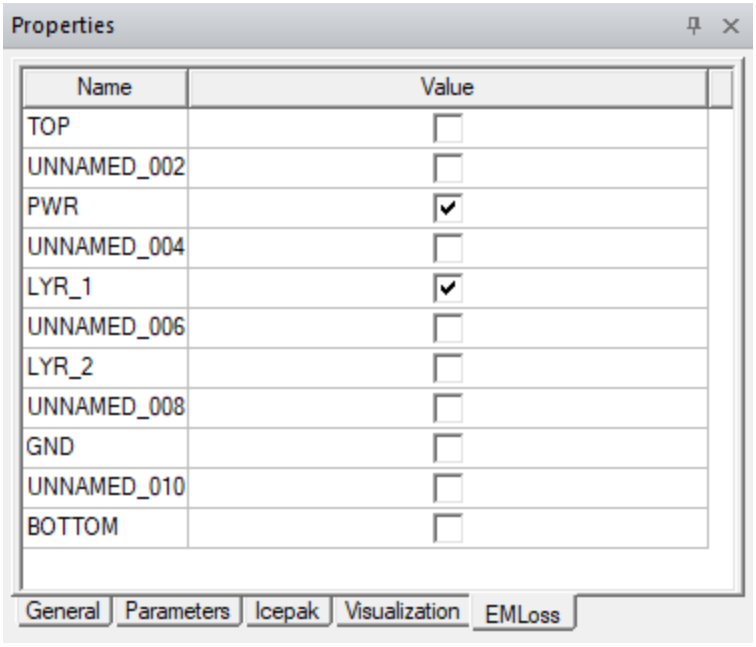


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displaying DC IR Loss

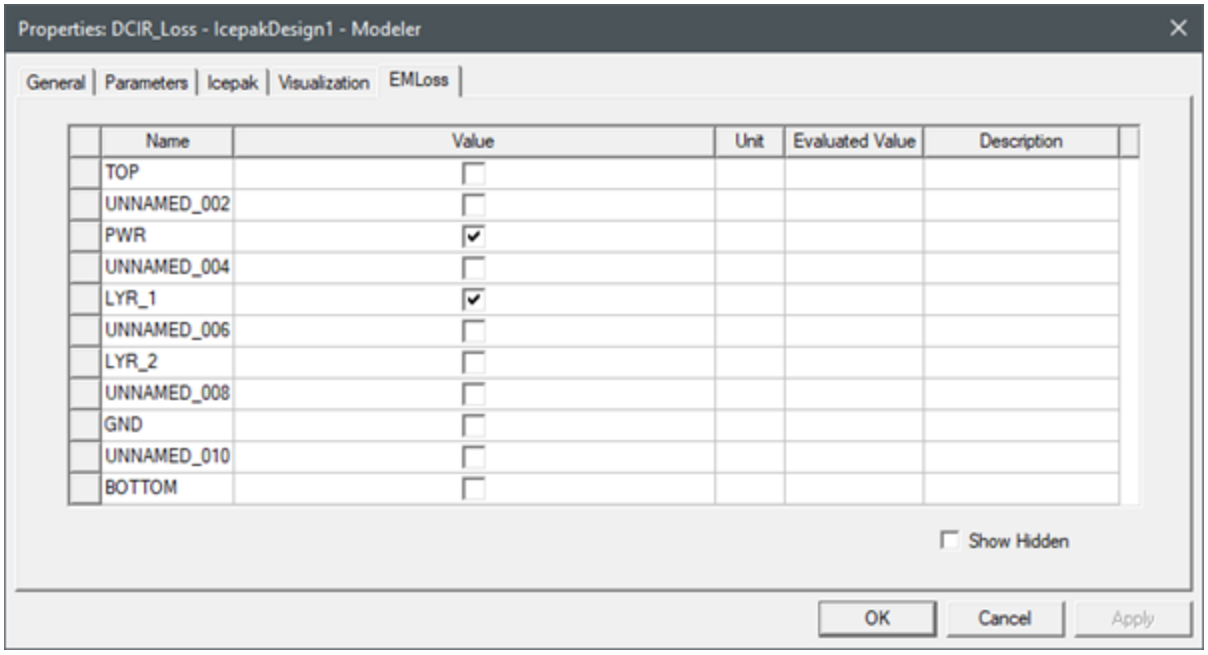
In Icepak, you can display SIwave DC IR loss data in the **3D Modeler** window by selecting layers on the **EM Loss** tab in the **Properties** window or **Properties** dialog box. The linked HFSS 3D Layout design must have results data for the EM loss maps to be displayed.

When you select a layer from the **Properties** window, the corresponding data is displayed in the **3D Modeler** window.

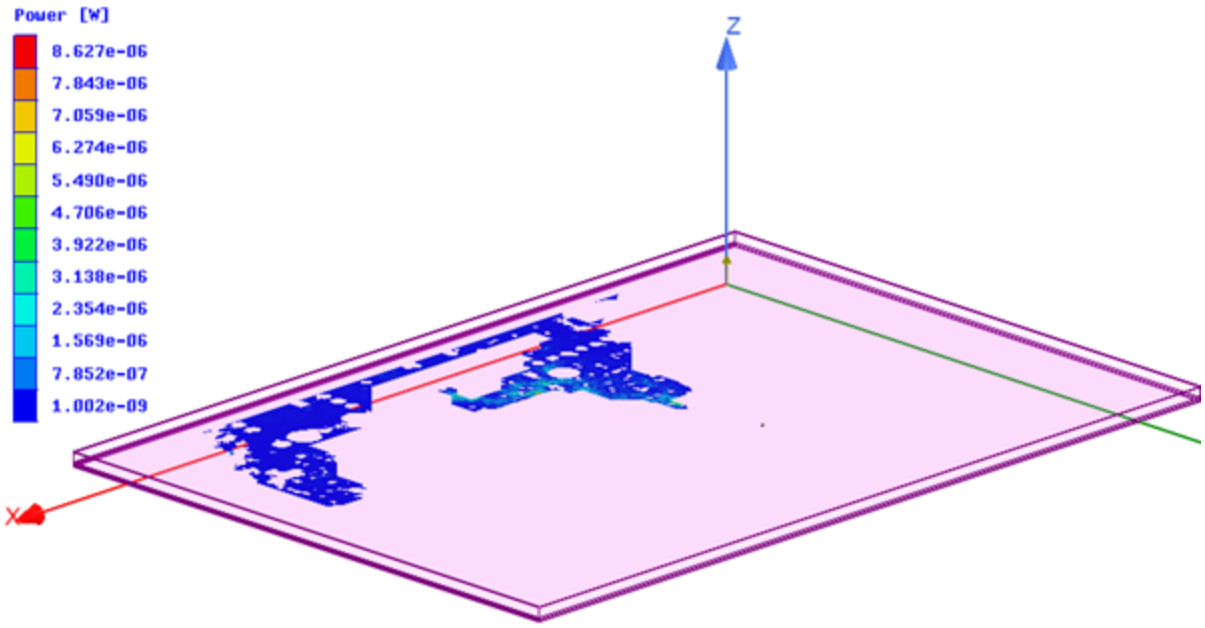


When you select a layer from the **Properties** dialog box, click **OK** or **Apply** to display the corresponding data in the **3D Modeler** window.

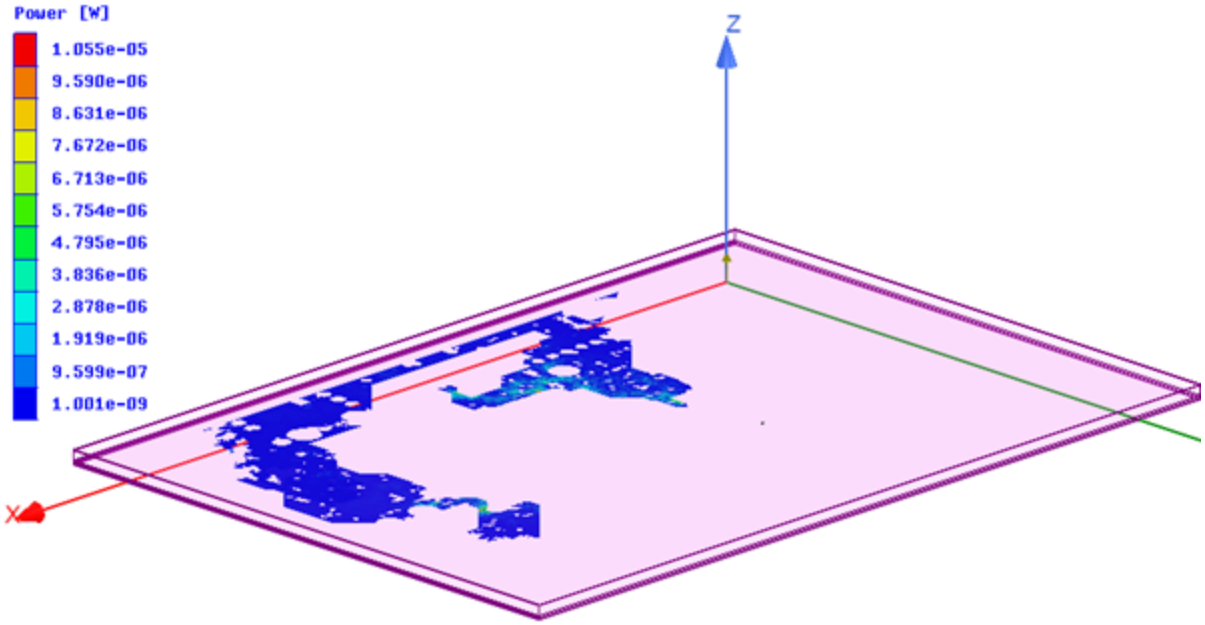
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: When multiple layers are selected to display, the range of values shown on the color legend reflects the local maximum and minimum of all layers displayed.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: The 3D Modeler color legend for EM loss maps can not be moved or edited.

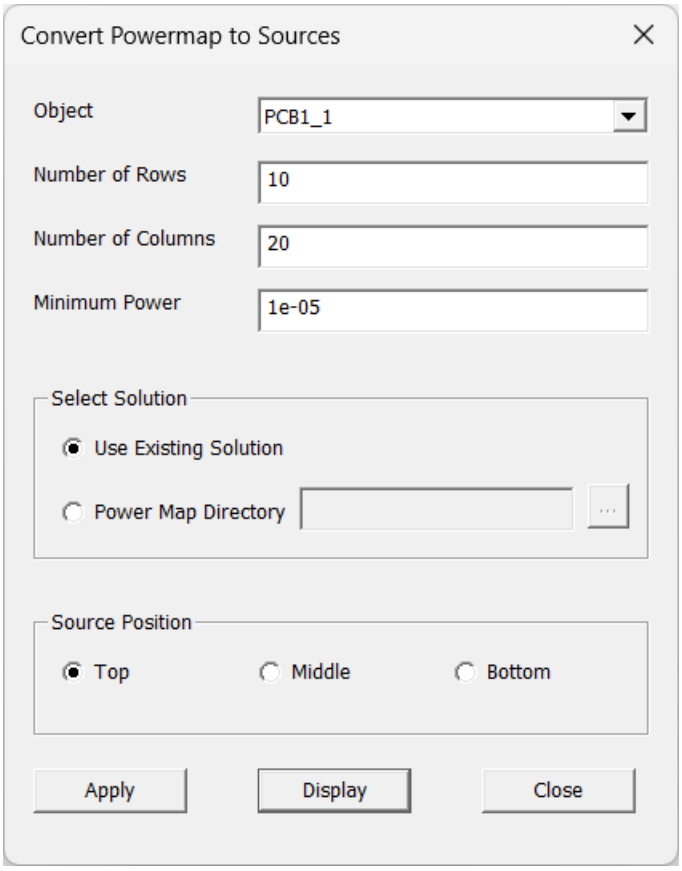
Note: You are currently unable to view EM loss map layer information (number of sources, power values, and power density values).

Note: If you use the **Clean Up Solutions** option in the linked HFSS 3D Layout design, the EM loss maps are still displayed in the Icepak design. To hide the EM loss maps, reselect the PCB in the **Project Manager**.

Convert a Powermap to Sources

When coupling an HFSS 3D Layout design using a PCB native component, you can convert the powermap into individual source boundary conditions with power values based on the selected solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Object	From the **Object** drop-down list, select the component hosting the powermap to be converted into sources.
Number of Rows	Specify the **Number of Rows** you want the powermap to be divided into.
Number of Columns	Specify the **Number of Columns** you want the powermap to be divided into.
Minimum Power	Enter the **Minimum Power** to filter out areas of the powermap to be excluded from the creation of sources.
Select Solution	
Use Existing Solution	Select **Use Existing Solution** to use the results from the currently loaded solution for the power map power values.
Power Map Directory	To use a different solution, select **Power Map Directory** and click the **[...]** button to specify the directory where the poiwer map data is saved.
Source Position	The linked design's EM loss results are provided in the form of spatial (X, Y, and Z) data. Icepak creates a 2D grid to create the sources. All points within the grid are summed to create power for the sources. Select a **Source Position** (**Top**, **Middle**, or **Bottom**) to specify where the sources are created with respect to the PCB layer's 3D bounding box.
Apply	Click **Apply** to create the sources based on the specified configuration and solution data.
Display	Click **Display** to preview the size and power values of the specified configuration in the **3D Modeler** window.
Close	Click **Close** to close the **Convert Powermap to Sources** dialog box.

Inclined PCBs

For inclined PCBs, in order to generate an acceptable mesh, you must assign a mesh region on a relative coordinate system with respect to the Global coordinate system. You can create a relative coordinate system before creating the PCB component on it. Also, if you reposition the PCB geometry, you can create a new relative coordinate system based on the current position of the geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

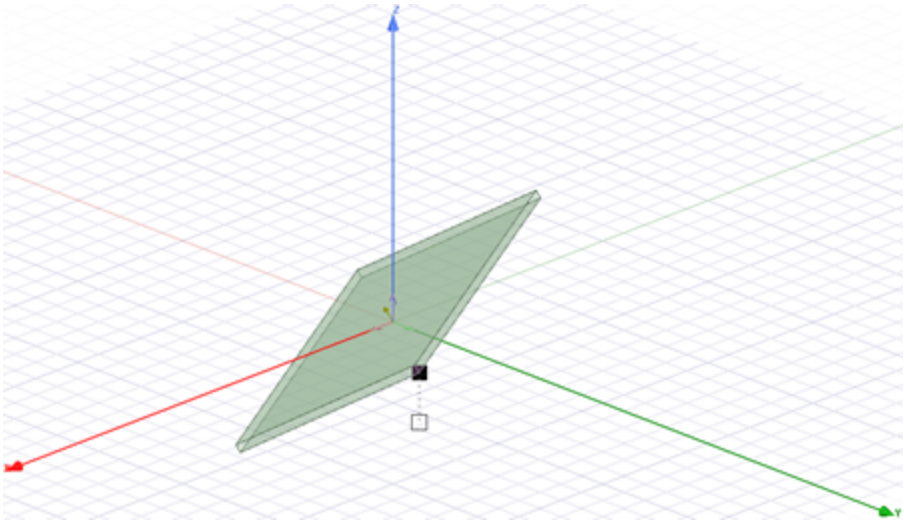
Note: If rotational and/or translational transformations are applied to the PCB in the Icepak design, the imported EM Losses will be accordingly transformed for the one-way or two-way coupling analyses.

Create a Relative Coordinate System

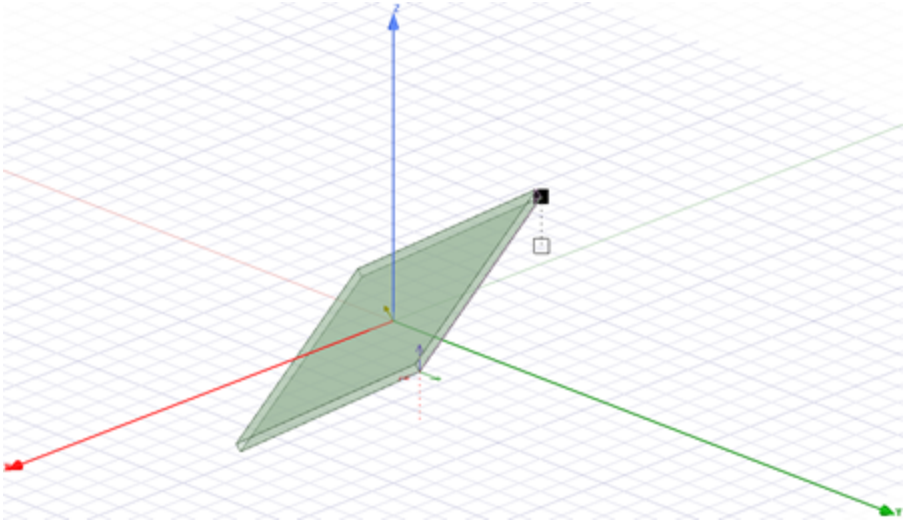
1. From the **Modeler** menu, select an option from **Coordinate System** > **Create Relative CS** and create the coordinate system.
2. In the history tree, expand **Coordinate Systems** and click on the relative coordinate system to set it as your working coordinate system.
3. In the **Project Manager**, right-click **3D Components** and select **Create** > **PCB** and create the PCB native component.
4. Select the PCB geometry, right-click in the **3D Modeler** window, and select **Assign Mesh Region**. The mesh region is created on the relative coordinate system.

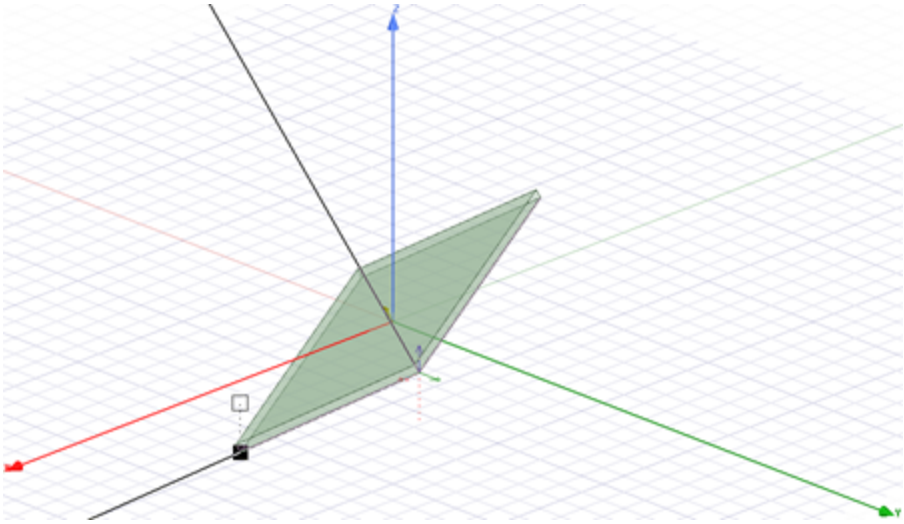
Create a Relative Coordinate System Based on Inclined Geometry

1. From the **Modeler** menu, select **Coordinate System** > **Create Relative CS** > **Both**.
2. Select three points on the base of the PCB geometry as in the following images.



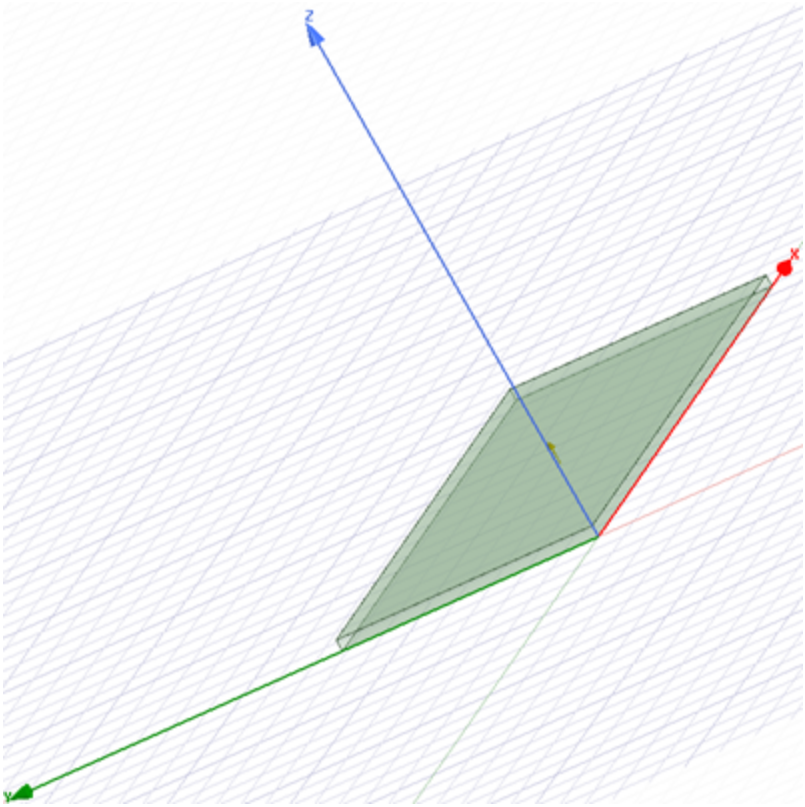
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





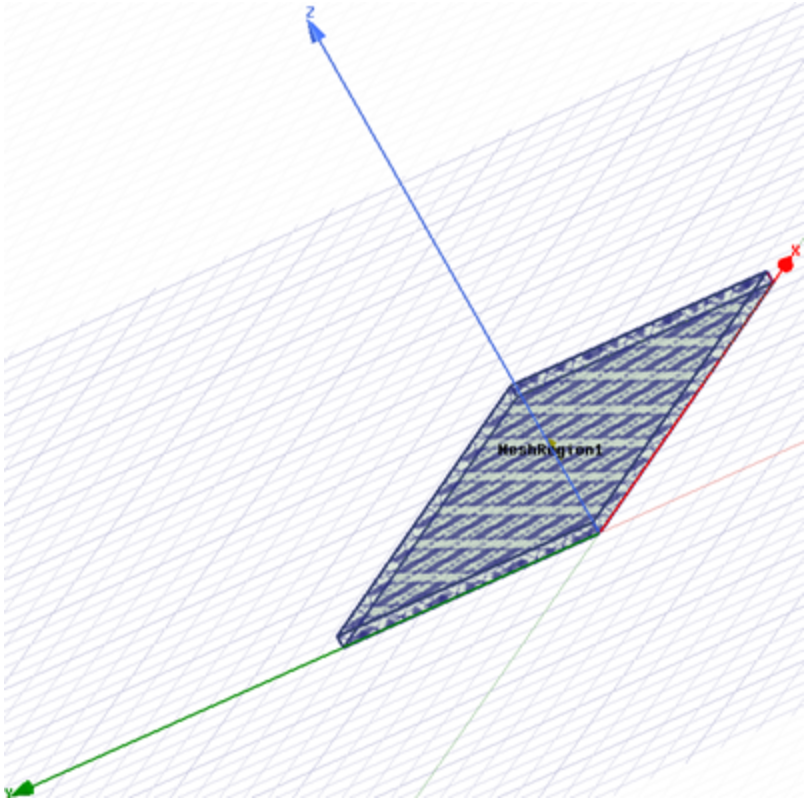
The relative coordinate system is created in the history tree under **Coordinate Systems** and is displayed in the **3D Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select the PCB geometry, right-click in the **3D Modeler** window, and select **Assign Mesh Region**. The mesh region is created on the relative coordinate system.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

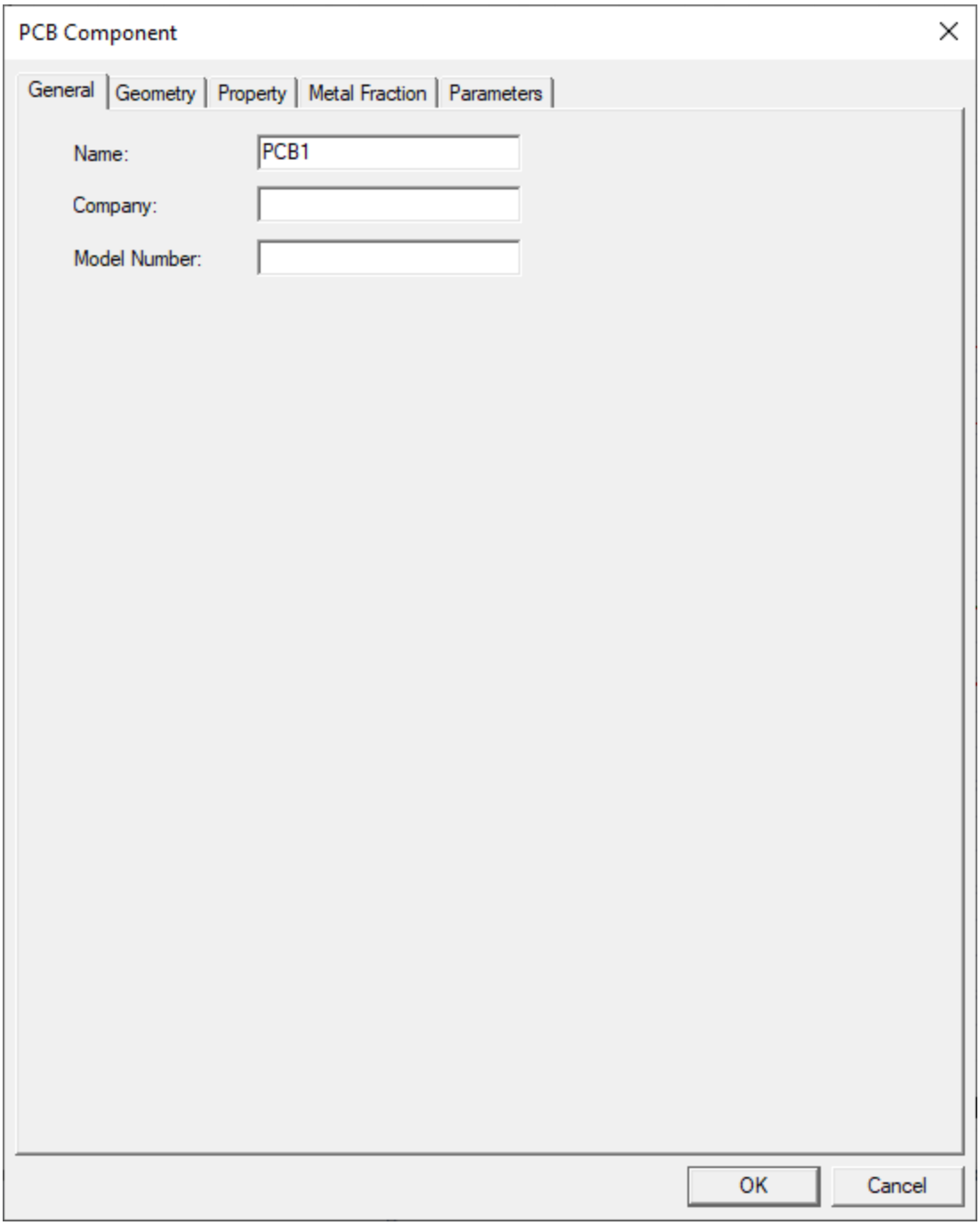


Creating PCBs

The Ansys Electronics Desktop uses a tabbed for the creation of a PCB component, including tabs to input **General**, **Geometry**, **Property**, **Metal Fraction**, and **Parameters** information. To create components using a wizard interface, access the **Icepak**>**Thermal** options in the **Options** dialog (**Tools**>**Options**>**General Options**) and select **Use Wizards for data input when creating new boundaries**.

Create a PCB by right-clicking on **3D Components** in the Project Manager and selecting **Create** > **PCB**. Then set up a link with an existing HFSS 3D Layout design and defining additional Icepak properties.

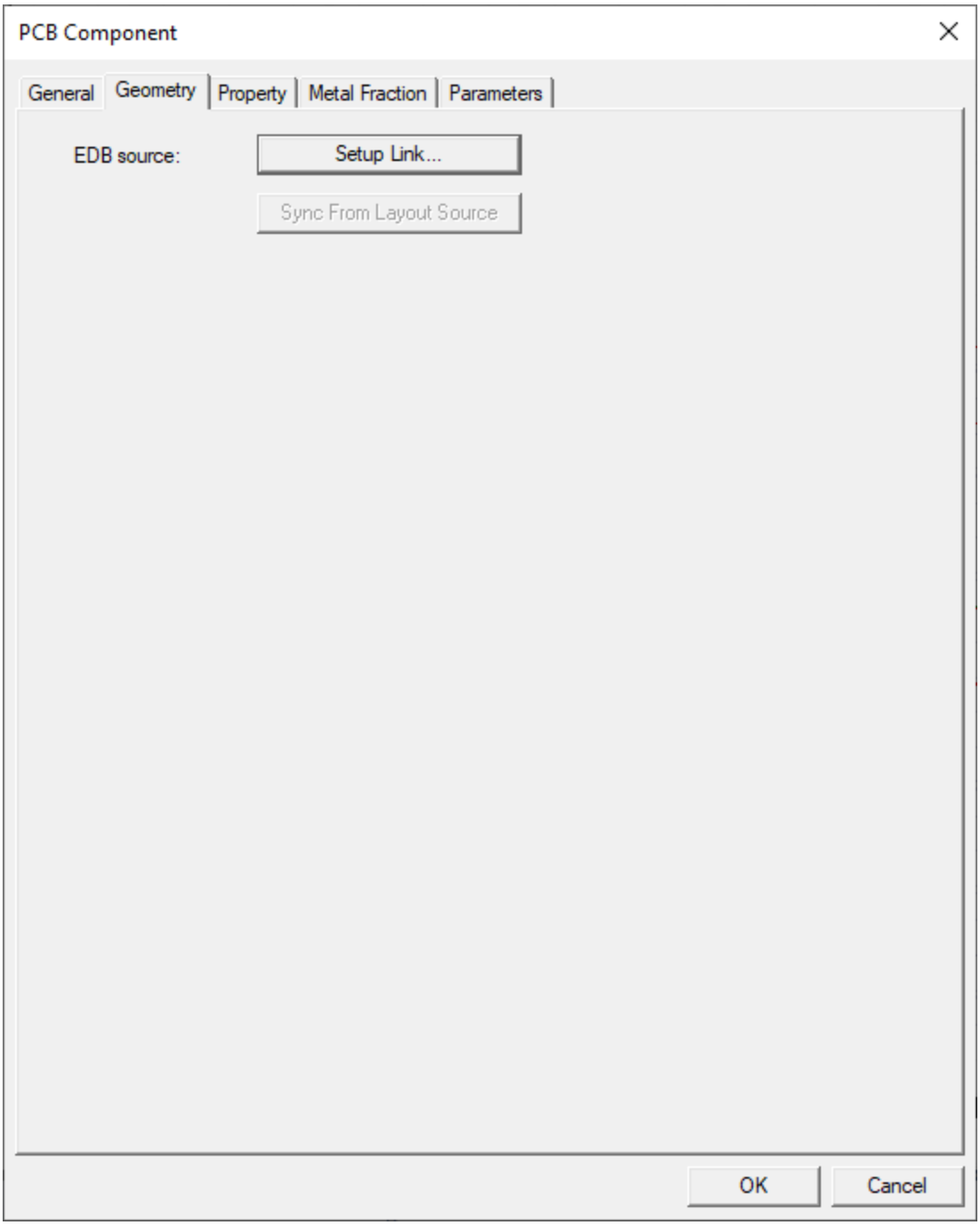
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **PCB Component** dialog box **General** tab contains the following general information for the selected object.

Name	The **Name** of the object appears in the **Project Manager** window and **3D Modeler** window.
Company	**Company** specifies the manufacturer of the PCB.
Model Number	**Model Number** specifies the model number of the PCB.

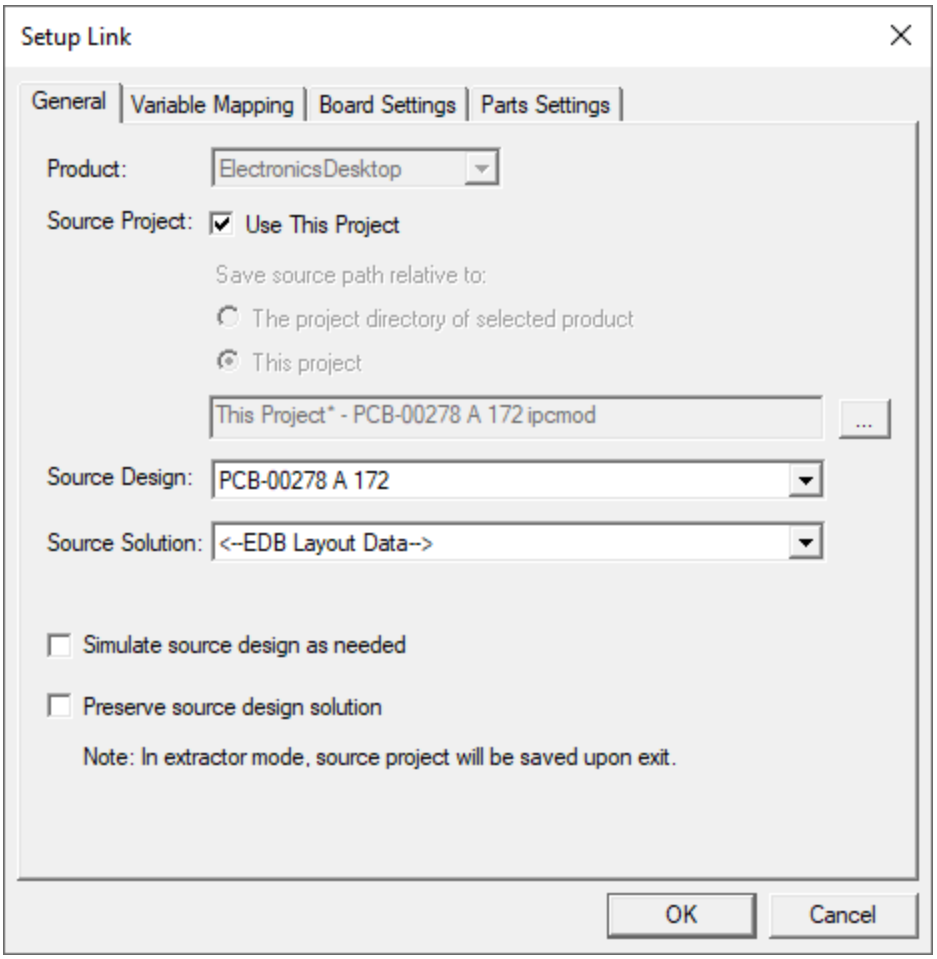
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **PCB Component** dialog box **Geometry** tab contains the following settings that define the shape of the PCB:

EDB source	Click **Setup Link** to open the **Setup Link** dialog box to specify a source 3D Layout design for the PCB geometry.
Sync Geometry From Source	Click **Sync Geometry From Source** to refresh the Icepak PCB component after making revisions to the source 3D Layout design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

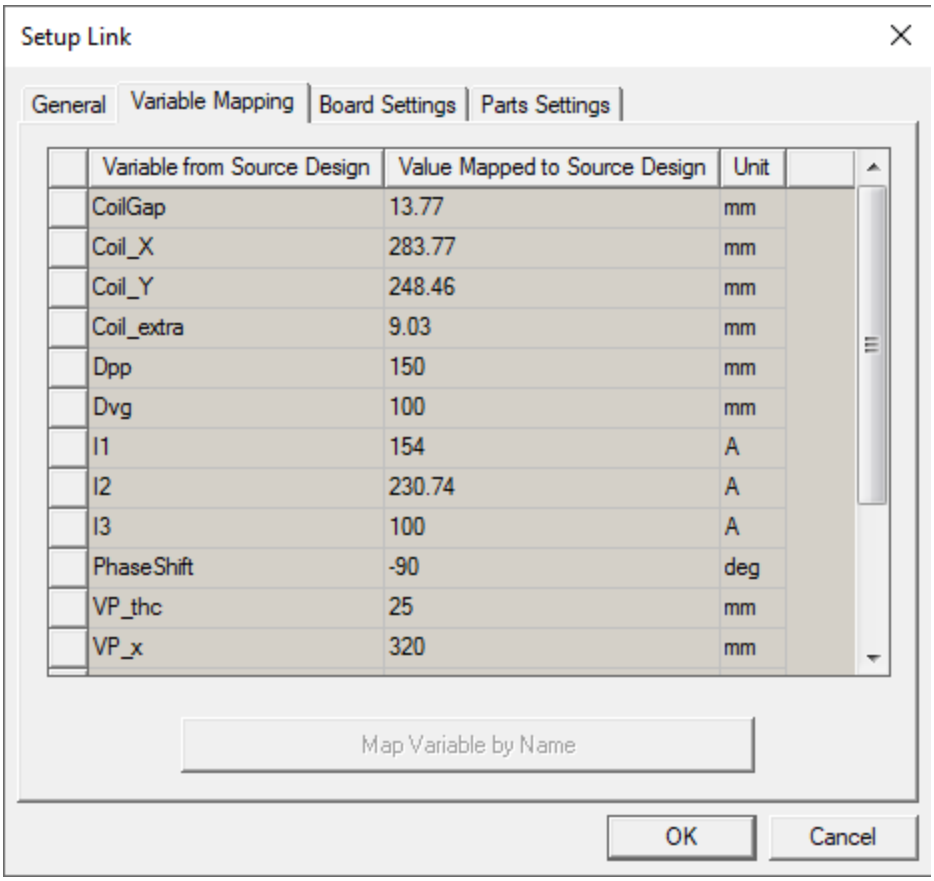


The **Setup Link** dialog box **General** tab contains the following settings that define the source project, design, and solution of the source 3D Layout design:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Tab	
Product	**Product** specifies the product in which you created the source design.
Source Project	**Source Project** specifies the project that has the source design. Select **Use This Project** to pick a design from the current working project.
Save source path relative to:	The option **The project directory of selected product** points to the default project directory location relative to the product that you selected from the drop-down menu. This is the same project directory that you set on the **General Options** dialog under **Tools> Options**. This option is useful especially, when you link across different products. The option **This project** points to the directory relative to where your current working project is located.
Source Design	**Source Design** specifies the source 3D Layout design. The **Source Design** drop-down menu lists all the designs in the specified **Source Project**.
Source Solution	**Source Solution** specifies the setup type from the **Source Solution** pull-down menu.
Simulate source design as needed	Select **Simulate source design as needed** if you modify the source design and you want an updated source design solution to be linked to the target design. If you select this option, make sure that you invalidate the cached link data in the target. To do this, right-click **Analysis** and select the option **Clear Linked Data** from the short-cut menu.
Preserve source design solution	**Preserve source design solution** is the last option on the **Setup Link** dialog box. When the source design is closed or it is not included in the project manager window containing the target design, the source design will not be saved.

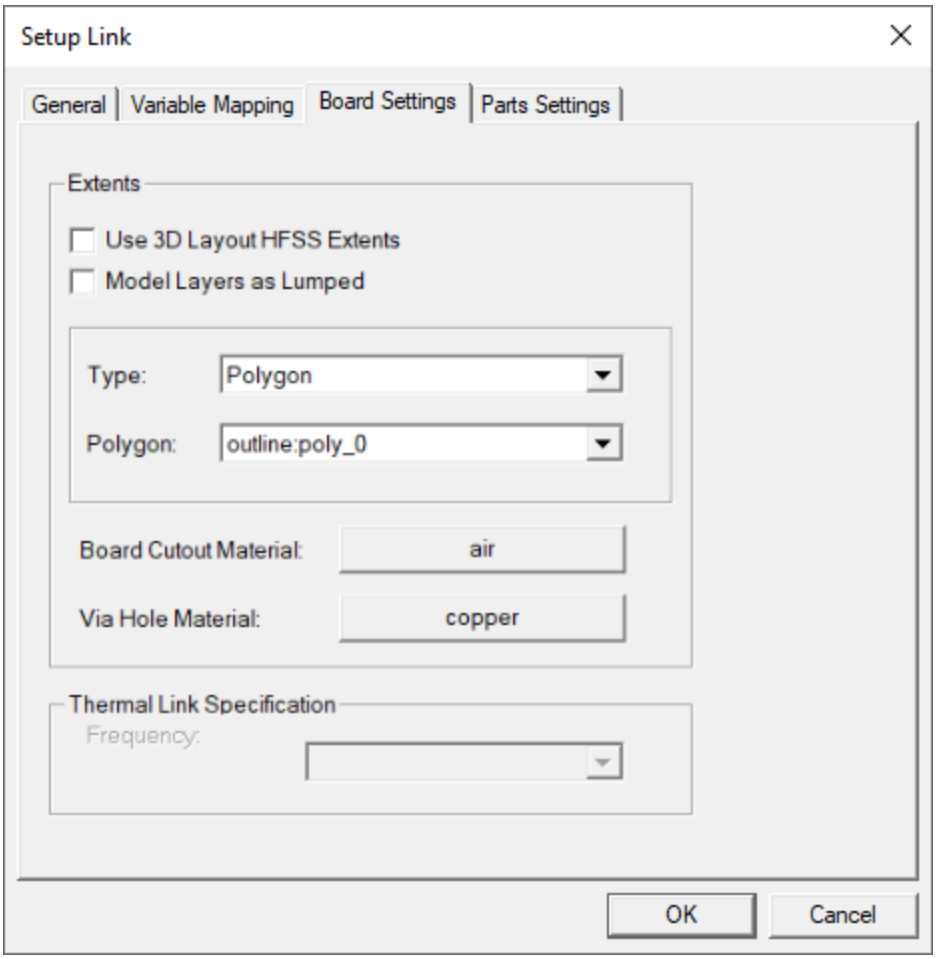
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Link** dialog box **Variable Mapping** tab contains variable information of the source 3D Layout design:

Variable Mapping Tab	
Variable from Source Design	**Variable from Source Design** displays the name of variables from the selected source design.
Value Mapped to Source Design	**Value Mapped to Source Design** displays the value assigned to the variable.
Unit	**Unit** displays the unit of the variable.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Link** dialog box **Board Settings** tab contains the following PCB settings:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Board Settings Tab: Extents	
Use 3D Layout HFSS Extents	**Use 3D Layout HFSS Extents** uses the source design's geometry to define the PCB geometry.
Model Layers as Lumped	**Model Layers as Lumped** creates a single piece of geometry to represent all PCB layers. While the thermal conductivity will vary across the surface of the PCB, it will be averaged and remain constant in the thickness direction.
Type	When **Use 3D Layout HFSS Extents** is disabled, **Type** specifies whether the geometry is displayed a **Bounding Box**, **Polygon**, or **SIwave per layer outline**. For **Polygon** types, cutouts in the PCB are assigned a fluid material (for example, air). **SIwave per layer outline** creates polygonal geometry for each PCB layer. When **Model Layers as Lumped** is enabled, **SIwave per layer outline** is not available.
Note: When linking a PCB to a DC IR SIwave setup in the source design, for **Type**: • If the design contains only a PCB and block components, select **Polygon**. • If the design contains a PCB with a package, select **SIwave per layer outline**.	
Polygon	When **Polygon** is selected for **Type**, **Polygon** specifies the number of sides the PCB geometry has.
Board Cutout Material	**Board Cutout Material** specifies the material assigned to the board cutout geometry, if it exists. Board cutout geometry exists if **Extents** is selected for **Polygon**, and that polygon contains void polygon(s). The cutout geometries are created from those void polygon(s). The default board cutout material is that of the global mesh region if present in the model. Otherwise, the default board cutout material is set to air-solid.
Via Hole Material	**Via Hole Material** specifies the material assigned to the via holes. The default material is set to copper.
Note: Conductivity for polygonal vias is calculated as follows: Conductivity = (via metal conductivity * (1.0 - via area ratio)) + (via hole material conductivity * via area ratio)	
Thermal Link Specification	
Frequency	**Frequency** specifies the frequency of the PCB.

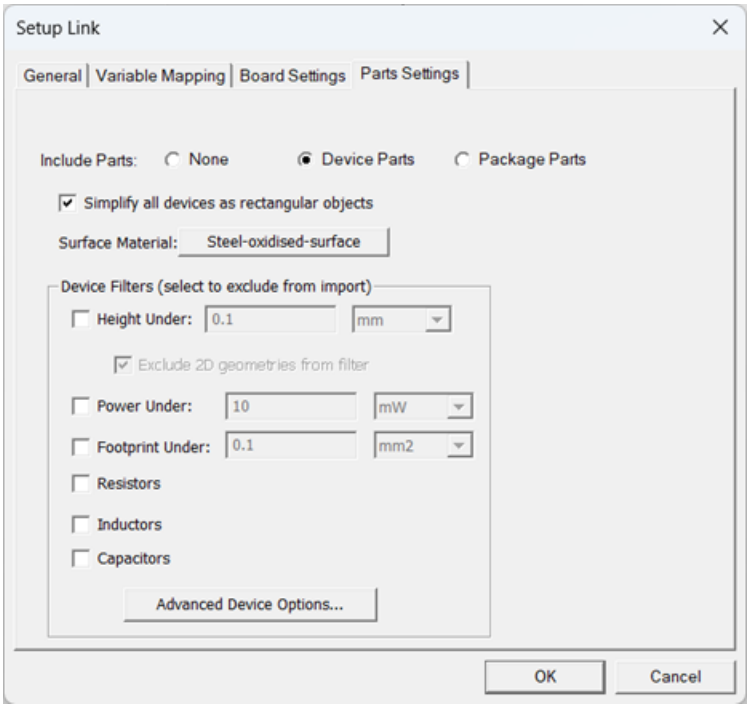
The **Setup Link** dialog box **Parts Settings** tab can be used to import device parts or package parts:

- **Device Parts**: Import component devices present on the PCB along with their thermal properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If **SIwave per layer outline** is enabled, devices on all layers of the stackup of the source design are imported. Otherwise, only devices on the top and bottom layers are imported.

- **Package Parts**: Import package components like the solderballs and solderbumps defined on the PCB substrate.

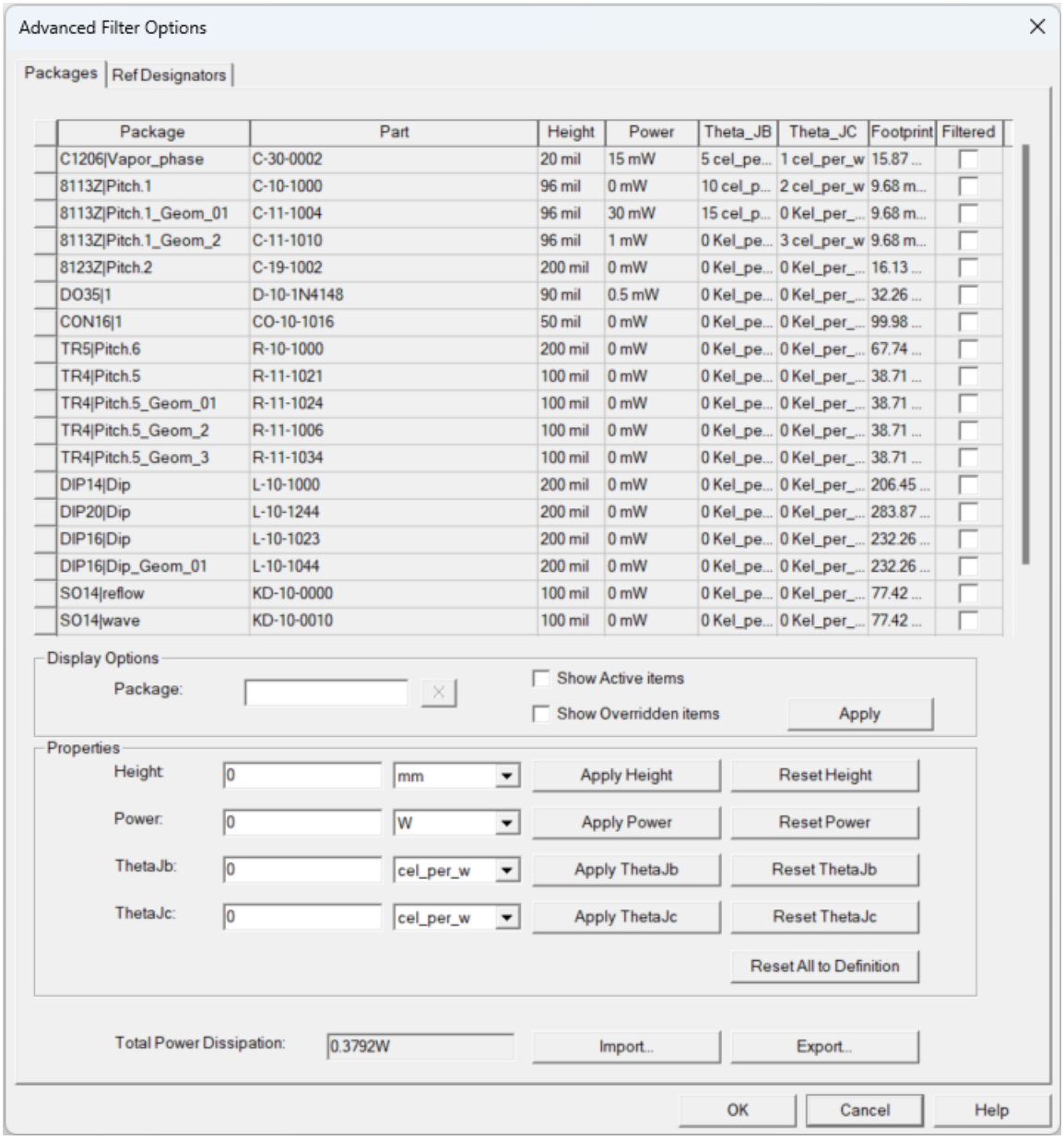


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include Electrical/Mechanical Parts	**Include Electrical/Mechanical Parts** specifies that the electrical and mechanical devices in the source design are imported. Select **Simplify all devices as rectangular objects** to import the devices as simple rectangular geometry.
Surface Material	Click the surface material button to open the **Select Definition** dialog box and select a surface material to assign to the devices.
Device Filters	
Height Under	**Height Under** specifies the minimum size of the components to be imported.
Power Under	**Power Under** specifies the minimum power of the components to be imported.
Footprint Under	**Footprint Under** specifies the minimum footprint of the components to be imported.
Resistors	**Resistors** specifies that resistors are not imported.
Inductors	**Inductors** specifies that inductors are not imported.
Capacitors	**Capacitors** specifies that capacitors are not imported.
Advanced Filter Options	Click **Advanced Filter Options** to edit the thermal properties of devices and enable advanced filtering options. **Note**: Expressions and variables are not supported for parameters specified in the **Advanced Filter vOptions** dialog box.

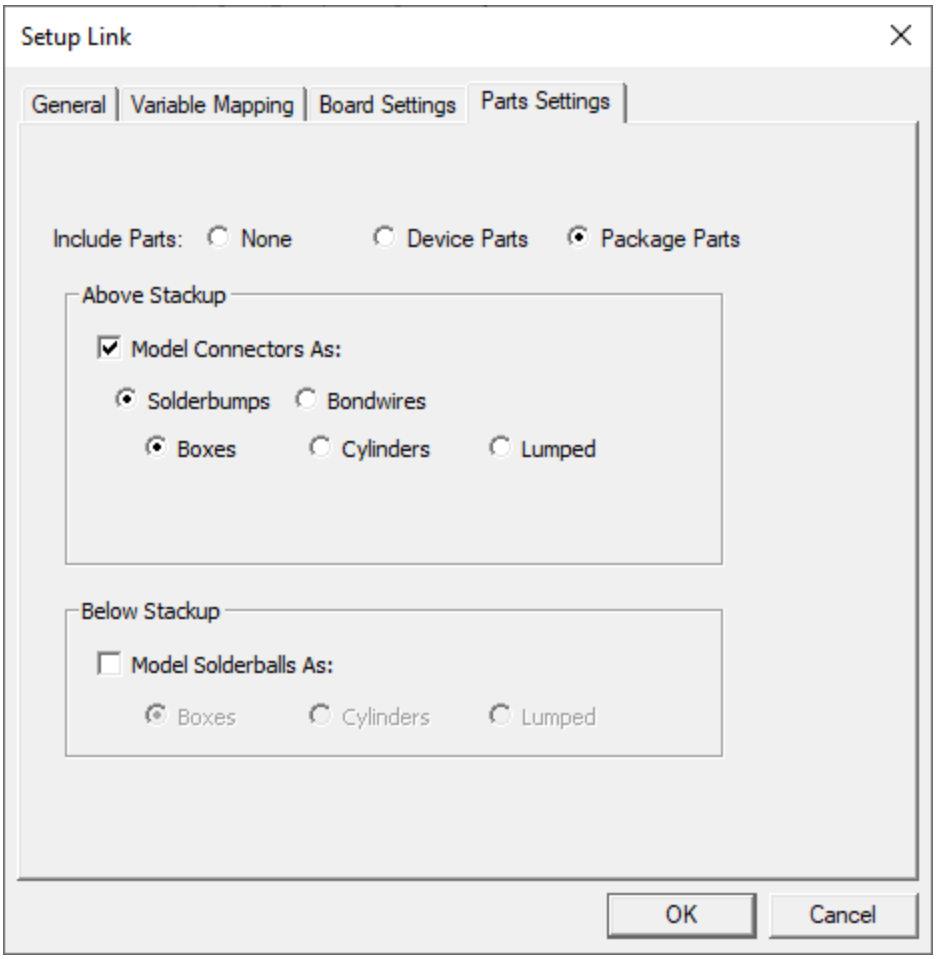
To edit layer properties, access the Edit Layers window from the Properties window in the HFSS 3D Layout design. See HFSS 3D Layout documentation for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For information on the **Advanced Filter Options**, see Import an IDF File.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



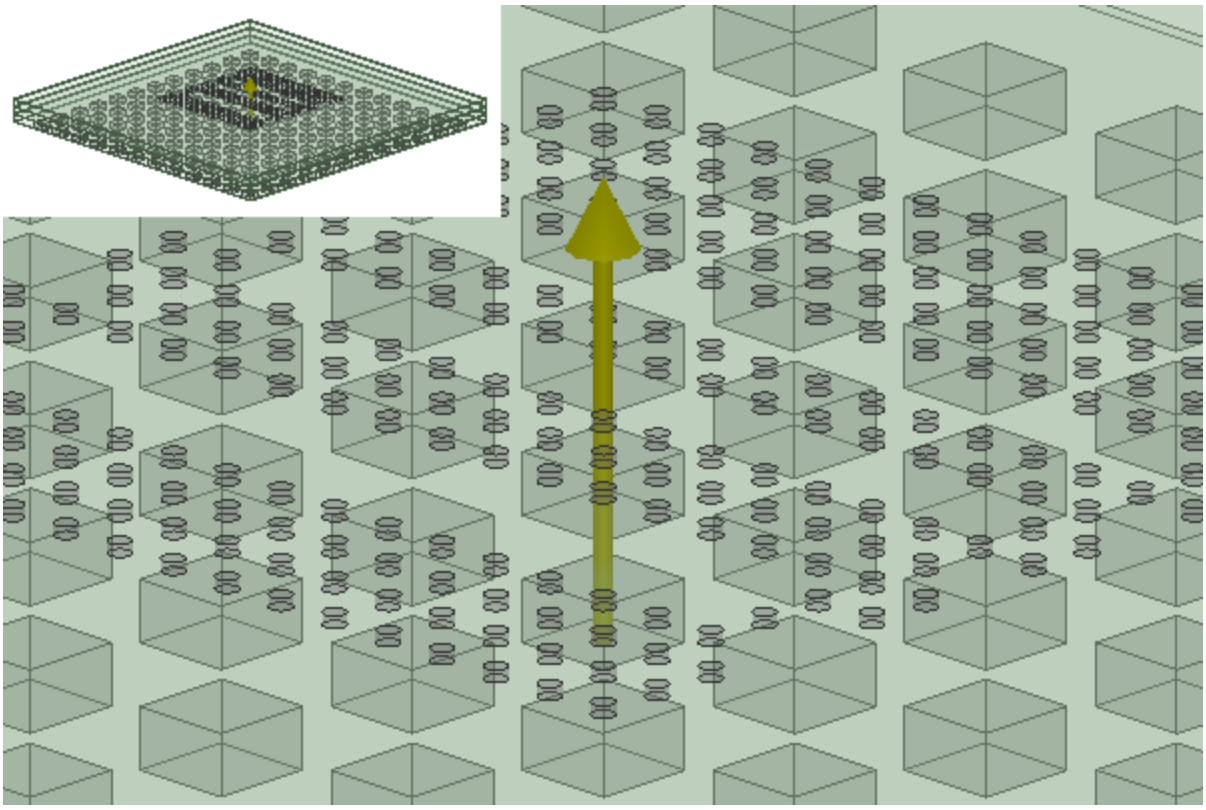
Above Stackup	
Model Connectors As	**Model As** specifies whether the connectors located above the stackup are modeled as solderbumps or bondwires. For **Solderbumps**, select whether the solderballs are modeled as box, cylinder, or lumped geometry. For **Bondwires**, select the **Bondwire Material** and specify the **Wire Diameter**.
Below Stackup	
Model As	**Model As** specifies whether the solderballs located below the stackup are modeled as box, cylinder, or lumped geometry.

To edit solderbump and solderball properties, use the Component Model window in the HFSS 3D Layout design. See HFSS 3D Layout documentation for more information.

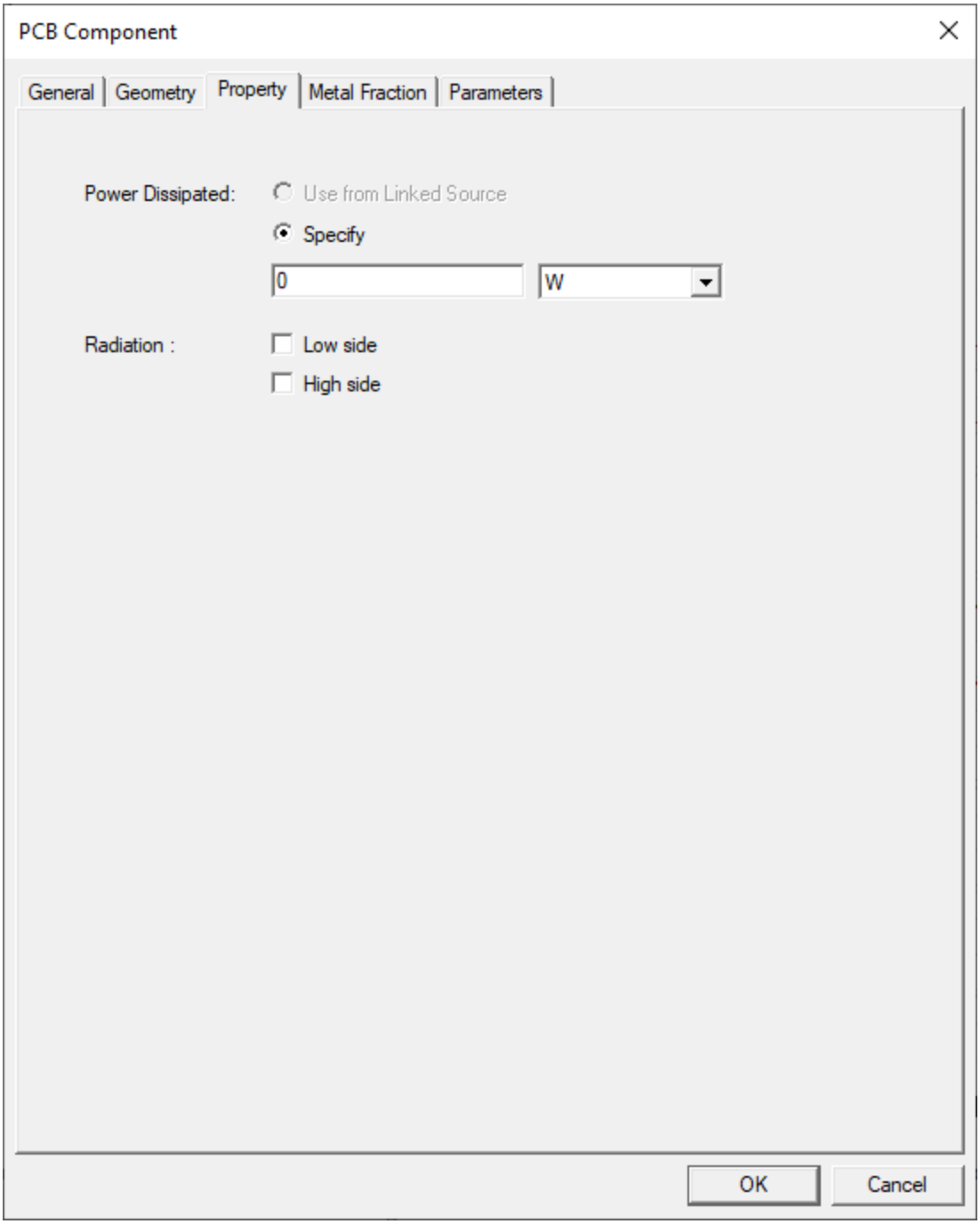
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Icepak only models HFSS 3D Layout solderballs above the stackup (solderbumps) and/or below the stackup (solderballs). Icepak does not model solderballs defined within the stackup in HFSS 3D Layout.

In the following example, the solderbumps are modeled as cylinders and the solderballs as boxes.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



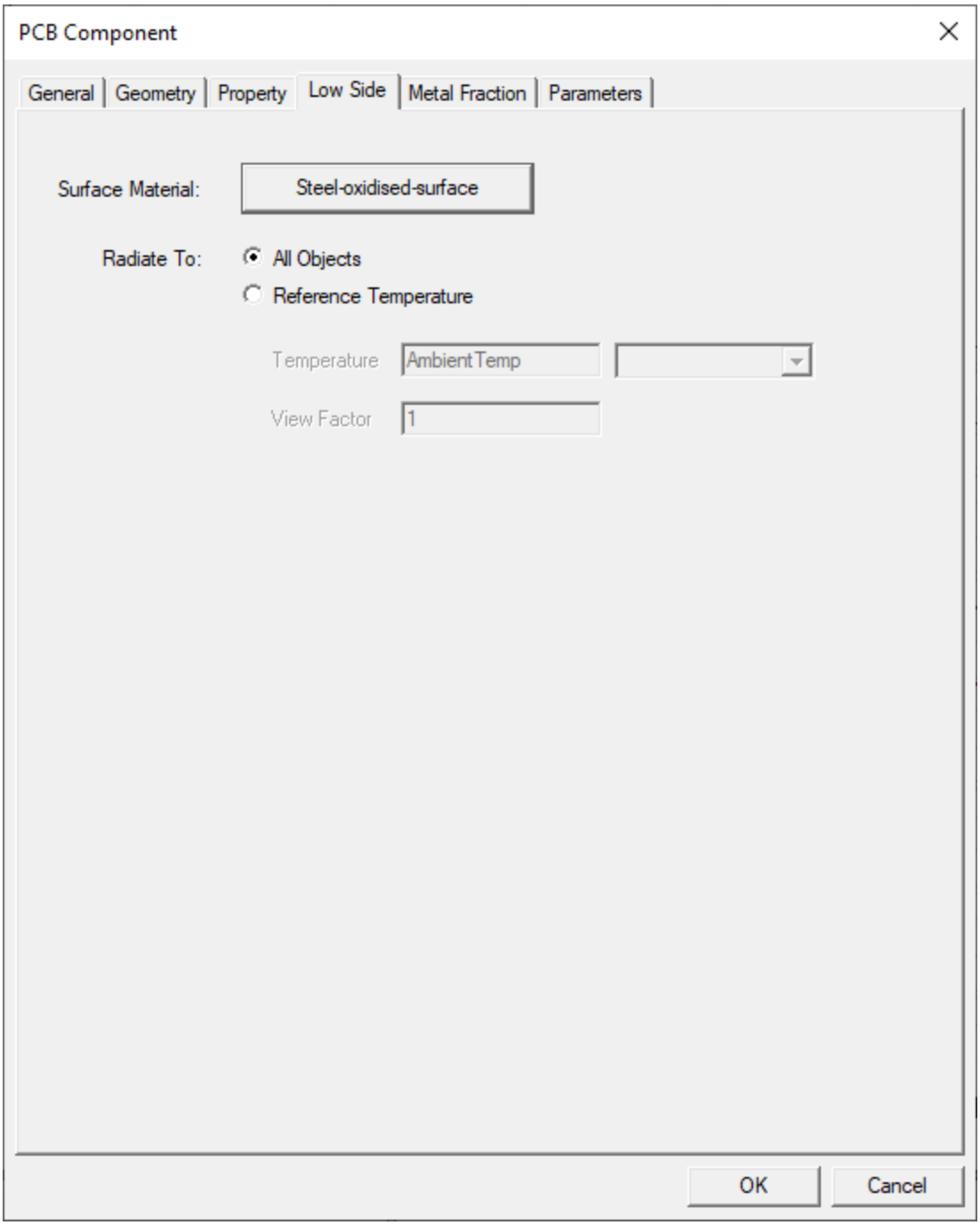
The **PCB Component** dialog box **Property** tab contains the following settings that define the thermal properties of the PCB:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Power Dissipated	**Power Dissipated** specifies the total power dissipated by the PCB. If the source HFSS 3D Layout design was solved using HFSS 3D Layout and consists of EM losses, you can select **Use from Linked Design**, or you can select **Specify** and enter a uniformly dissipated total power.
Radiation	Select **Low side** and/or **High side** to specify different physical characteristics for each side of the PCB.

If you select **Low side** and/or **High side**, the **Low Side** and/or **High Side** tabs contains the following thermal information.

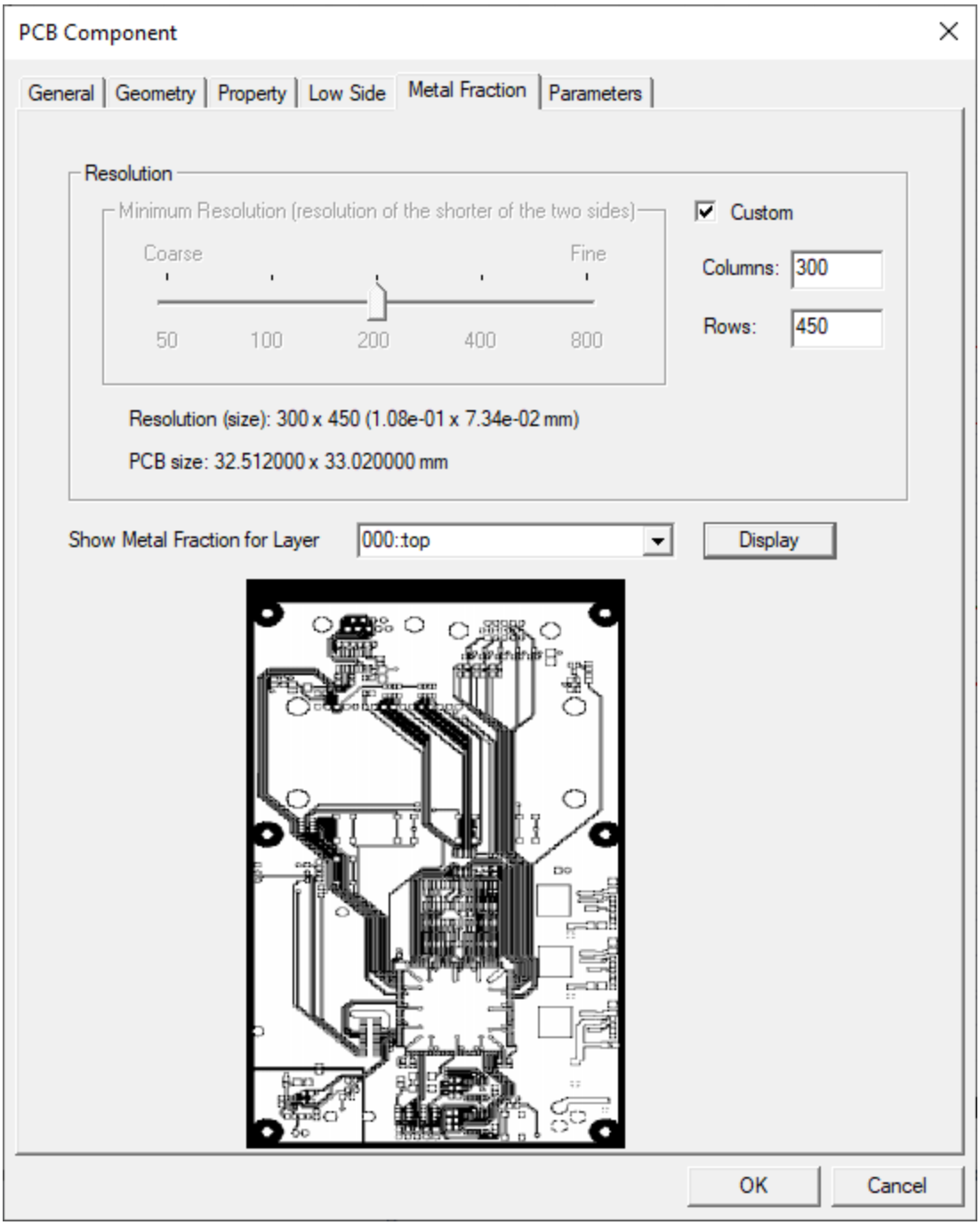
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Material	**Surface Material** specifies the material type for the side of the plate using the **Select Definition** dialog box.
Radiate To	Select **All Objects** or **Reference Temperature**. **All Objects** specifies that the object or surface exchanges radiation with all other objects and surfaces also specified as subject to radiation with heat transfer. **Reference temperature** specifies a fixed temperature to which the selected sides of the object radiate heat. Enter a **Temperature** and **View factor**, which is the proportion of radiation that leaves the plate and contacts surrounding objects. The default view factor is 1.0 and the value of the ambient temperature is the **Temperature** specified in the **Icepak Design Settings** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **PCB Component** dialog box **Metal Fraction** contains the following settings for defining the PCB grid resolution and previewing the metal fraction information of each layer of the PCB.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resolution	The **Resolution** slider bar defines the resolution of the smaller of the width and height of the PCB outline. The resolution on the larger side is derived proportionally.
Custom	The **Custom** check box allows you to specify the resolution by the number of **Columns** and **Rows**. See "PCB Grid Resolution" below for more information.
Show Metal Fraction for Layer	The **Show Metal Fraction for Layer** drop-down menu displays all of available layers of the PCB. Select a layer and click **Display** to show an image of the metal fraction for the selected layer and resolution.

PCB Grid Resolution

The grid resolution that is used to create the thermal conductivity distribution of the board can be customized. Depending on the trace resolution and the computational costs desired, you can change the values for the columns and rows to receive optimum results. The size of the grid is in parentheses next to the column and row count.

> **Note:** When linked from an HFSS 3D Layout design, circles and arcs in traces are approximated as polygons with a maximum of 12 points in an Icepak design.

The following points outline best practices for balancing accuracy and cost of computing the cell by cell orthotropic thermal conductivity:

- For board-level simulations, enter column and row counts that yield at most a grid size equal to the minimum trace width or minimum via diameter. If possible, use column and row counts that yield a grid size equal to half the minimum trace width or half the minimum via diameter.
- For system-level simulations, enter column and row counts to yield at most a grid size that equals four to eight times the minimum trace width or minimum via diameter.
- For all models, the CFD mesh on the board should equal four to eight times the grid size.
- Column and row counts or CFD meshes that significantly deviate from the above guidelines can adversely affect accuracy.

> **Note:** When using the slider bar to set the grid resolution, the Ansys Electronics Desktop application uses the aspect ratio to determine the resolution of the longer side. At low resolution, you may observe a slight grayscale pixel difference and asymmetry at the trace boundaries. Also, you may observe slight differences between using the slider bar and custom grid resolution settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: You may observe slight differences between Classic Icepak and Icepak in the Electronics Desktop as their metal fraction computation and display methods differ.



The **PCB Component** dialog box **Parameters** tab contains the following information about the component variables defined for the object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Creating a PCB

> **Note:** You can define component variables in Icepak native 3D components. See Defining Native Component Instance Parameters for more information.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **PCB**.
2. On the **PCB Component** dialog box **General** tab, enter following information:
   - **Name**
   - **Company** (if known)
   - **Model Number** (if known)
3. Click the **Geometry** tab and then click **Setup Link**.
4. On the **Setup Link** dialog box **General** tab, select **Edit Link** radio button and do the following:
   - Select a **Product**.
   - Select **Use This Project**.

     > **Note:**
     >
     > Currently, only links between designs within the same project are supported. **Use This Project** is enabled by default.

   - Select a **Source Design**.
   - Select a **Source Solution**.
5. On the **Variable Mapping** tab, verify the variable settings and edit them if needed.
6. On the **Board Settings** tab, do the following:
   - Under **Extents**, enable **Use 3D Layout HFSS Extents** or select a **Type** and, if needed, **Polygon**.
   - Under **Thermal Link Specification**, select a **Frequency**.
7. On the **Part Settings** tab, if importing device or package parts, select **Device Parts** or **Package Parts**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Device Parts**: If importing electrical and/or mechanical devices, select **Include Electrical/Mechanical Parts** and, if needed, **Simplify all devices as rectangular objects**. Under **Device Filters**, enable the desired filters to be applied during import.
- **Package Parts (Above Stackup)**: If modeling solderbumps or bondwires, enable **Model Connectors As** and select **Solderbumps** or **Bondwires**. For solderbumps, select the desired type of geometry. For bondwires, select the **Bondwire Material** and specify the **Wire Diameter**.
- **Package Parts (Below Stackup)**: If modeling solderballs, enable **Model Solderballs As** and select the desired type of geometry.

> **Note:**
>
> Only bondwires that are part of a component group in 3D Layout are supported.

1. Click **OK** to close the **Setup Link** dialog box.
2. On the **PCB Component** dialog box **Property** tab, do the following:
   - Select a **Power Dissipated** option.
   - If needed, select to specify **Low side** and/or **High side** thermal properties.
3. If you selected either **Low side** or **High side**, do the following on the **Low side** and/or **High side** tabs:
   - Click the button next to **Surface Material** to open the **Select Definition** dialog box. On the **Materials** tab, select the material for the side of the plate and click **OK**
   - For **Radiate To**, click **All Objects** or **Reference Temperature**. If you selected **Reference Temperature**, enter a **Temperature** and **View Factor**.
4. On the **Metal Fraction** dialog box, do the following:
   - Use the **Minimum Resolution** slider bar to set the minimum resolution for the calculation of metal fractions.

     > **Note:**
     >
     > You can select **Custom** and enter the number of columns and rows for a custom grid resolution. See "PCB Grid Resolution" on page 10-55 for more guidance on custom grid resolution.

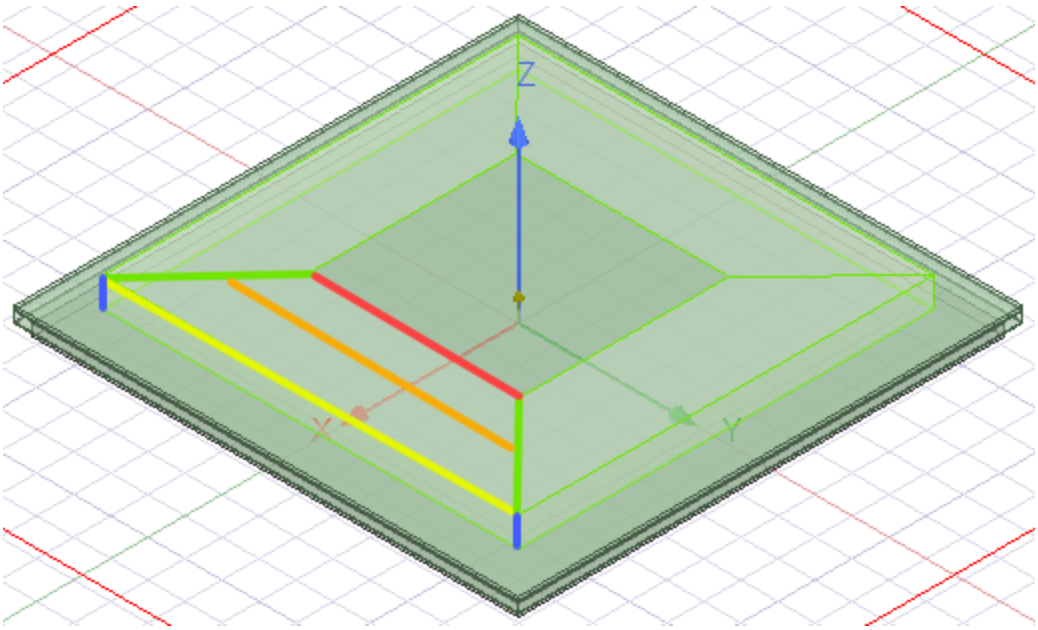
   - Select a layer from the **Show Metal Fraction for Layer** drop-down menu and click **Display** to show a preview image of the selected layer's metal fractions. Repeat this step for the desired layers of the PCB.
5. If you defined any component variables, click the **Parameters** tab to review the variable (s).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If needed, select **Map instance parameters to variables** and select an option from the drop-down list.
7. Click **OK**.

Bondwire Sheets

When importing a PCB with bondwires above the stackup, Icepak simplifies the bondwires by creating up to eight sheets (rectangular and trapezoidal). The geometry created in Icepak is based on properties defined in HFSS 3D Layout.



In the image above, the red segment of the bondwire sheet is the start length, the yellow segment is the end length, and the orange segment represents the average length between the start and end lengths. The blue segments represent the length of the vertical drop between the bondwire's start and end points. The following image illustrates a cross-section of bondwire sheet geometry:



Bondwire sheets are composed of the bondwire material and surrounding material. To define the material composition for each bondwire sheet, Icepak calculates what fraction of a cross-section is composed of bondwire material. First, Icepak uses the following equation to calculate the total area of the cross-section.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total area = bondwire diameter * length, where

length = the length (rectangular sheets) or the average of the start and end lengths (trapezoidal sheets)

With the total area of the cross-section defined, Icepak uses the following equation to determine what fraction of the total area is composed of bondwire material.

bondwire area/total area, where

bondwire area = number of bondwires per side * πr^2

Importing Surface Materials from SIwave

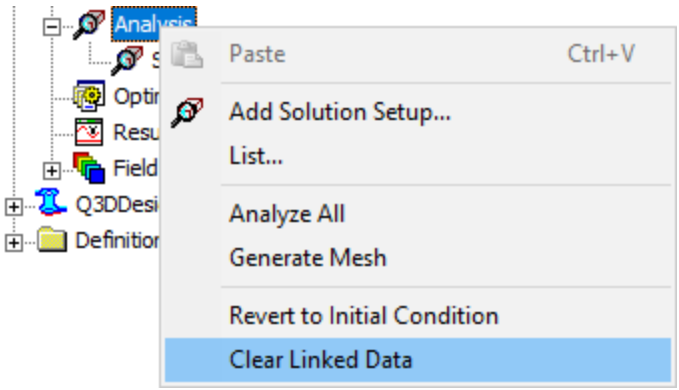
When importing a PCB with from an SIwave source design, consider the following regarding the assigned surface materials.

- If no surface material is defined for an imported device, the default surface material defined on the **Package Settings** tab is assigned.
- If an imported device has an existing surface material with zero surface emissivity, it is replaced with default surface material defined on the **Package Settings** tab is assigned.
- If an imported device's surface material does not exist in Icepak's material library, it is replaced with default surface material defined on the **Package Settings** tab is assigned.

Clearing Linked Data

Data linked from other design types (e.g., EM Loss data, metal fraction data) can be removed from the design containing the data. There are two ways to clear the linked data.

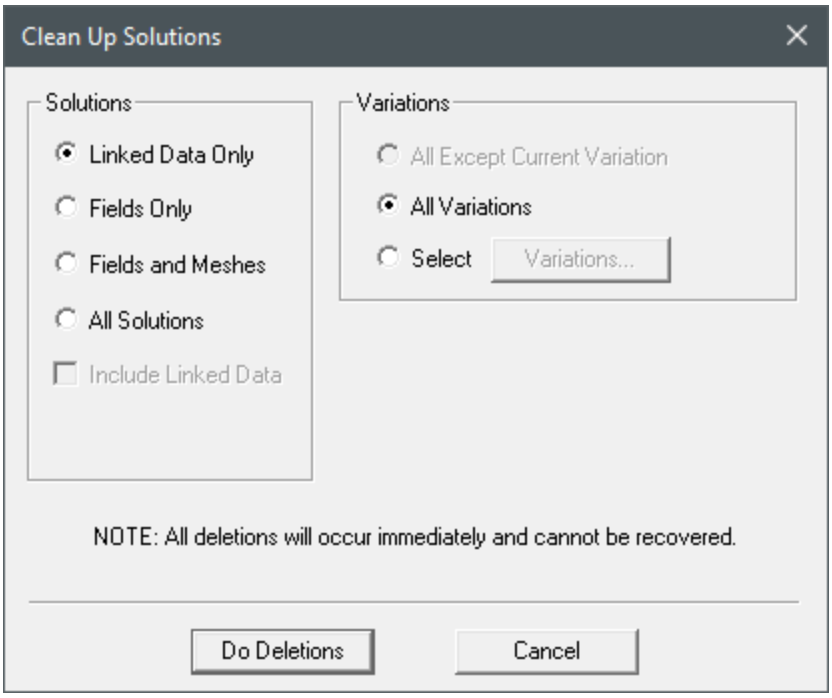
- In the **Project Manager**, right-click **Analysis** and select **Clear Linked Data**.



- From the **Icepak** menu, select **Results** > **Clean Up Solutions**. In the **Clean Up Solutions** dialog box, select **Linked Data Only** or another option and **Include Linked**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data. Then click **Do Deletions**.

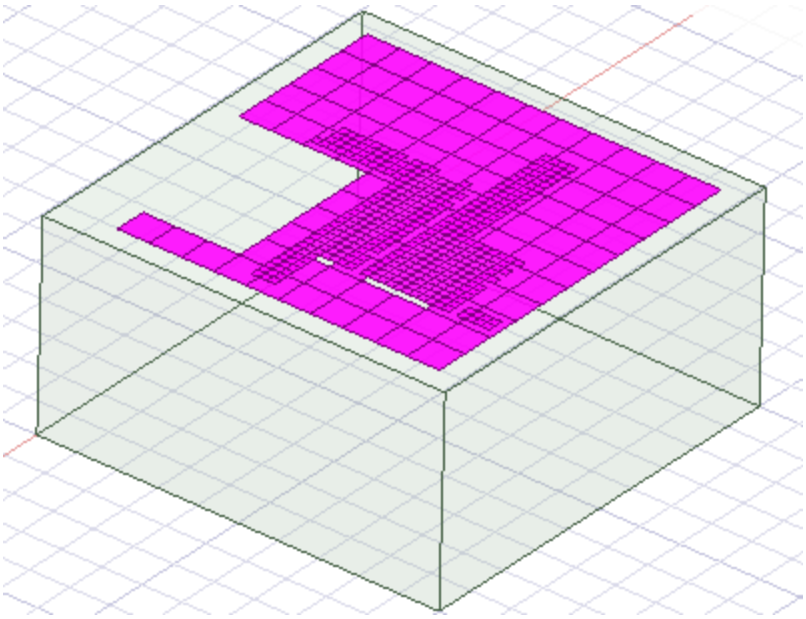


Chip Thermal Models (CTMs)

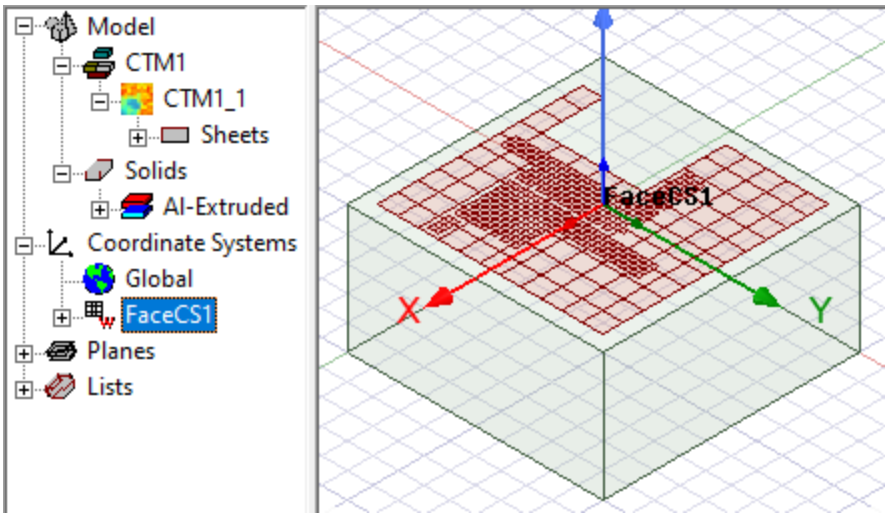
CTMs are components generated from Apache RedHawk powermap files. CTMs are imported as two-dimensional sheets representing the geometry of the chip powermap. After creating the CTM, you can display the imported power data. See CTM Visualization for more information.

> **Note:** Plain text files (.txt) can also be used to import CTMs. See "Text File Format" on page 10-64 for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A face coordinate system is created at the center of the selected face.

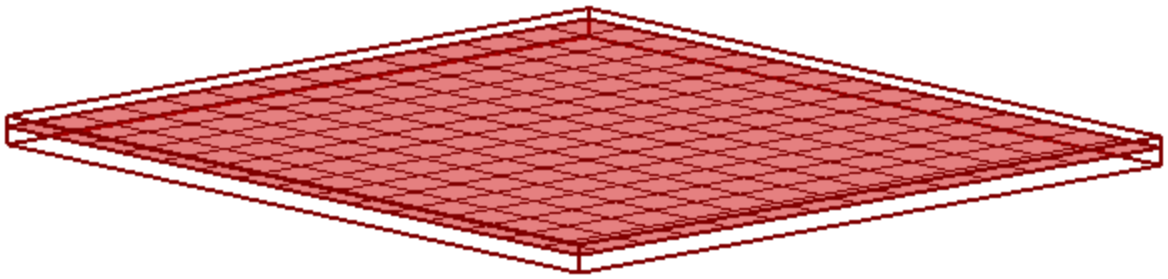


CTMs with Substrate and Routing Layers

If the CTM import files contain substrate and routing layer information, the CTM sources are created in between the bottom substrate layer and the top routing layer. The thickness of these layers can be viewed in the CTM properties.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The visualization of the substrate and routing layers and the display of their thickness properties are a beta feature. See Beta Options in General Options: Desktop Configuration for more information on enabling beta features.



Properties

Name	Value	Unit	Evaluated
Type	CTM		
Geometry			
CTM import file	C:\Users		
Rotation	0deg		
Direction	Up		
Density number	500		500
Resolution	4		4
Do not create Face CS			
Security	None		
Flip Stackup	No		

General Parameters Icepak Visualization

Note: Icepak imports and displays CTM substrate and routing layer data that are encrypted in the source powermap file. Unencrypted ASCII formatted substrate and routing layer data are not supported.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Text File Format

The plain text file format (.txt) is supported for importing chip thermal models into Icepak. The following image displays the CTM plain text file format.

> **Important:** In each unit definition, you must include a space after the unit to successfully import the CTM.

> **Note:** Irradiance units are used for heat generated in a solid object and, therefore, should be used for heat flux for CTMs.

The following images display the text file format with and without the #FloorPlans section.

```
#DistanceUnit: mm
#TemperatureUnit: cel
#Power/HeatFluxUnit: W
#CellLocations
0.0 0.0 1.0 1.0
0.0 0.0 -1.0 -0.5
0.0 1.0 1.0 1.5
#Temperatures
10 20 30 40 50 60
#PowerMapOfCells
0.1 0.15 0.2 0.25 0.3 0.35
0.1 0.13 0.16 0.19 0.22 0.25
0.1 0.12 0.14 0.16 0.18 0.2
```

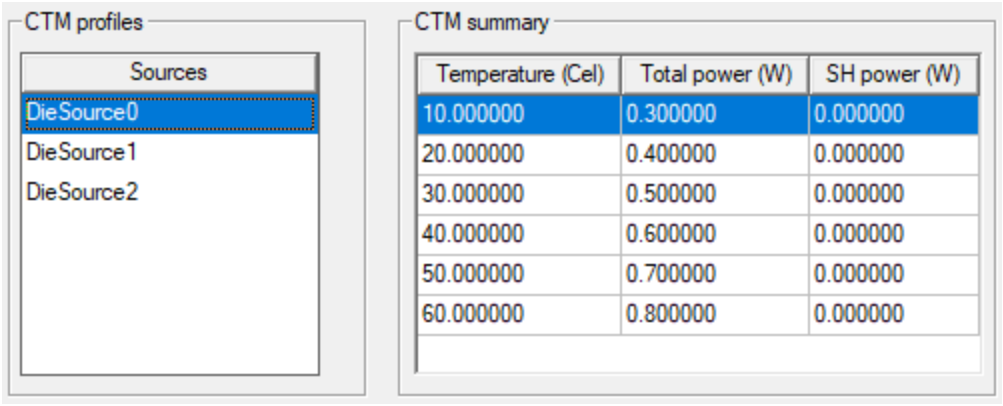

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#DistanceUnit: mm
#TemperatureUnit: cel
#Power/HeatFluxUnit: W
#CellLocations
Tile 0 0 0.18 0.18
Tile1 0 0.18 0.18 0.36
Tile2 0 0.36 0.18 0.54
Tile3 0 0.54 0.18 0.72
Tile4 0 0.72 0.18 0.9
P1 0 -0.3 0.3 -0.3 0.3 -0.6 0.2 -0.5 0 -0.5 0 -0.3
P2 0.4 0.3 0.7 0.3 0.7 0.6 0.5 0.7 0.3 0.7 0.4 0.3
#Temperatures
25 30.0 35.0 40.0 45.0 50.0 55.0 60.0 65.0 70.0 75.0
80.0 85.0 90.0 95.0 100.0 105.0 110.0 115.0 120.0 125.0
130.0 135.0 140.0 145.0 150.0 155.0 160.0 165.0 170.0
175.0 180.0 185.0 190.0 195.0 200.0 205.0
#PowerMapOfCells
2.15e-09 2.59284784874e-09 3.12691161242e-09
3.77097955695e-09 4.54770987529e-09 5.48442779852e-09
6.61408689252e-09 7.97642835844e-09 9.61937912082e-09
1.16007378882e-08 1.3990208501e-08 1.68718521001e-08
2.03470443823e-08 2.45380419789e-08 2.9592283422e-08
3.56875760046e-08 4.30383510093e-08 5.19032073617e-08
6.25940090932e-08 7.54868566609e-08 9.10353180936e-08
1.09786385432e-07 1.32399717808e-07 1.59670848135e-07
1.92559169811e-07 2.32221688e-07 2.80053722865e-07
3.37737996679e-07 4.07303831687e-07 4.91198541289e-07
5.92373526085e-07 7.14388103608e-07 8.61534724467e-07
1.03898997997e-06 1.25299671369e-06 1.51108364352e-06
1.82233022063e-06
#FloorPlans
F1 0 0 0.4 0.0 0.4 0.4 0 0.4 0 0
F2 -0.2 0.4 0.5 1
```

#DistanceUnit	**#DistanceUnit** specifies the unit of measure for the cells defined under **#CellLocations**.
#TemperatureUnit	**#TemperatureUnit** specifies the thermal unit for the temperatures defined under **#Temperatures**.
#Power/HeatFluxUnit	**#Power/HeatFluxUnit** specifies the unit for the power or heat flux values defined under **#PowerMapOfCells**.
#CellLocations	Under **#CellLocations**, list the x y coordinate pairs for tiles or polygons. For tiles, specify start and end point coordinate pairs

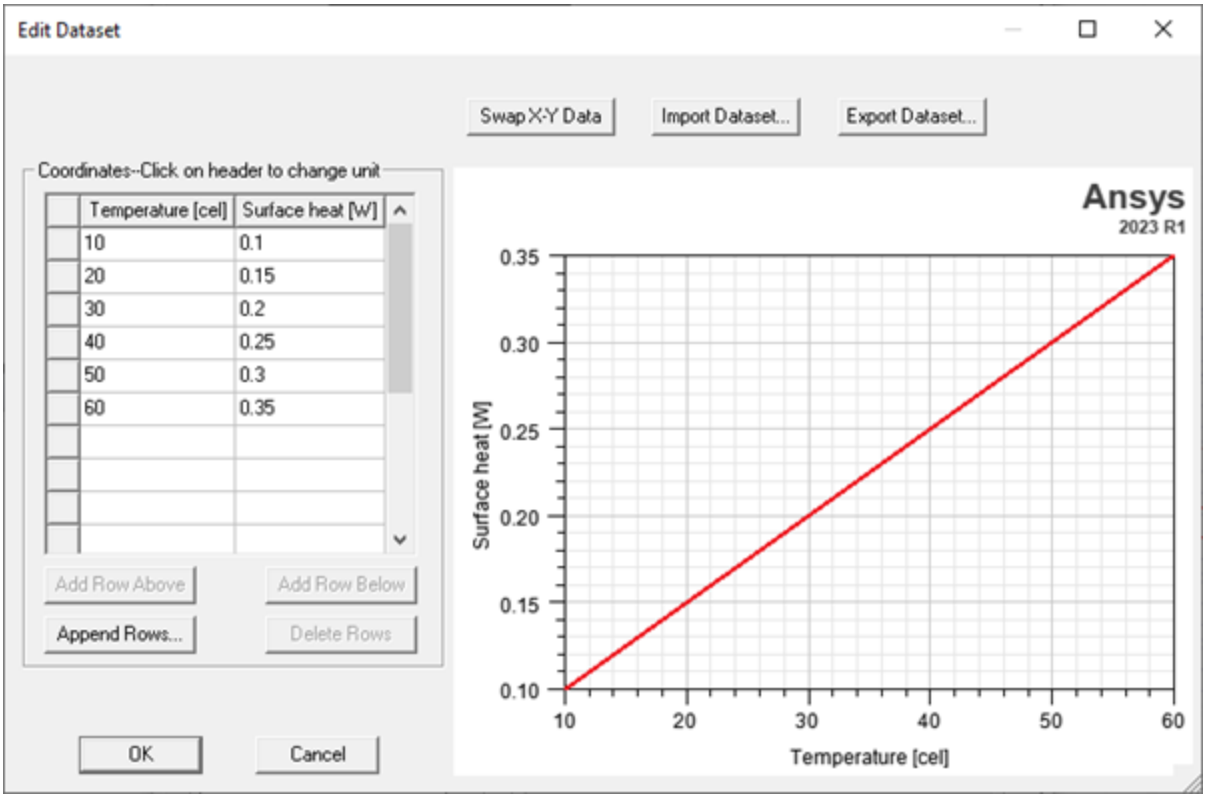
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

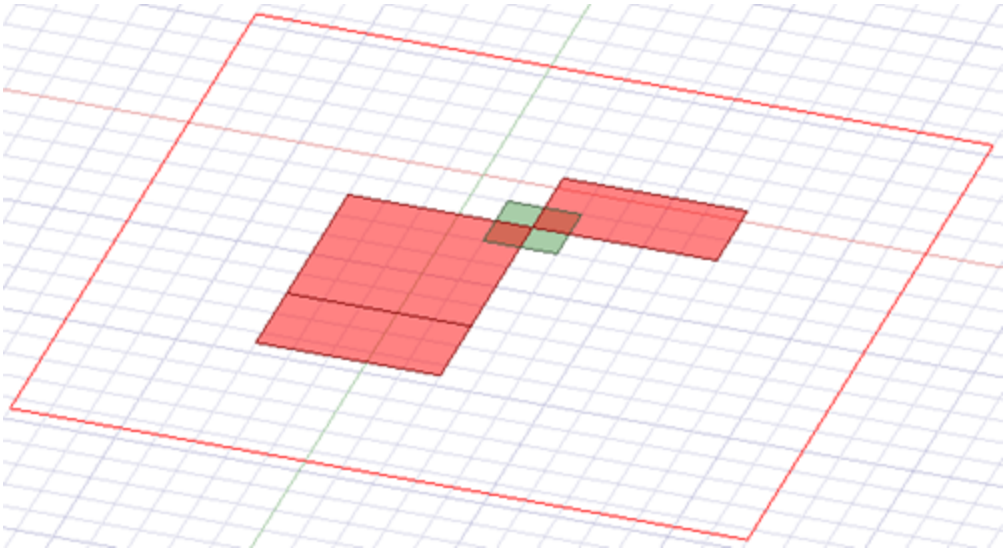
	of each cell separated by a space. For polygons, specify coordinate pairs for each vertex of the polygon. Before the list of coordinate pairs, you can enter a name for each cell. Each row in the text file defines a cell.
#Temperatures	Under **#Temperatures**, list the temperatures used in the CTM profile dataset.
#PowerMapOfCells	Under **#PowerMapOfCells**, list the power or heat flux values used in the CTM profile dataset. The number of power maps defined should equal the total number of cells listed under the **#Power/HeatFluxUnit** section. The order of the power map values corresponds to the order of cells.
#FloorPlans	Under **#FloorPlans**, list the x y coordinates pairs for floor plans. To create a square or rectangular floor plan, specify start and end coordinate pairs. For a polygonal floor plan, specify coordinate pairs for each vertex of the polygon. Floor plans are imported as non-model 2D sheet geometry to coincide with cells with powermaps. Floor plan geometry can be selected and used to display thermal data on field overlays or in the Fields Summary report. Before the list of coordinate pairs, you can enter a name for each floor plan. Each row in the text file defines a floor plan.



Note: Under **CTM Profiles**, double-click on each source to view its temperature-dependent power dataset.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



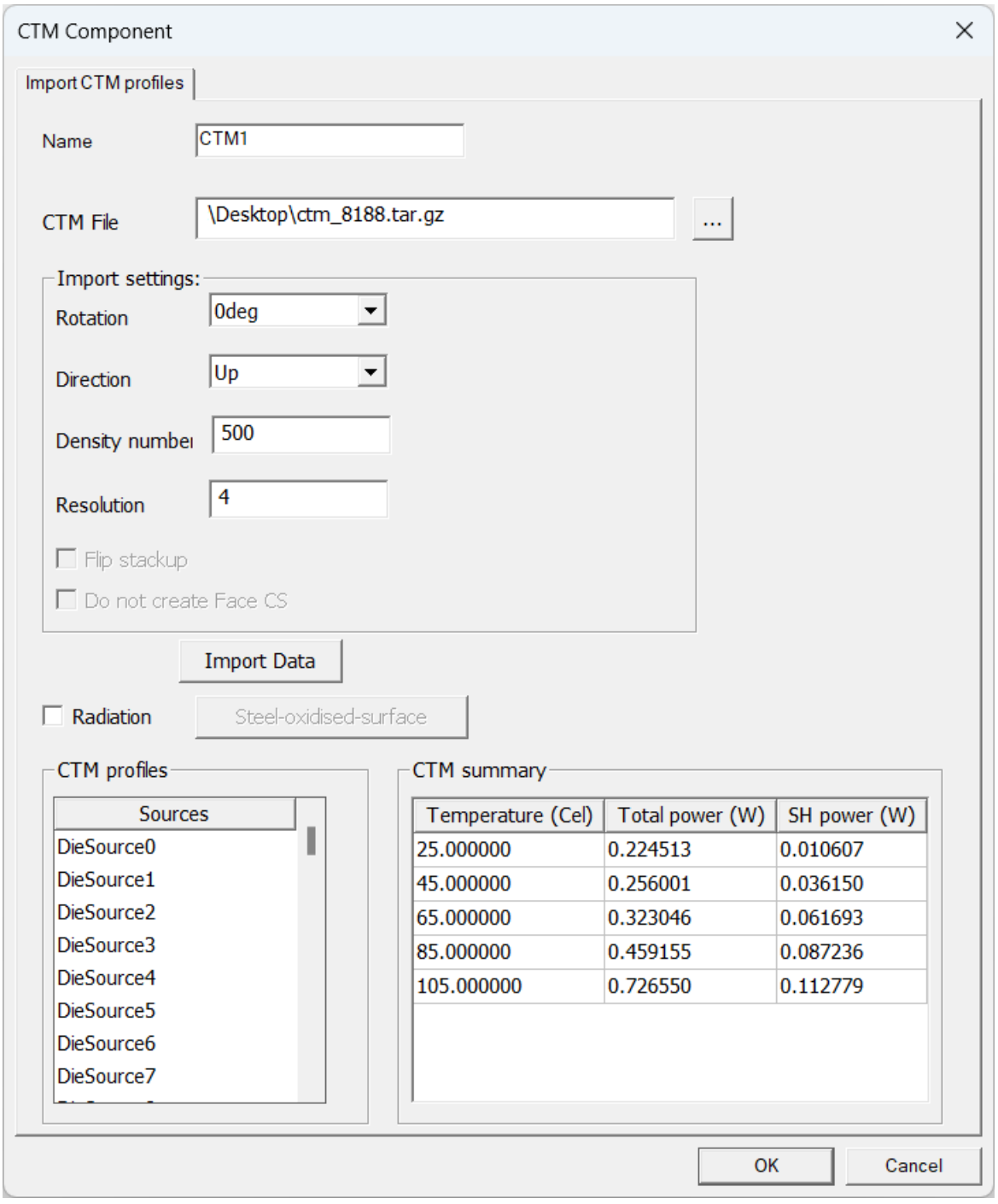


Note: When creating a CTM, the centroid of the selected face is the point of origin for the cell's location as seen in the image above.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating CTMs

Note: To create multiple CTMs, you must create multiple CTM native components. You can not insert an instance of a CTM. See Inserting Component Instances in Icepak for how to insert instances for supported Icepak 3D native components.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **CTM Component** dialog box **Import CTM profiles** tab contains the following information for the selected object.

Name	The **Name** of the object appears in the **Project Manager** window and **3D Modeler** window.
CTM File	**CTM File** allows you enter the file path to the CTM file or select the CTM file using the **Select CTM Import File** dialog box.
Import settings	
Rotation	**Rotation** uses the left-handed rotation rule. The direction of the rotational axis is along the outward direction/positive normal of the die face.
Direction	**Direction** specifies the direction the powermap faces. Select **Up** to import in the direction specified in the powermap file. Select **Down(X)** to flip the powermap face down along the X axis. Select **Down(Y)** to flip the powermap face down along the Y axis.
Density number	**Density number** specifies the approximate number of heat sources to create based on the number and resolution.
Resolution	**Resolution** specifies a factor by which to multiply the size of the smallest sources in the powermap. This is helpful when using a high **Density number**. The default entry of 1 retains the default size of the smallest sources.
Do not create Face CS	**Do not create Face CS** allows you to place the CTM relative to the active coordinate system in the 3D Modeler window. When **Do not create Face CS** is disabled, a coordinate system aligned with the selected face is created, and the CTM is placed relative to it.
Import Data	Click **Import Data** to populate the **CTM profiles** list and **CTM summary** table.
Radiation	Enable **Radiation** to include the effects of radiation on the surface material of the sources. Click the button to open the **Select Definition** dialog and select a material. The other objects in the 3D-IC package are assigned default surface materials.
CTM profiles	The **CTM profiles** list displays the sources in the powermap. Double-click a source to view its temperature-dependent surface heat curve in the **Edit Dataset** dialog box.
CTM summary	The **CTM summary** table displays the **Total power (W)** and **SH Power (W)** (self-heat power) for each **Temperature (Cel)** point.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a CTM

> **Note:** Icepak automatically generates a .bin file in the project directory. The binary file contains temperature data for import into Ansys RedHawk. You can customize the output directory on the **Export Settings** tab in the Design Settings.

1. Press F to enable face-selection mode.
2. Select a two-dimensional geometry component in the **3D Modeler** window or the history tree.
3. In the **Project Manager**, right-click **3D Components** and select **Create** > **CTM**.
4. On the **CTM Component** dialog box **Import CTM profiles** tab, enter following information:
   - **Name**
   - For **CTM File**, enter the file path or click the browse button to select the CTM file using the **Select CTM Import File** dialog box.

> **Note:** Supported file types include .tar.gz and .txt. If you import a .txt file, temperature data will not be available to be exported.

5. Under **Import settings**, do the following:
   - For **Rotation**, select the clockwise rotation of the powermap.
   - For **Direction**, select the direction the powermap faces.
   - For **Density number**, enter the number of heat sources in the powermap.
   - For **Resolution**, enter the factor by which to multiply the size of the smallest sources in the powermap.
6. Click **Import Data**.
7. If needed, select **Radiation**. Click the button to open the **Select Definition** dialog box and select a material.
8. In the **CTM profiles** list, review the powermap sources.

> **Note:**
>
> Double-click a source to view its temperature-dependent surface heat curve in the **Edit Dataset** dialog box.

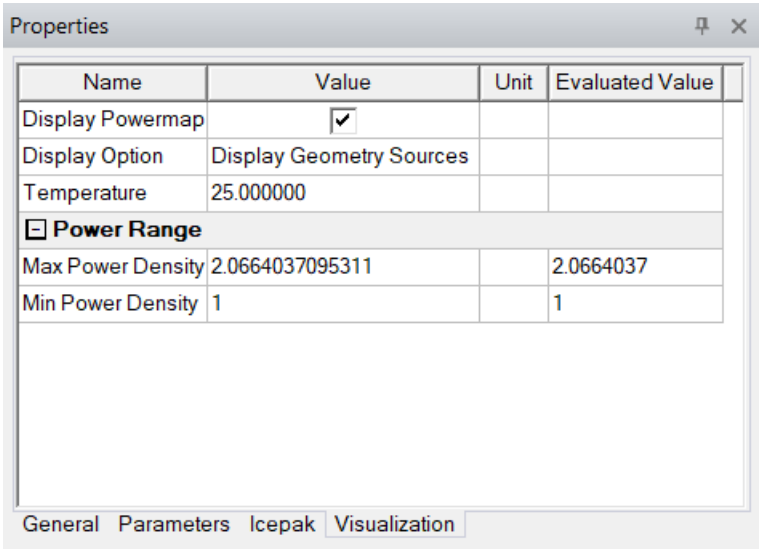
9. In the **CTM summary** table, review the **Total power (W)** and **SH Power (W)** (self-heat power) for each **Temperature (Cel)** point.
10. Click **OK** to create the CTM component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If the CTM does not contain any polygonal sources, the simulation runs faster and shortens in duration.

CTM Visualization

After you create a chip thermal model (CTM), you can display the imported power data using options on the **Visualization** tab of the **Properties** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

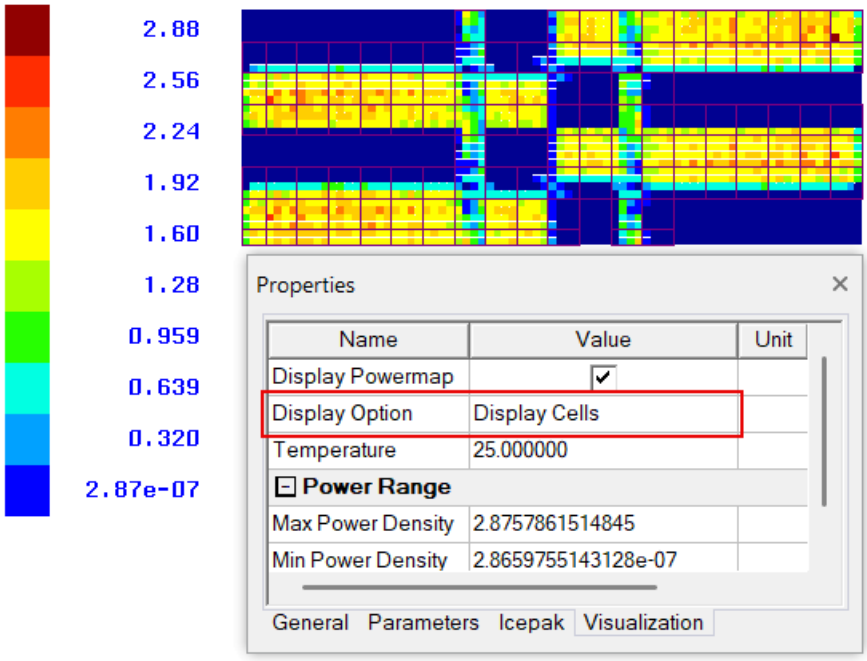
Display Powermap	Enable **Display Powermap** to visualize the power data mapped onto the chip in the **3D Modeler** window.
Display Option	From the drop-down list, select **Display Cells** or **Display Geometry Sources**. **Display Cells** shows the power map data at the resolution as defined in the CTM import file. **Display Geometry** shows the data mapped onto the sheet geometry created while importing the CTM. See "Display Options" below for examples of CTM visualization.
Temperature	From the drop-down list, select a **Temperature**. The powermap is updated to display power values at the selected temperature.
Power Range	
Max Power Density	Enter a maximum power value for the range displayed in the powermap. The powermap is updated to display the maximum power entered.
Min Power Density	Enter a minimum power value for the range displayed in the powermap. The powermap is updated to display the minimum power entered.

Display Options

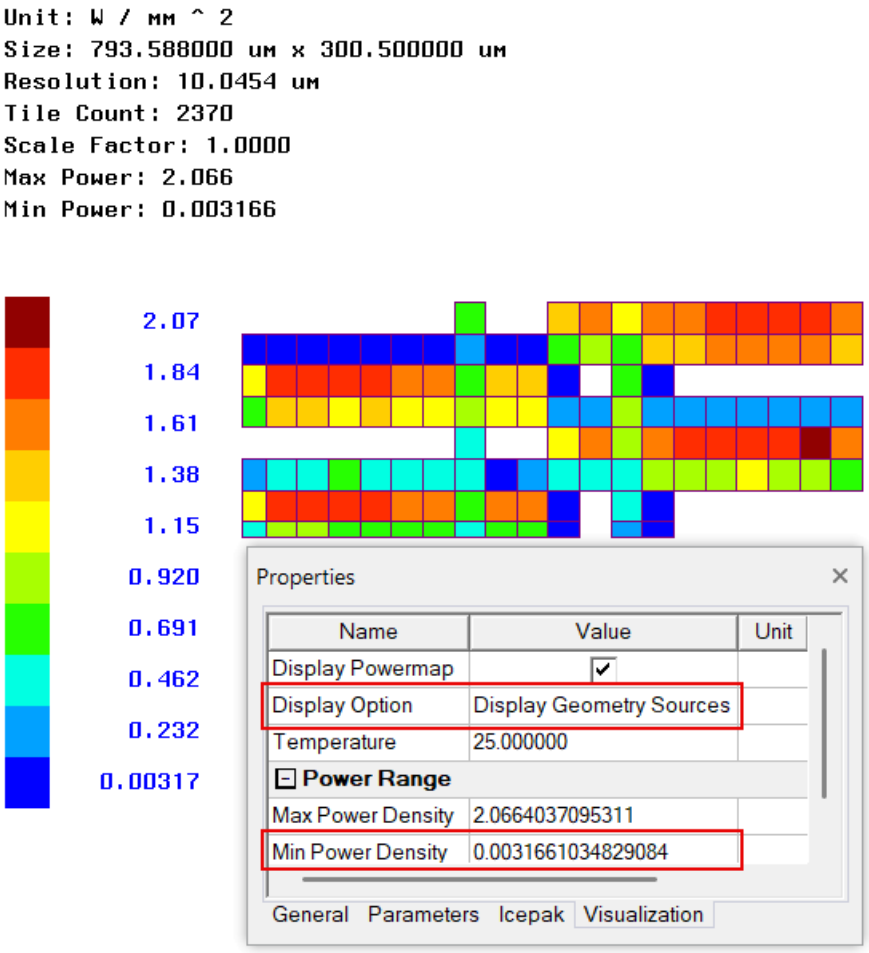
The following images display a CTM visualized using **Display Cells** and **Display Geometry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Unit: W / mm ^ 2
Size: 793.588000 um x 300.500000 um
Resolution: 10.0454 um
Tile Count: 2370
Scale Factor: 1.0000
Max Power: 2.876
Min Power: 2.866e-07
```



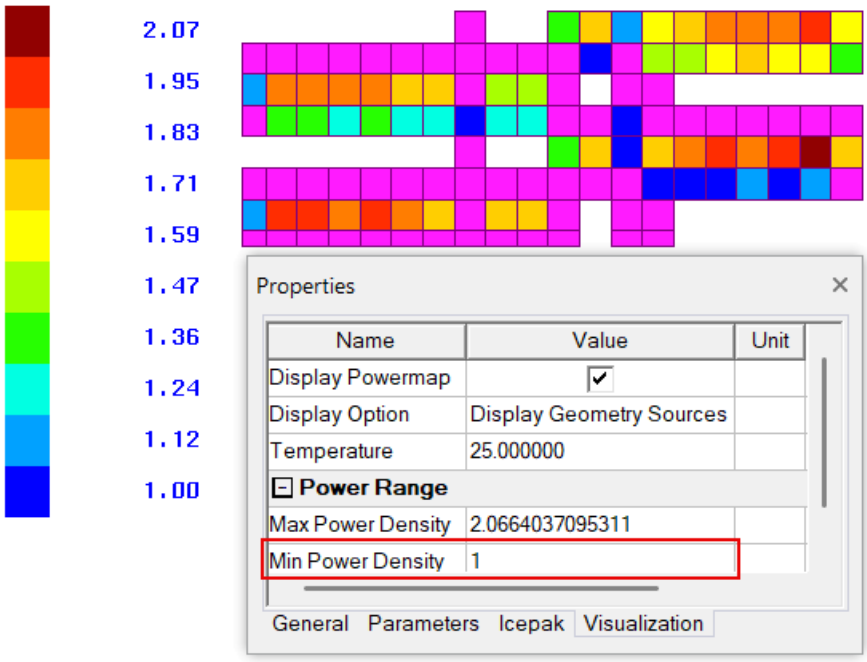
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the image above, the **Min Power Density** is set to the default of 0.00317. In the image below, the **Min Power Density** is set to 1.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Unit: W / mm ^ 2
Size: 793.588000 um x 300.500000 um
Resolution: 10.0454 um
Tile Count: 2370
Scale Factor: 1.0000
Max Power: 2.066
Min Power: 0.003166
```



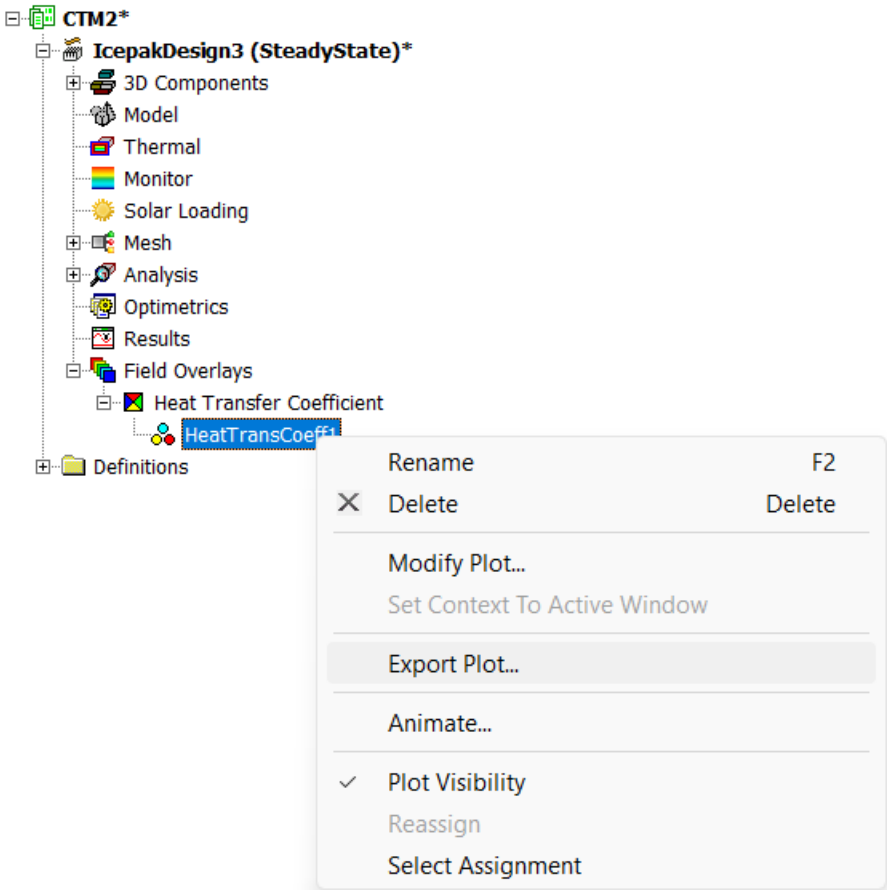
Export a CTM Field Plot

After solving a design with a CTM, you can export heat transfer coefficient field plot data as a .tab file. The exported file can be imported into Ansys Redhawk-SC to run electrothermal simulations.

After you create a heat transfer coefficient field plot, do the following to export the data.

1. In the **Project Manager** under **Field Overlays**, right-click on the field plot and select **Export**.

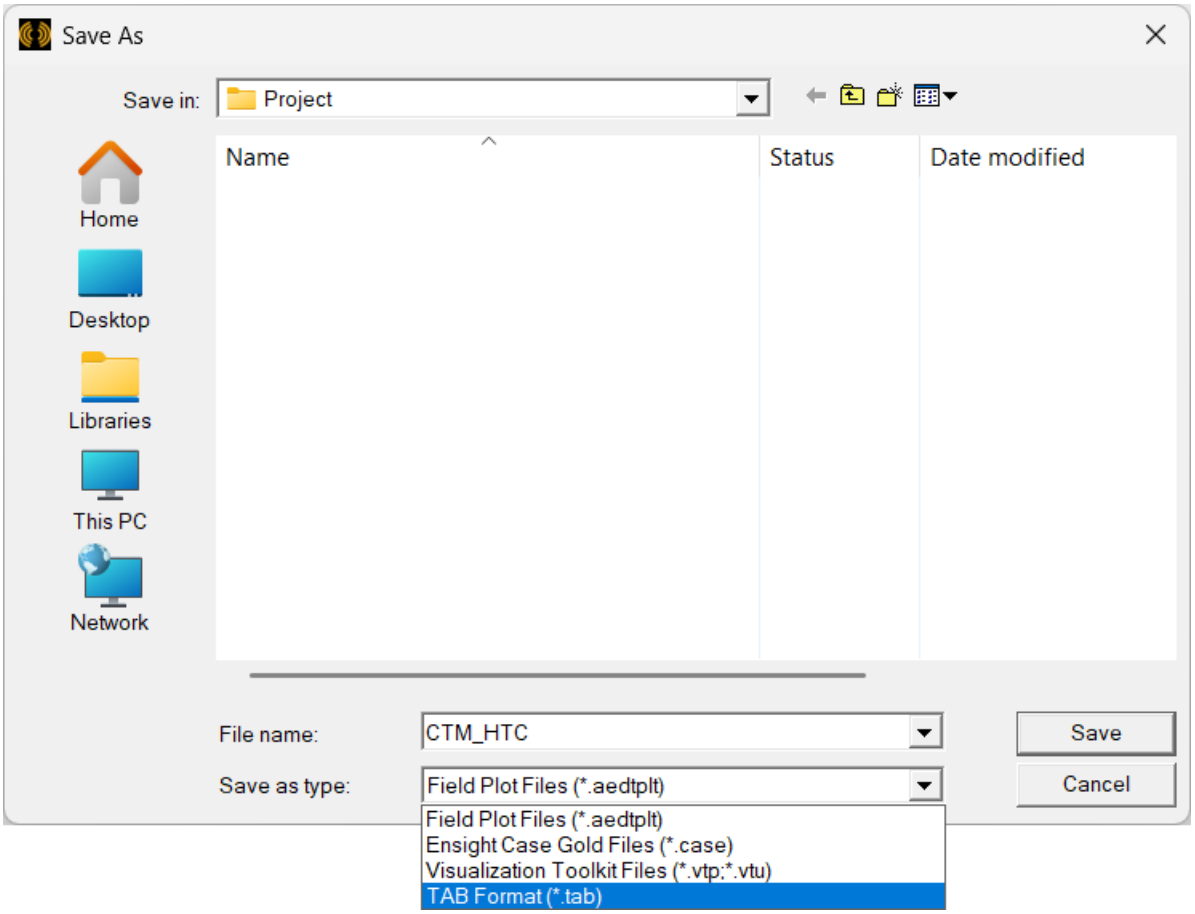
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Save As** dialog box, browse to your working directory.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Save as Type** drop-down list, select **TAB Format (*.tab)**.

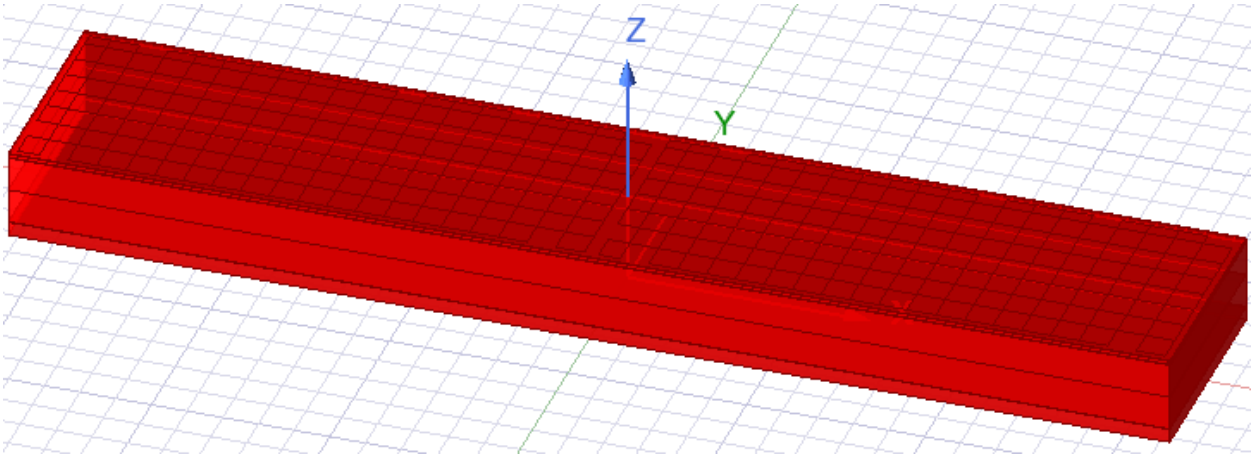


4. Enter a **File name**.
5. Click **Save**.

3D Integrated Circuits (3D-ICs)

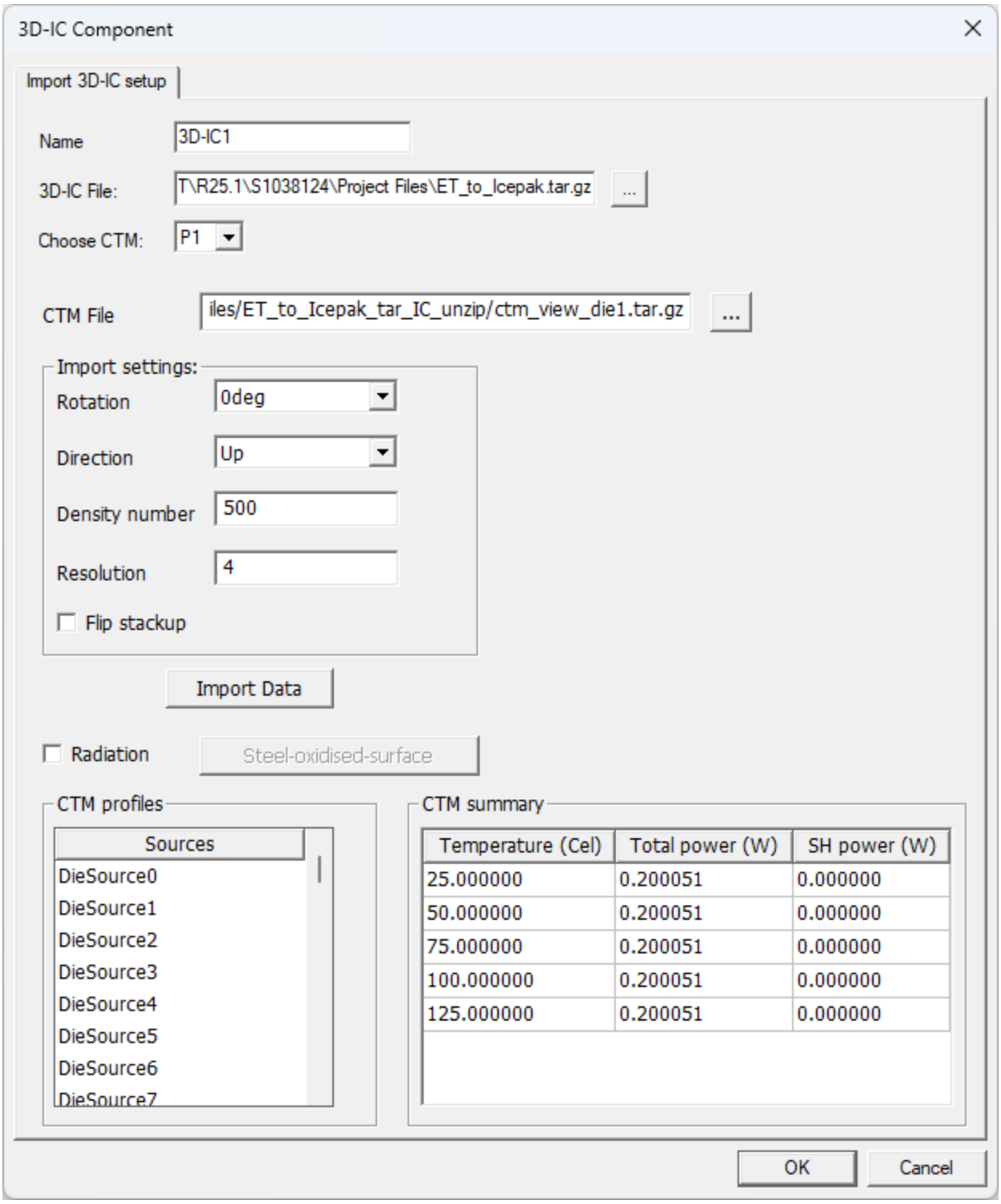
3D-ICs are multi-die thermal models generated for multiphysics simulation. The Ansys Icepak application can process CTM (v1 or v2) for individual dies. The 3D-IC workflow allows Icepak to bring in multi-die assemblies using an .xml file import. The .xml file includes the block structure, the position of the CTM (v1 or v2) powermap, and the material properties of all the layers in the package. All the necessary files for this system-level simulation are packaged in a *.tar.gz file generated by RedHawk-SC Electrothermal. Icepak processes this file and creates necessary geometries inside an Icepak design as a 3D-IC native component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 3D-ICs



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **3D-IC Component** dialog box **Import 3D-IC setup** tab contains the following information for the selected object.

Name	The **Name** of the object appears in the **Project Manager** window and **3D Modeler** window.
3D-IC File	**3D-IC File** allows you enter the file path to the 3D-IC file or select the 3D-IC file using the **Select 3D-IC Import File** dialog box. The CTM files are packaged in the 3D-IC file and are populated as options in the Choose CTM drop-down list.
Choose CTM	**Choose CTM** allows you to view the 3D-IC's CTM properties for the selected CTM.
CTM File	The **CTM File** location is automatically set based on the **Choose CTM** selection. You can enter a different path to the CTM file or select the CTM file using the **Select CTM Import File** dialog box.
Import settings	
Rotation	**Rotation** uses the left-handed rotation rule. The direction of the rotational axis is along the outward direction/positive normal of the die face.
Direction	**Direction** specifies the direction the powermap faces. Select **Up** to import in the direction specified in the powermap file. Select **Down(X)** to flip the powermap face down along the X axis. Select **Down(Y)** to flip the powermap face down along the Y axis.
Density number	**Density number** specifies the approximate number of heat sources to create based on the number and resolution.
Resolution	**Resolution** specifies a factor by which to multiply the size of the smallest sources in the powermap. This is helpful when using a high **Density number**. The default entry of 1 retains the default size of the smallest sources.
Import Data	Click **Import Data** to populate the **CTM profiles** list and **CTM summary** table.
Radiation	Enable **Radiation** to include the effects of radiation on the surface material of the sources. Click the button to open the **Select Definition** dialog and select a material. The other objects in the 3D-IC package are assigned default surface materials.
CTM profiles	The **CTM profiles** list displays the sources in the powermap. Double-click a source to view its temperature-dependent surface heat curve in the **Edit Dataset** dialog box.
CTM summary	The **CTM summary** table displays the **Total power (W)** and **SH Power (W)** (self-heat power) for each **Temperature (Cel)** point.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a 3D-IC

> **Note:** Icepak automatically generates a .bin file in the project directory. The binary file contains temperature data for import into Ansys RedHawk. You can customize the output directory on the **Export Settings** tab in the Design Settings.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **3D-IC**.
2. On the **3D-IC Component** dialog box **Select 3D-IC Import File** tab, enter following information:
   - **Name**
   - For **3D-IC File**, enter the file path or click the browse button to select the 3D-IC file using the **Select 3D-IC Import File** dialog box.

     > **Note:** Selecting a 3D-IC package automatically populates the CTM file options. You can optionally define a different CTM file location.

   - From the **Choose CTM** drop-down list, select a CTM to import into the 3D-IC component.
   - For **CTM File**, enter the file path or click the browse button to select the CTM file using the **Select CTM Import File** dialog box.
3. Under **Import settings**, do the following:
   - For **Rotation**, select the clockwise rotation of the powermap.
   - For **Direction**, select the direction the powermap faces.
   - For **Density number**, enter the number of heat sources in the powermap.
   - For **Resolution**, enter the factor by which to multiply the size of the smallest sources in the powermap.
4. Click **Import Data**.
5. If needed, select **Radiation**. Click the button to open the **Select Definition** dialog box and select a material.
6. In the **CTM profiles** list, review the powermap sources.

   > **Note:**
   >
   > Double-click a source to view its temperature-dependent surface heat curve in the **Edit Dataset** dialog box.

7. In the **CTM summary** table, review the **Total power (W)** and **SH Power (W)** (self-heat

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

power) for each **Temperature (Cel)** point.

8. Click **OK** to create the 3D-IC component.

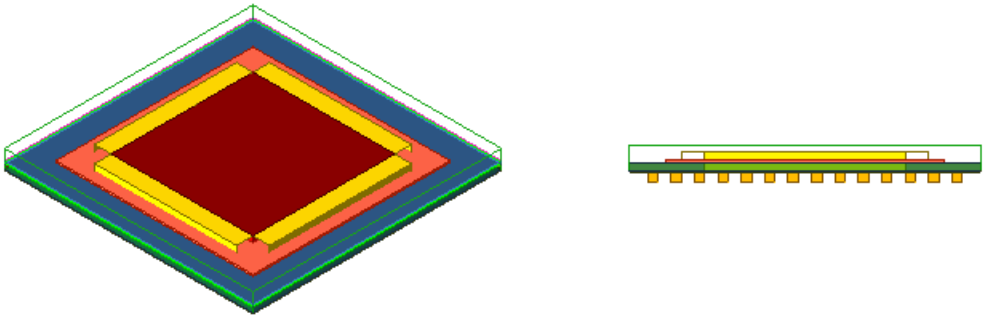
Packages (Beta)

> **Note:** The package native component is currently a beta feature. For information on enabling beta options, see General Options: Desktop Configuration.

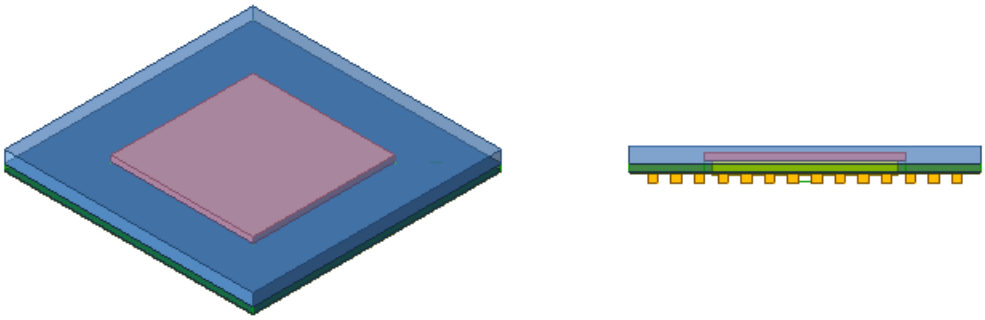
Packages are composite modeling objects representing microelectronic packages. Given a minimal amount of input, you can create a package in Ansys Icepak to model the thermal characteristics of different devices at the system, subsystem, board, or package level.

There are seven types of packages available:

- PBGA

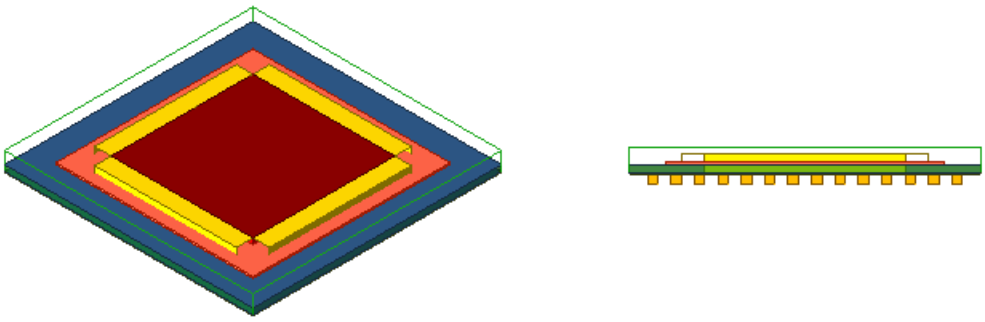


- Cavity Down BGA

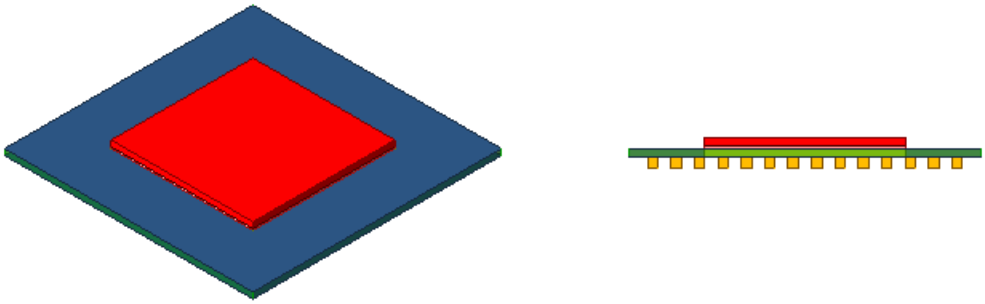


- FPBGA

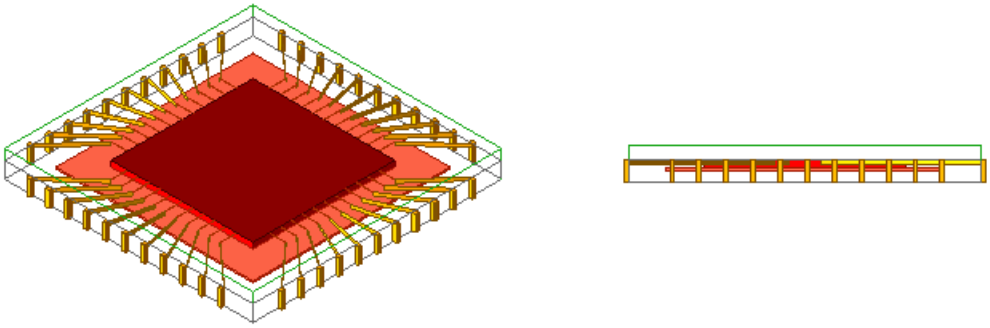
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



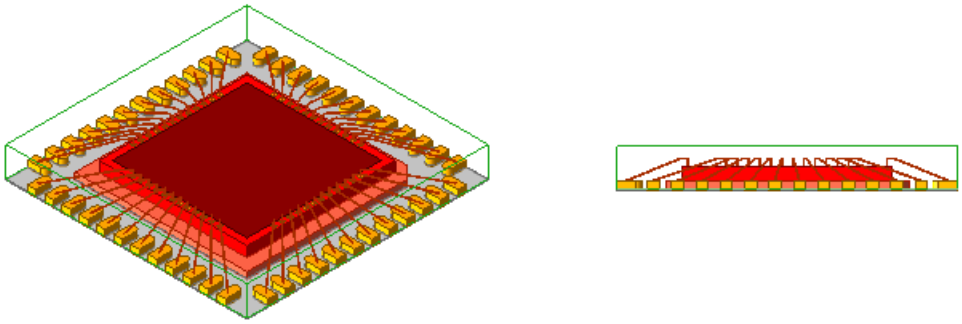
- Flip-chip BGA



- QFP

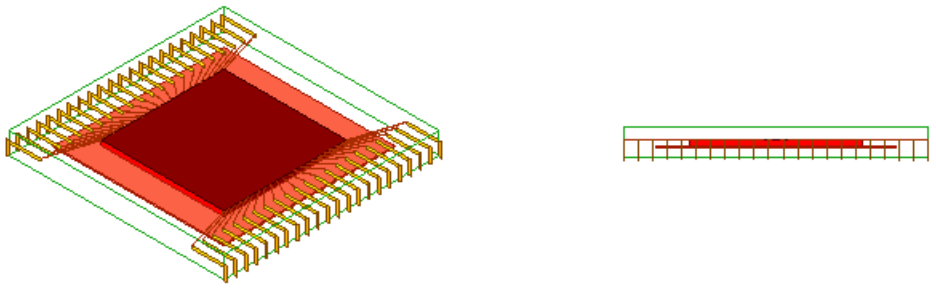


- QFN



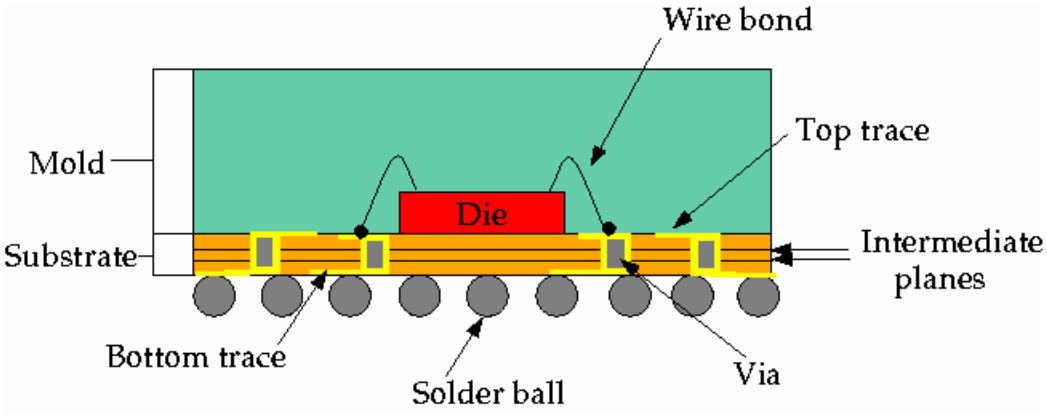
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Dual



Detailed Packages

To model a device at the package level, it can be as simple as a package on a board with wall-heattransfer boundary conditions applied to every surface that is exposed to air. For such a simple model, you can model almost all of the details of the package. As an example of a detailed package, a cross-section of a typical ball grid array (BGA) package is shown in the following figure:



Of the components shown in the figure above, some can be modeled in detail, and others are approximated.

- Detailed Features
- Approximated Features

Detailed Features

The *die* and *mold* are volumetric solids made of user-specified materials. The die also has a *power source* distributed uniformly on its surface and can be of specified dimensions. For lead-frame packages (for example, QFPs), each *lead* is modeled as an angled conducting thin plate

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of user-specified material and dimensions. For BGA packages (such as PBGAs), each *solder ball* is modeled as a discrete cuboid for which the cross-sectional area is that of a cylinder with the specified solder diameter.

Wire bonds connect signals between the die and the leads (in lead-frame packages) or substrate traces (in BGA packages) and are made of gold. In a detailed lead-frame package model, each bond wire is modeled as a conducting thin plate. In a detailed BGA model, the bond wires are lumped into a planar thin conducting entity connecting the die with the trace edge. The properties of the lumped entity are volume-averaged, with the conductivity being isotropic.

The *adhesive* and the *die attach* are modeled as contact resistances of user-specified material and thickness. The *die paddle* or *flag* is modeled as a volumetric solid, also with a user-specified material and thickness.

Approximated Features

Substrates are usually encountered in BGA packages. The *substrate* is the medium that houses the *traces* that carry the signals from the bond wire to the solder balls. A signal from a trace in one layer is connected to trace in another layer by means of *vias*. Vias are cylindrical holes that are either plated or filled with copper. Some BGA packages also have thermal vias that enhance the heat transfer between the die and the thermal balls.

The substrate is represented as solid block. The effect of the copper content in the vias is incorporated in the form of averaged properties. The density and specific heat are averaged based on volume using the following equation:

$$k_{eff} = \sum_i Z_i k_i$$

$$\rho_{eff} = \sum_i Z_i \rho_i$$

$$C_{p,eff} = \sum_i Z_i C_{p,i}$$

where Z_i is the volume ratio for component *i*.

The substrate conductivity is modeled as orthotropic. Conductivity in the direction normal to the substrate plane is determined using thermal resistances in parallel in the following equation:

$$k_{parallel} = \sum_i R_{area,i} k_i$$

where k_i is the conductivity of component *i* (that is, the conductivities of the lead and the mold, respectively). $R_{area,i}$ is the cross-sectional area ratio for component *i*. For example, the cross-sectional area ratio for the lead would be

$$\mathscr{R}_{area,lead} = \frac{A_{lead,cross}}{A_{lead,cross} + A_{mold,cross}}$$

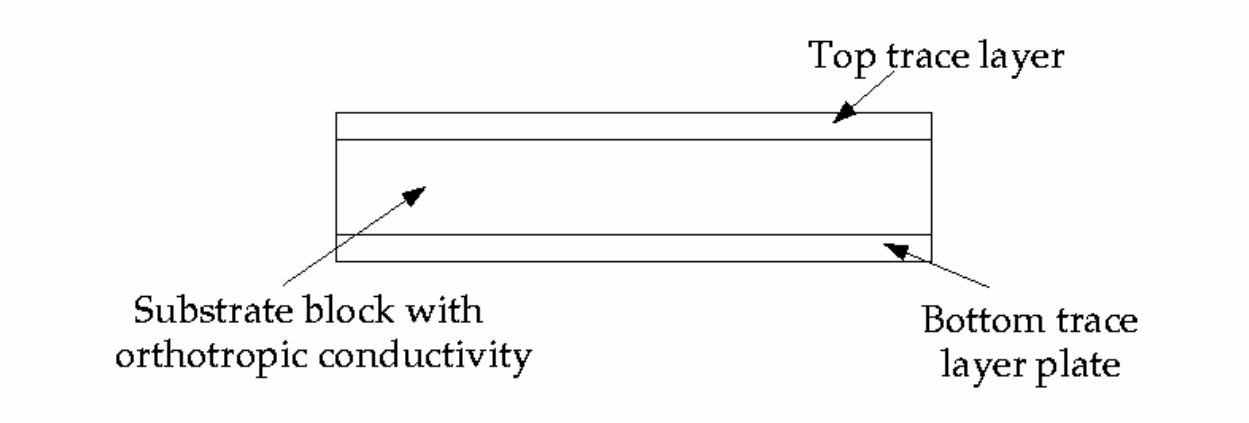
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal resistances of the substrate and vias in the in-plane direction are in series. The following equation is used for the conductivity in the plane directions:

$$k_{series} = \frac{1}{\sum_i \frac{\mathscr{R}_{length,i}}{k_i}}$$

where $\mathscr{R}_{length,i}$ is the ratio of the length of component *i* to the total length of the composite in the heat flow direction. For example, $\mathscr{R}_{length}$ for the lead would be equal to the lead width divided by the lead pitch.

At the detailed level, each layer of trace is represented as a thin conducting entity (similar to a conducting thin plate) with volume-averaged properties (see "where Zi is the volume ratio for component i." on the previous page). A typical substrate may be constructed as shown in the following figure using standard Icepak objects.



Compact Conduction Model (CCM) Packages

To model a package at the board, subsystem, or system level, you can use the compact conduction model (CCM). In the CCM, all repetitive features, such as the leads or the solder balls, are approximated using effective orthotropic conductivities. Certain features are not modeled in the CCM so that the model is simple enough to use at system levels.

- Lead-Frame Packages
- Ball Grid Array Packages

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lead Frame Packages

For a lead-frame-type package, the lead wires and wire bonds are lumped into thin conducting entities that connect the die side with the package side. The lead wires that connect the package side with the mounting PCB are also conducting thin entities. The conductivity, density, and specific heat of these entities are volumetrically averaged using the following equation:

$$k_{eff} = \sum_i Z_i k_i$$

$$\rho_{eff} = \sum_i Z_i \rho_i$$

$$C_{p,eff} = \sum_i Z_i C_{p,i}$$

Ball Grid Array (BGA) Packages

For a BGA-type package, solder balls are approximated as a contact resistance with the height of the balls and an effective conductivity (the following equation). The density and specific heat are volume averaged using this equation as well.

$$k_{eff} = \sum_i Z_i k_i$$

$$\rho_{eff} = \sum_i Z_i \rho_i$$

$$C_{p,eff} = \sum_i Z_i C_{p,i}$$

The substrate is modeled as a solid object with orthotropic properties. The in-plane conductivity is determined using the following equation:

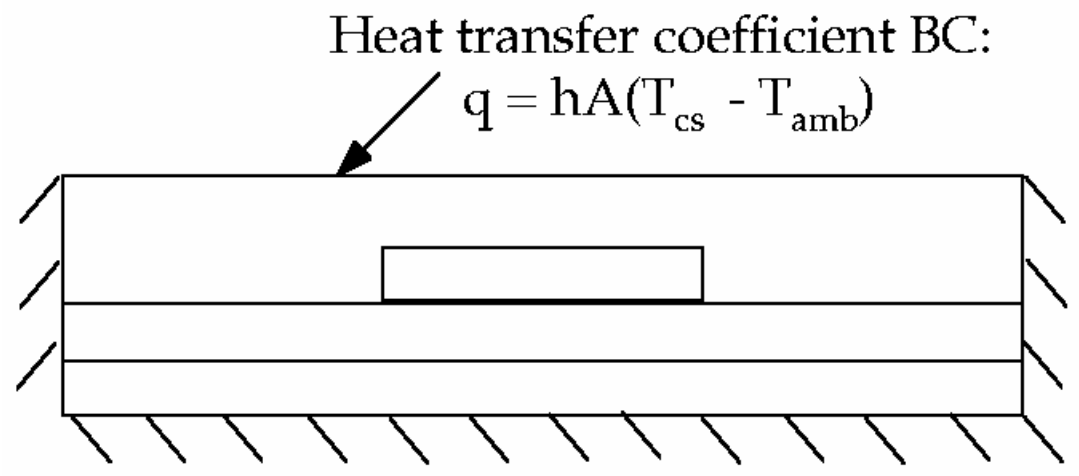
$$k_{parallel} = \sum_i R_{area,i} k_i$$

where the area ratio is the ratio of the total copper trace thickness to the thickness of the substrate.The conductivity normal to the plane is either the conductivity of the substrate material or the parallel conductivity using this equation in substrates with vias. The parallel conductivity in the normal direction is determined using the fraction of the cross-sectional area occupied by vias.

Junction-to-Case Characterization Model

The junction-to-case (JC) thermal resistance, R_{jc} , can be determined using a model of the package surrounded with adiabatic surfaces on all sides except the package top (see the following figure), where a constant heat-transfer-coefficient boundary condition is applied.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$dq = h\left(T_{cs} - T_{amb}\right) dA_s$$

where

dq = heat dissipated through each surface element *T*

cs = temperature of discrete case surface *T*

amb = constant ambient temperature

dA_s = area of the surface element

R_{jc} can then be determined by

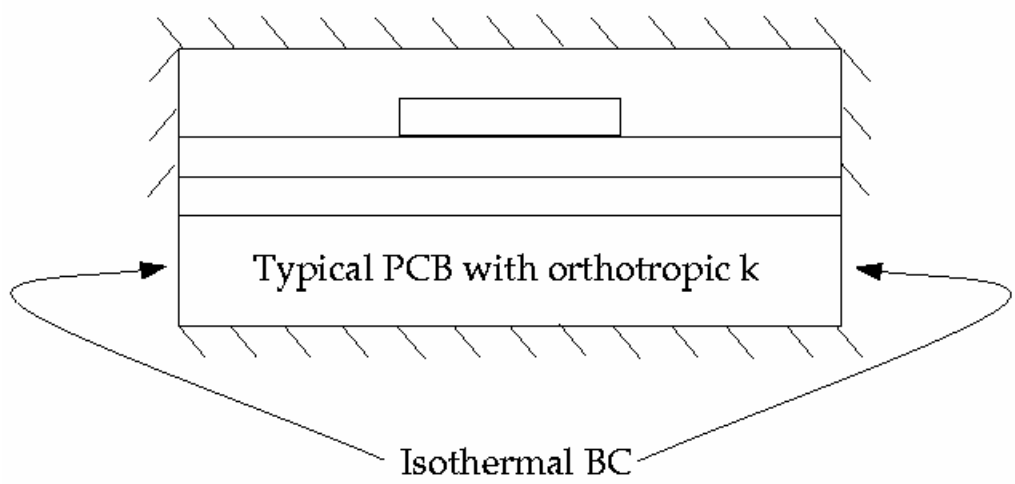
$$R_{jc} = \frac{T_{die} - T_{c,mean}}{q}$$

where T_{die} is the maximum die temperature. $T_{c,mean}$ is the average case temperature, which can be determined by generating a temperature report for the wall object covering the case. *q* is the full power dissipated through the case, which is equal to the power generated by the die since all other surfaces are adiabatic. See Post-processing and Generating Reports for details about generating reports.

Junction-to-Board Characterization Model

The junction-to-board (JB) thermal resistance, R_{jb} , can be determined using a model of the package mounted on a board that is 5 mm wider than the package on all sides. All surfaces are insulated except for the periphery of the board (see the following figure), which is subject to an isothermal boundary condition.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



R_{jb} can then be determined by

$$R_{jb} = \frac{T_{die} - T_{iso}}{q}$$

Note that if T_{iso} is set to 0° C and if q is 1 W, then R_{jb} will be the same as T_{die}.

Creating a Package

The Ansys Electronics Desktop uses a tabbed for the creation of a package component, including tabs to input geometry and thermal boundary information. To create components using a wizard interface, access the **Icepak**>**Thermal** options in the **Options** dialog (**Tools**>**Options**>**General Options**) and select **Use Wizards for data input when creating new boundaries**.

> **Note:** You can define component variables in Icepak native 3D components. See Defining Native Component Instance Parameters for more information.

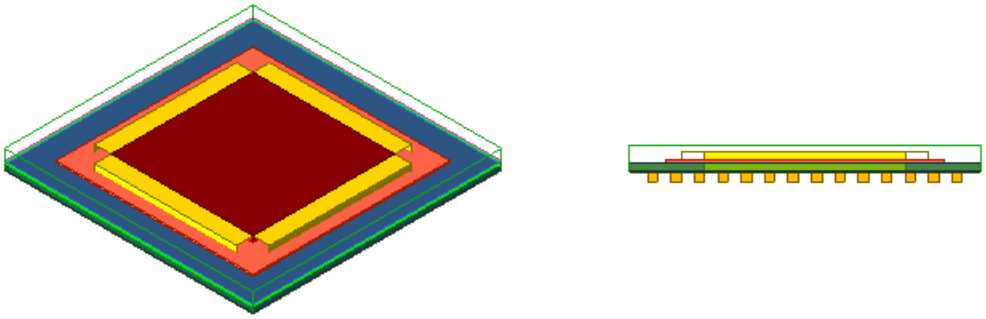
1. In the **Project Manager**, right-click **3D Components** and select **Create** >**Package**.
2. On the **Package** dialog box **Package** tab, enter a **Name** for the component.
3. From the drop-down list, select a **Package Type**.
4. For information on defining the parameters of each package type, click the following links:
   - Defining PBGA Parameters
   - Defining Cavity Down BGA Parameters
   - Defining FPBGA Parameters
   - Defining Flip-chip BGA Parameters
   - Defining QFP Parameters

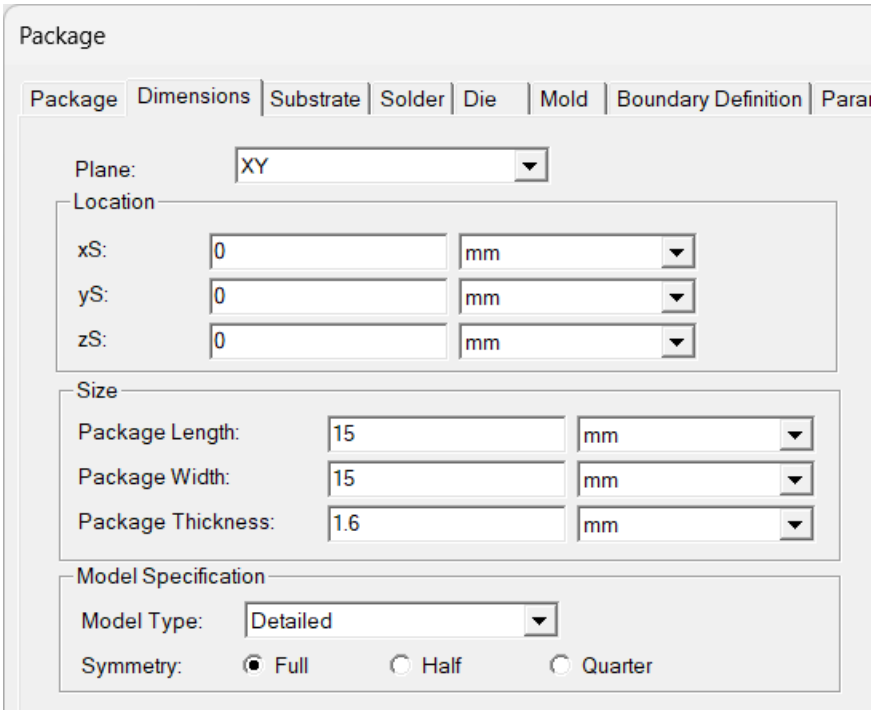
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining QFN Parameters
- Defining Dual Parameters

Defining PGBA Parameters

PBGA enables you to create geometry and boundary conditions to represent an plastic ball grid array package.

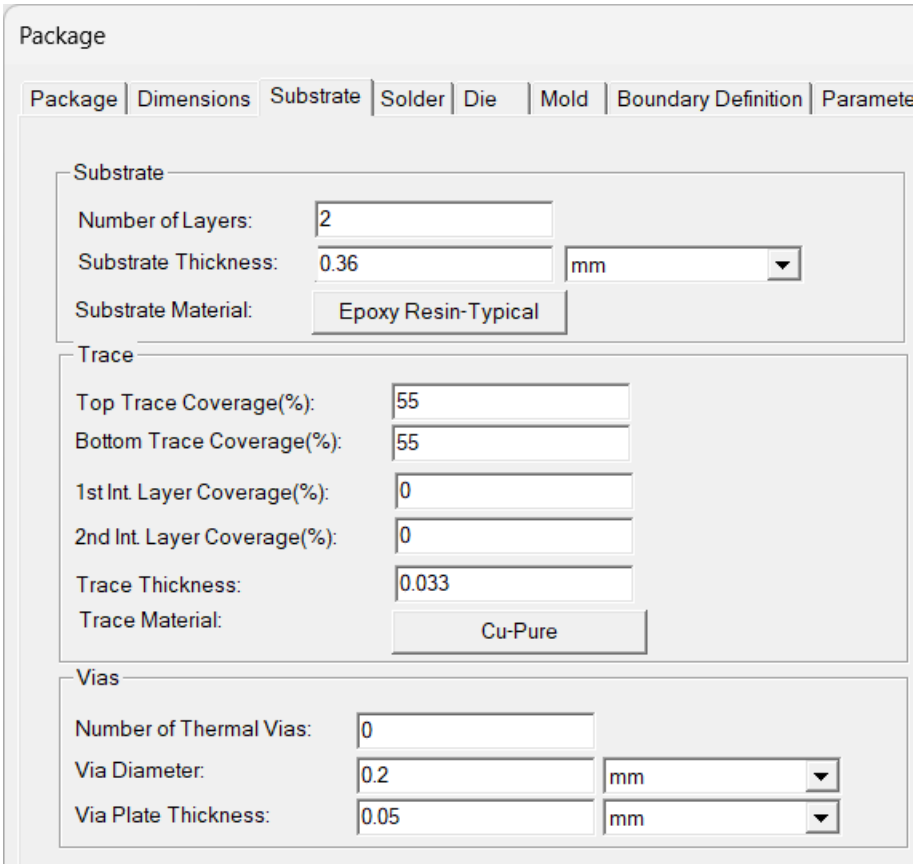




Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.

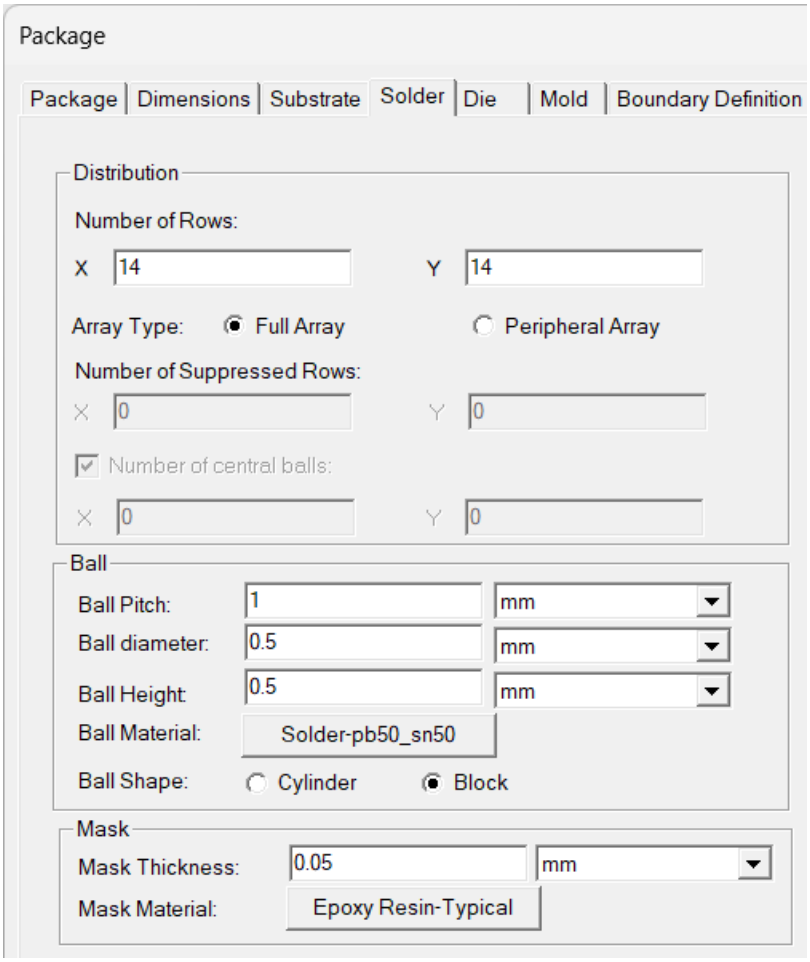
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specification	Under **Model Specification**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.



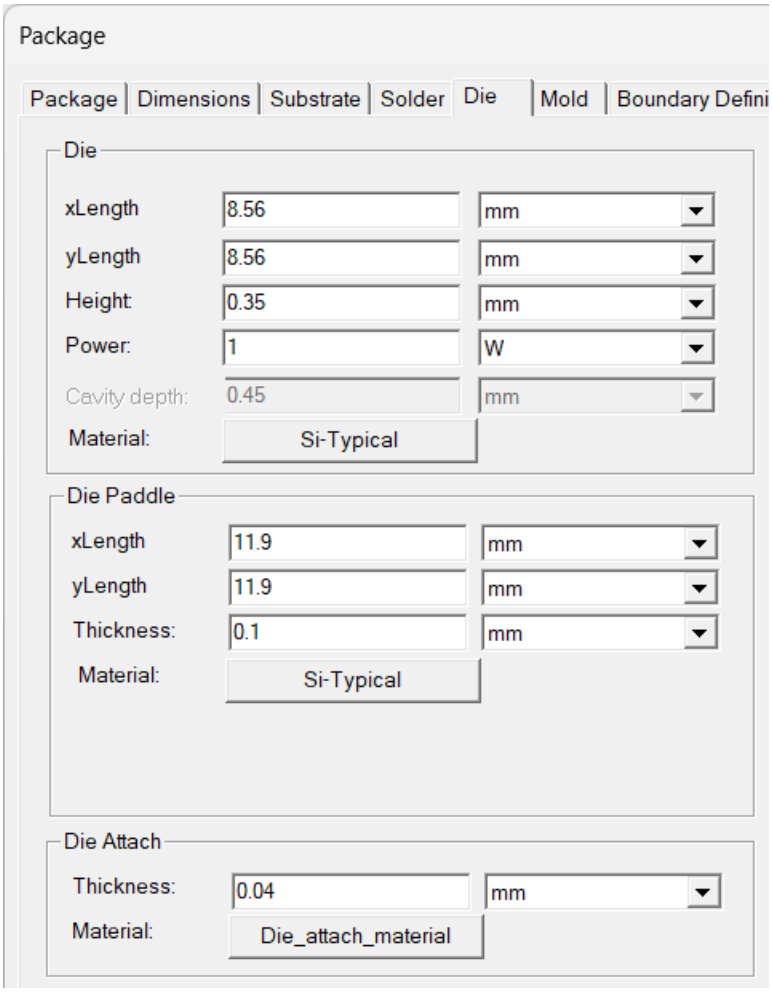
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



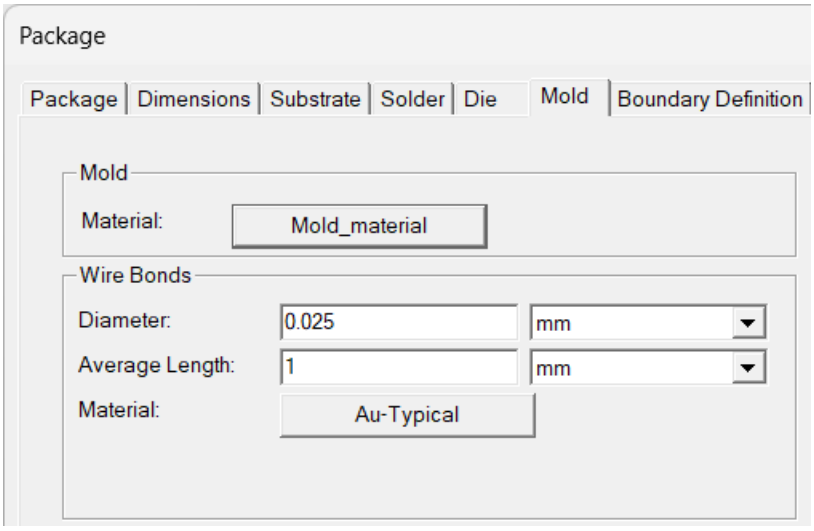
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed rows and number of central balls.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls and their shape.
Mask	Under **Mask**, enter the solder ball mask thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



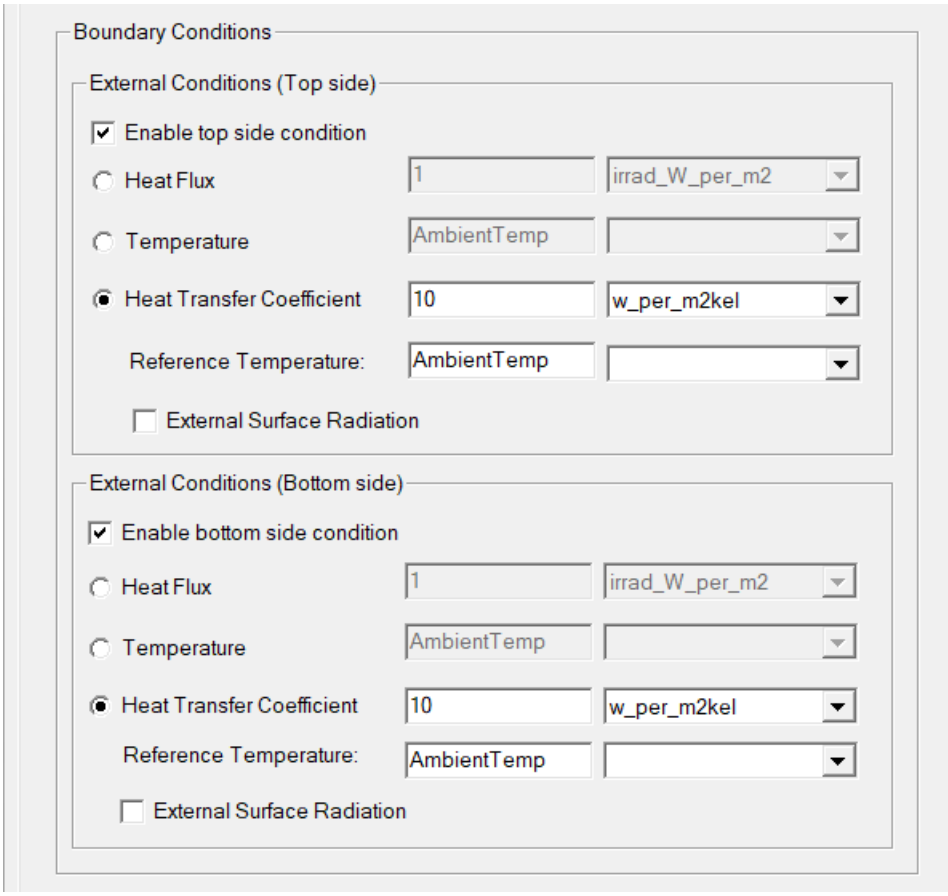
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

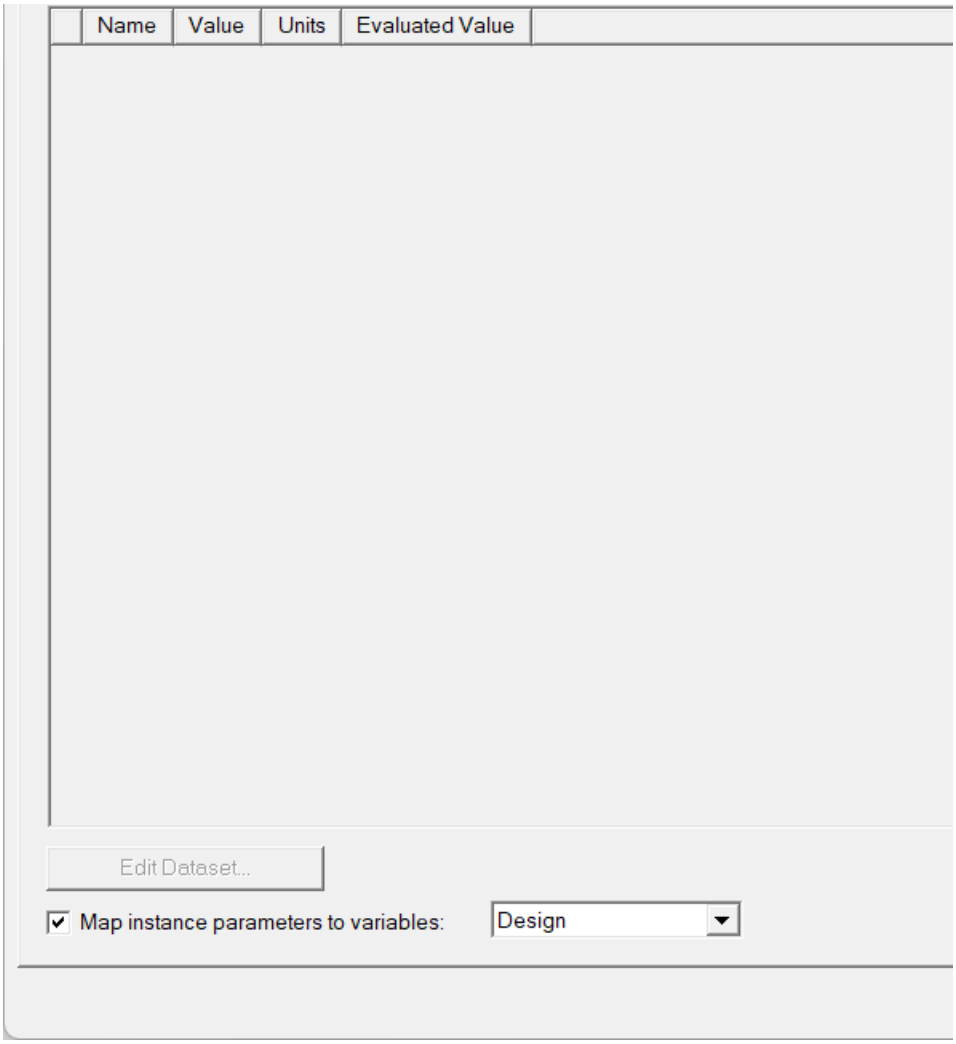


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

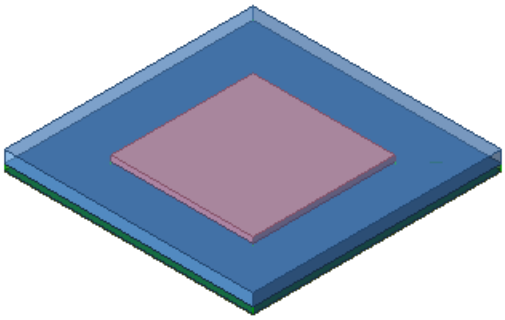


Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

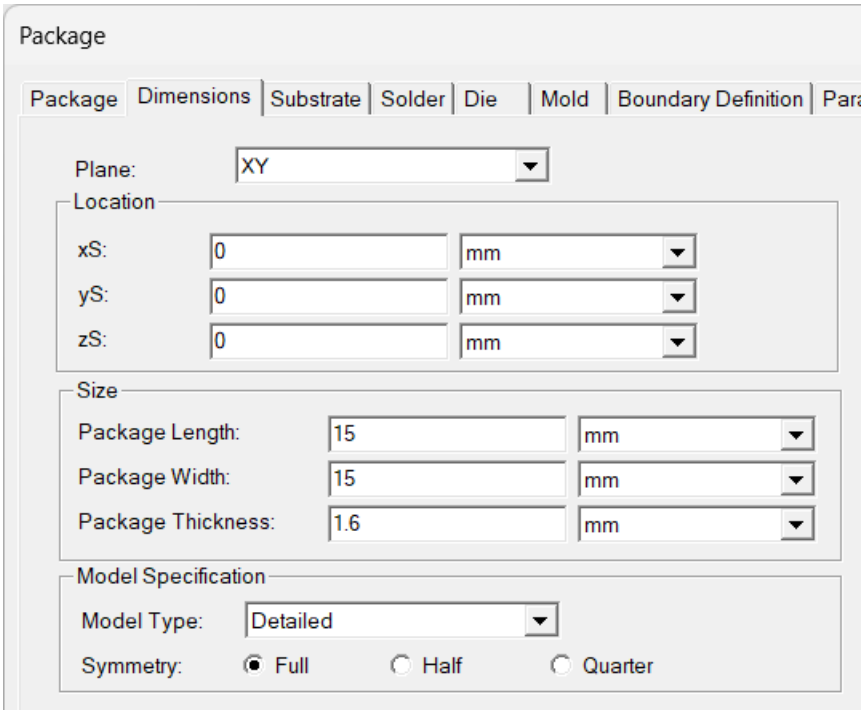
Defining Cavity Down BGA Parameters

Cavity Down BGA enables you to create geometry and boundary conditions to represent a cavity-down ball grill array package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

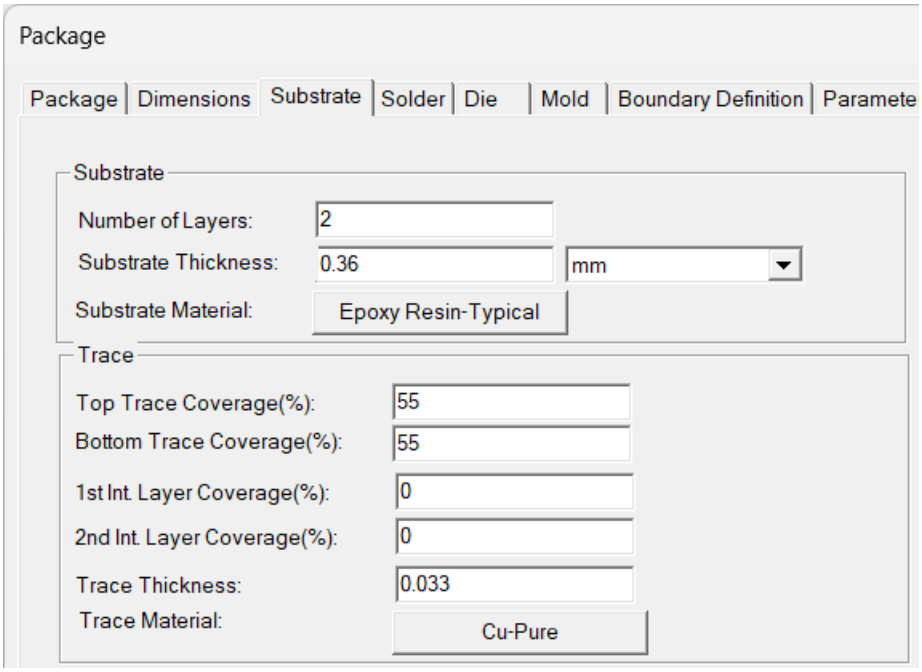






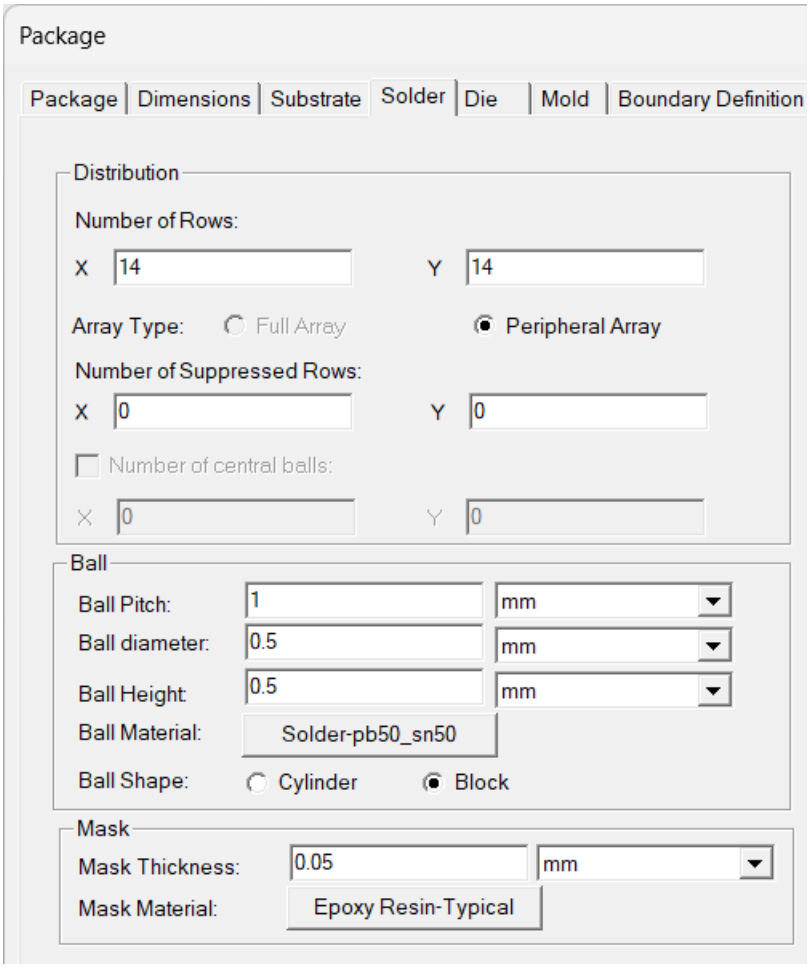
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



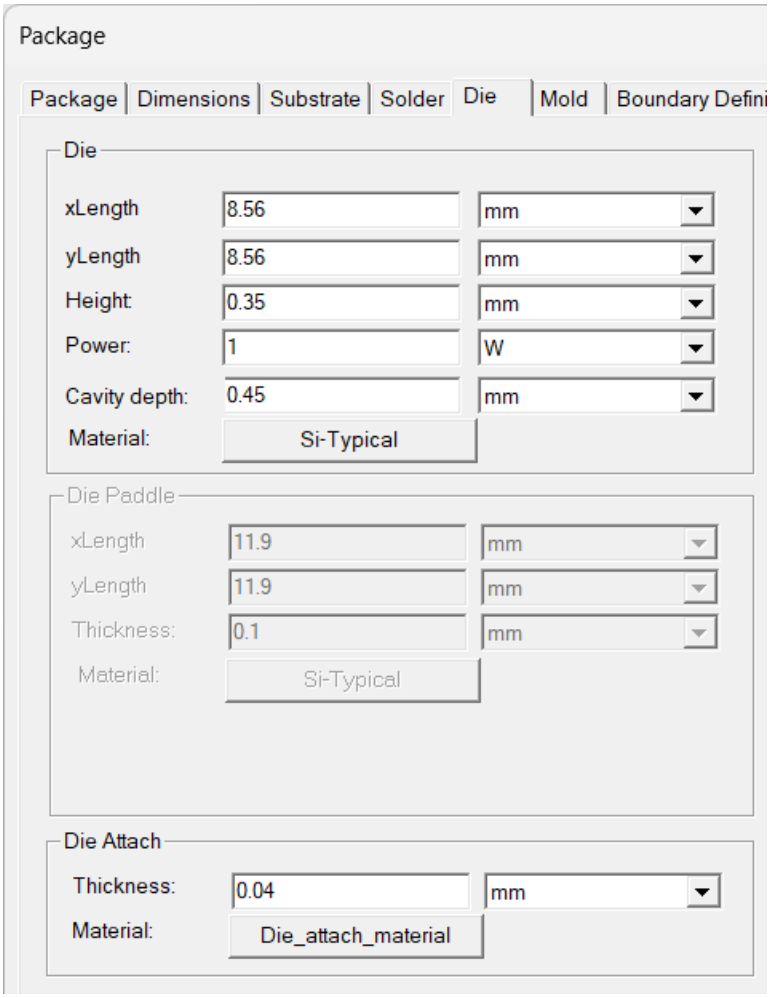
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



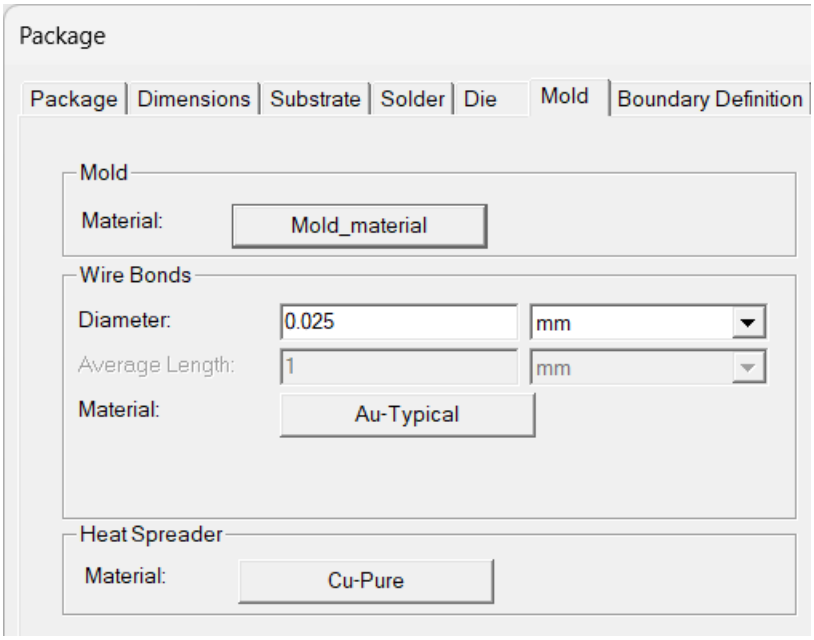
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed rows and number of central balls.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls and their shape.
Mask	Under **Mask**, enter the solder ball mask thickness and mask material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



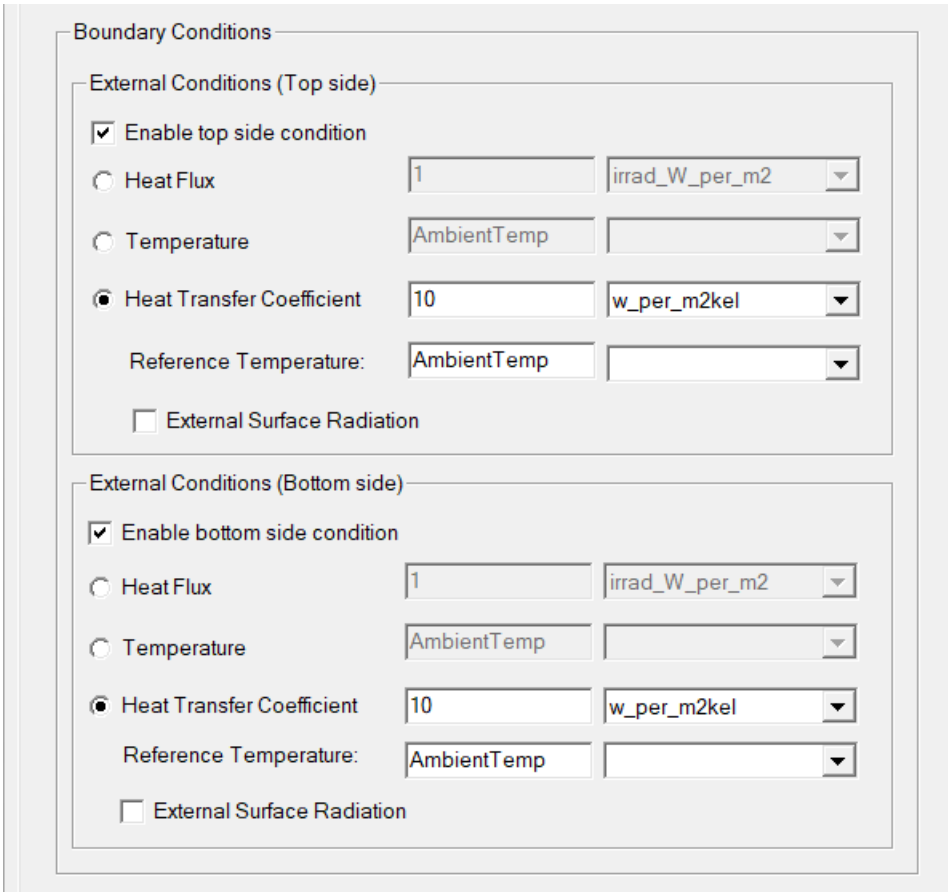
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material and the depth of the die cavity.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter and material of the wire bonds.
Heat Spreader	Under **Heat Spreader**, select the material for the heat spreader.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

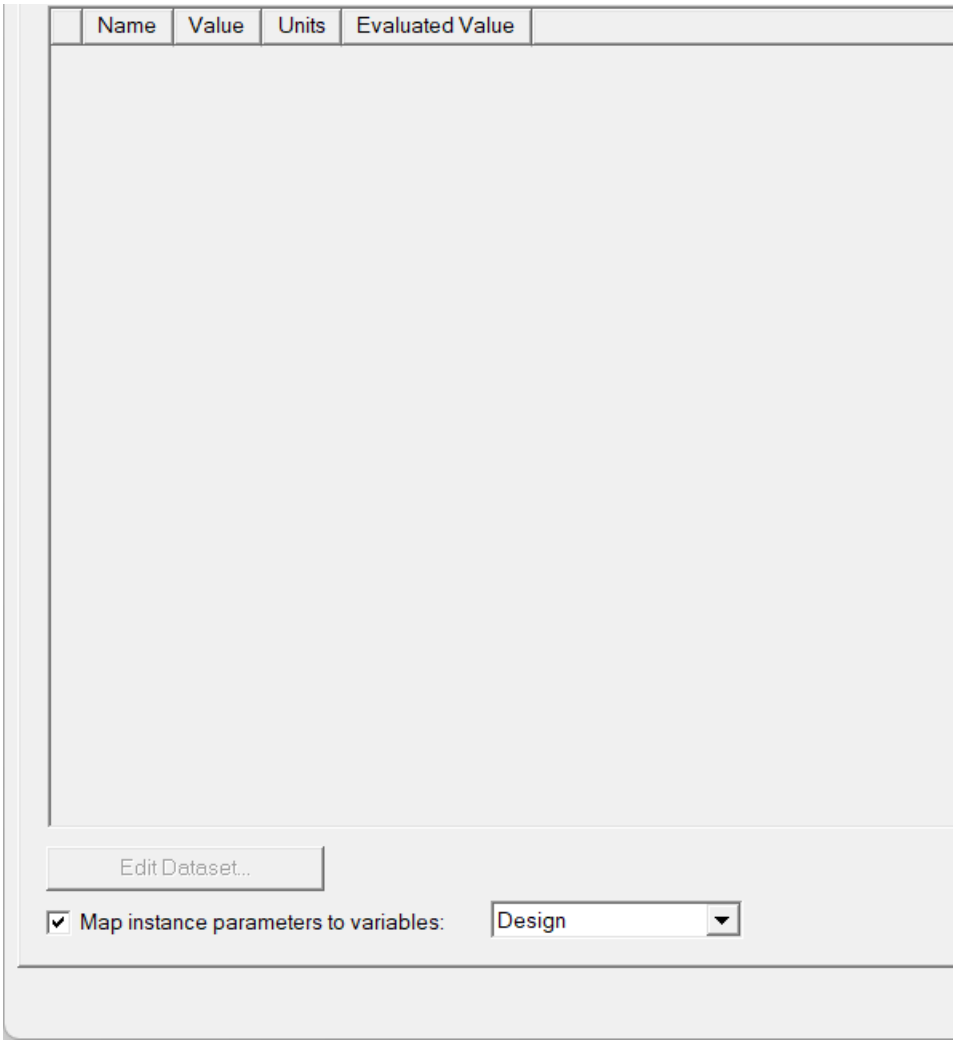


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

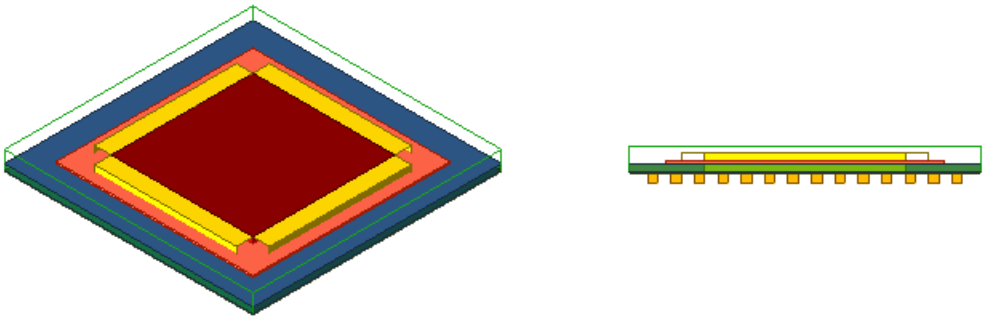


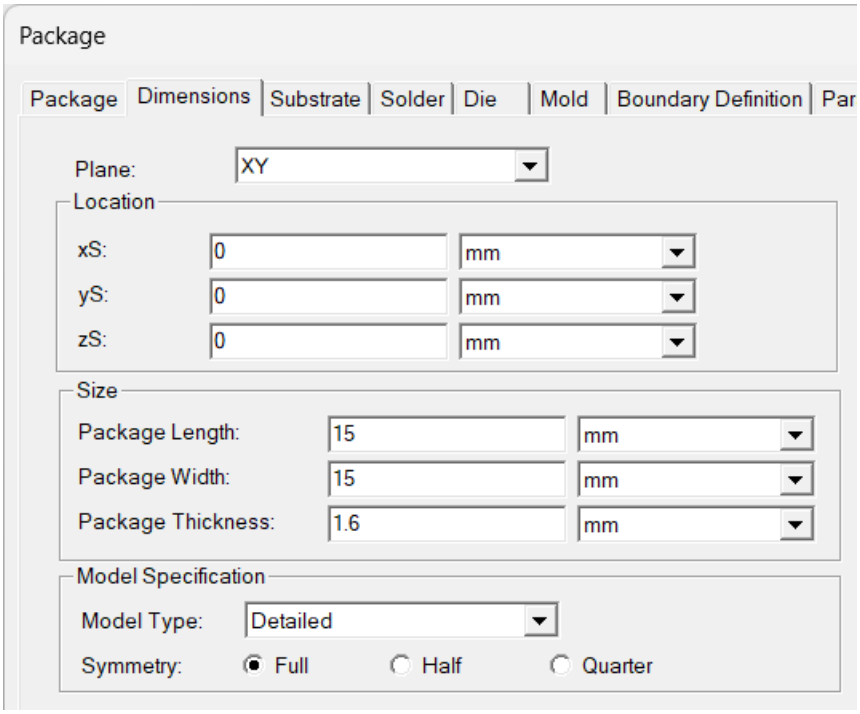
Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Defining FPGBA Parameters

FPBGA enables you to create geometry and boundary conditions to represent a fine pitch ball grid array package.

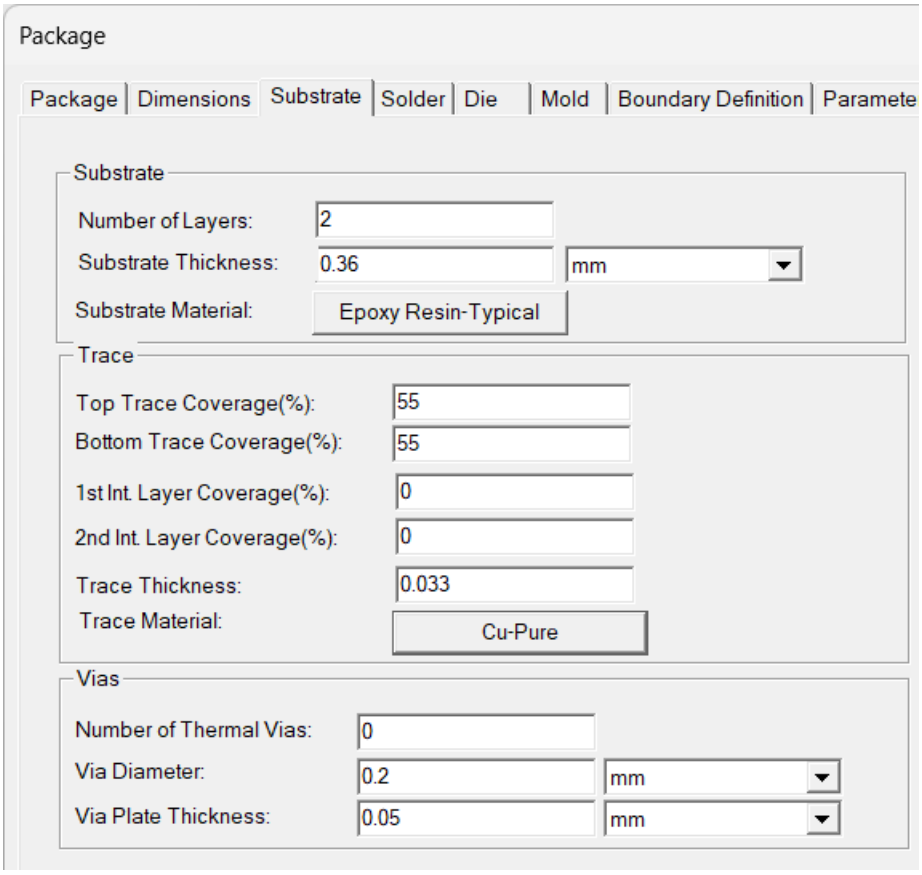
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





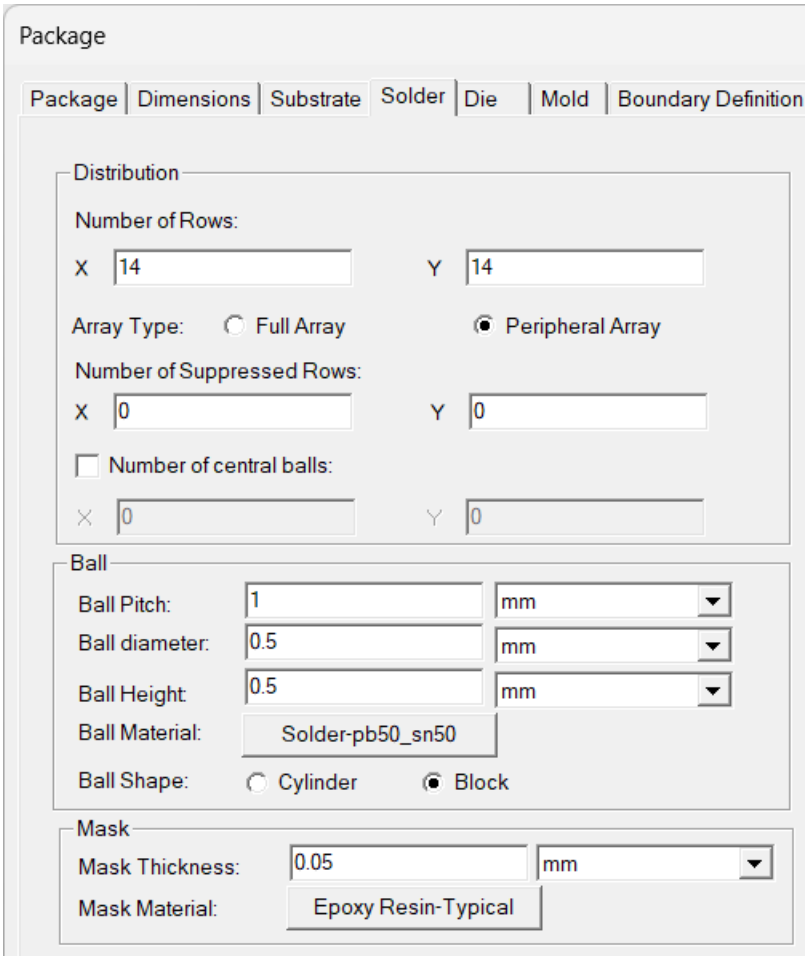
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



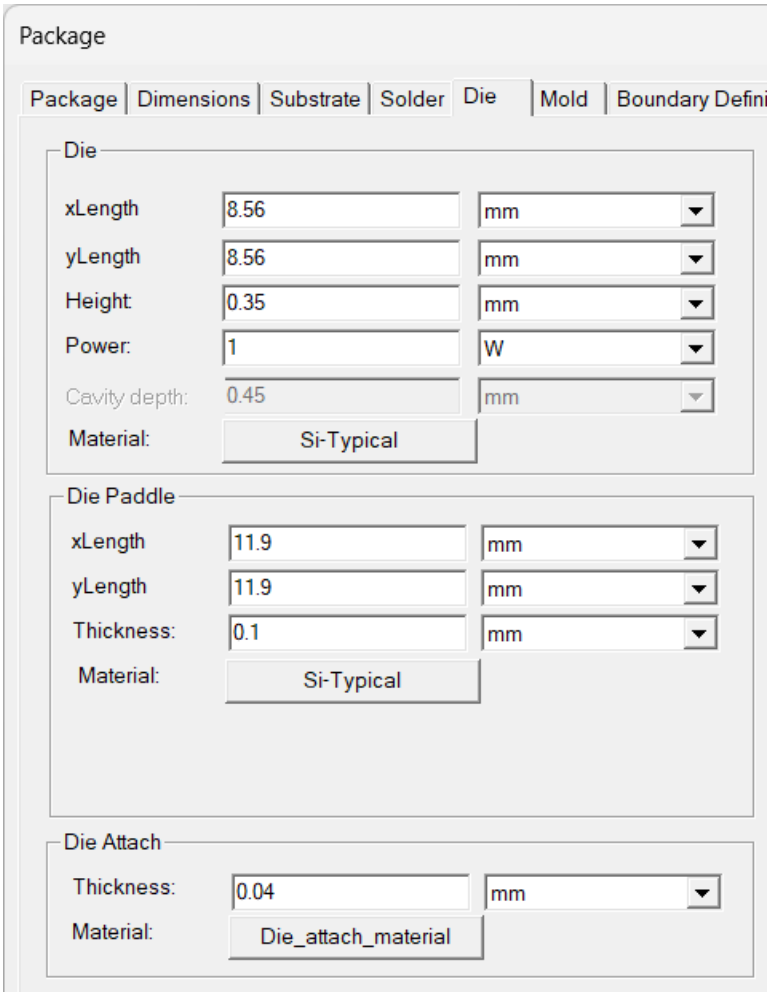
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



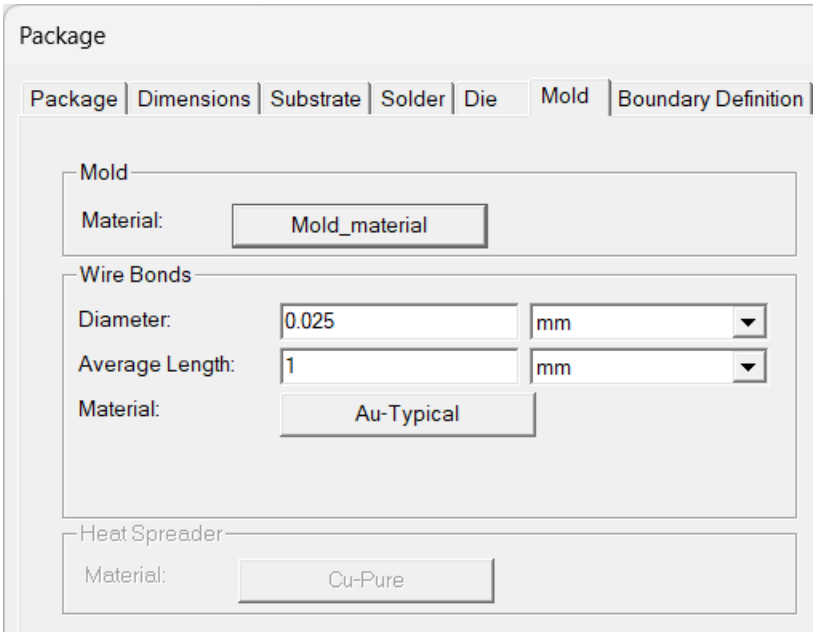
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed rows and number of central balls.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.
Mask	Under **Mask**, enter the solder ball mask thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



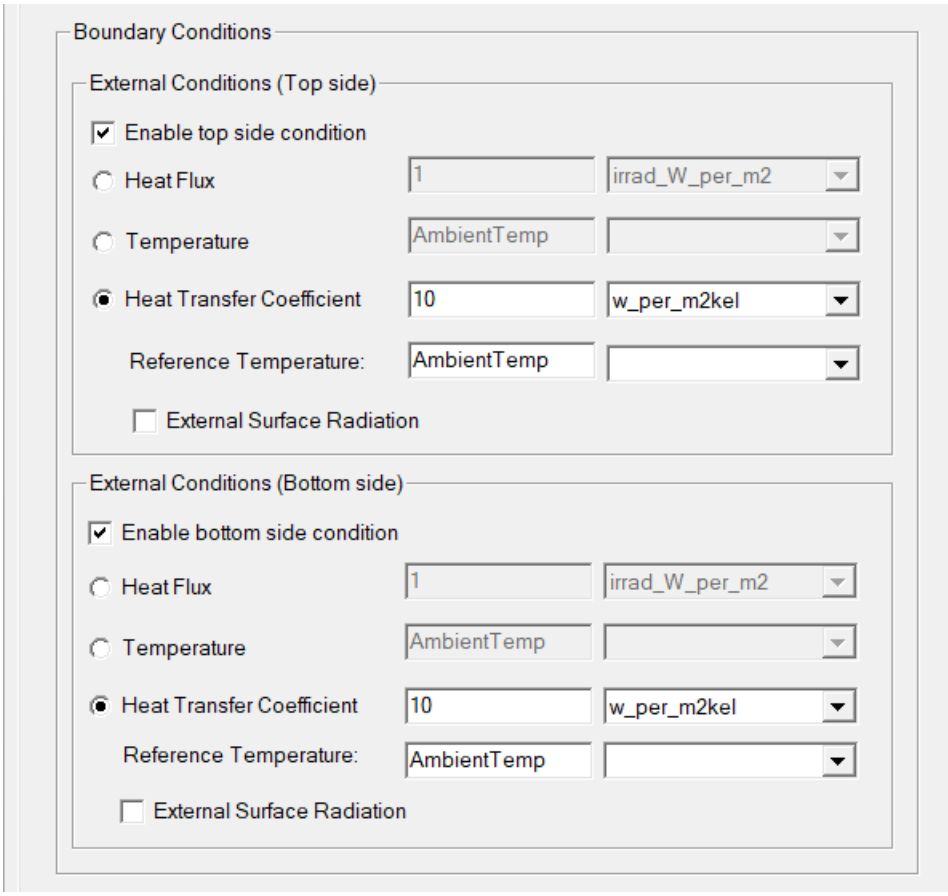
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

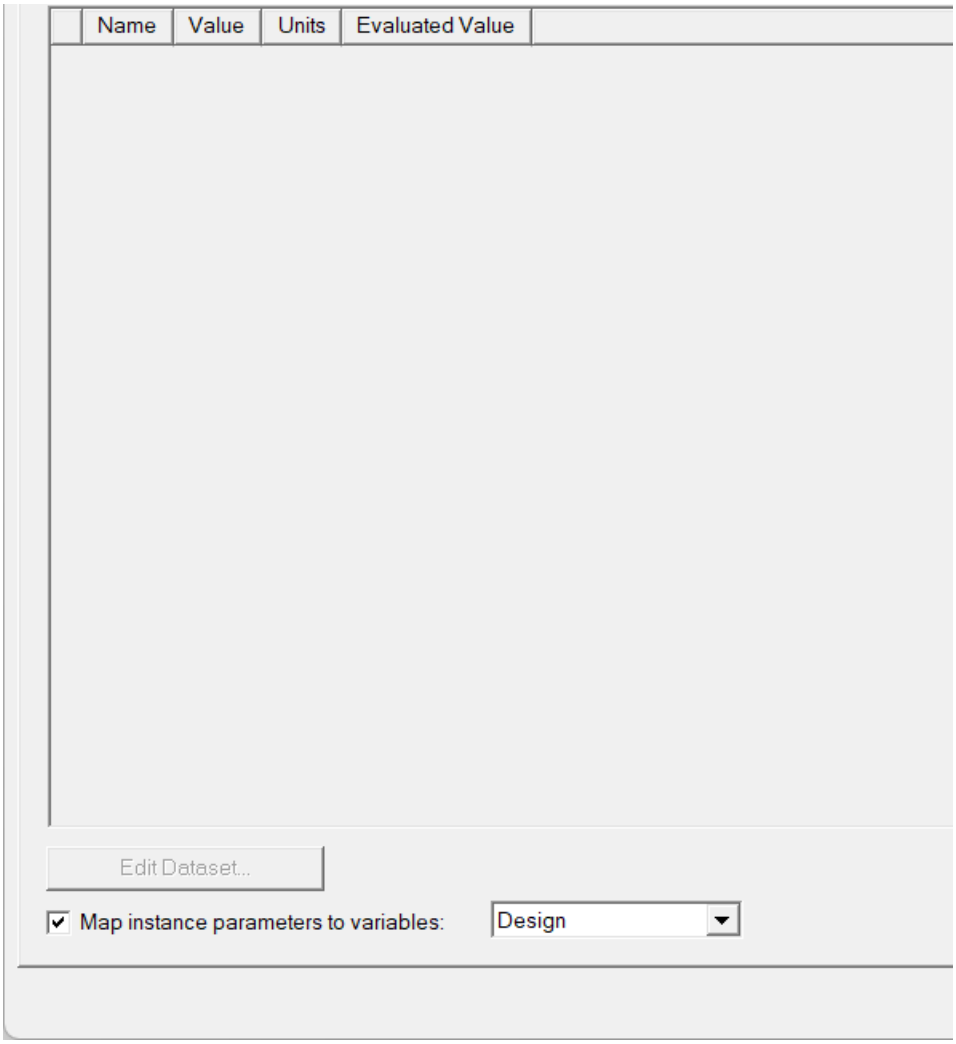


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

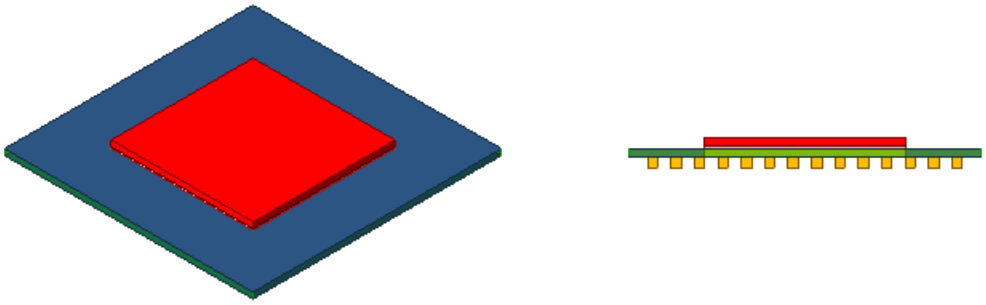


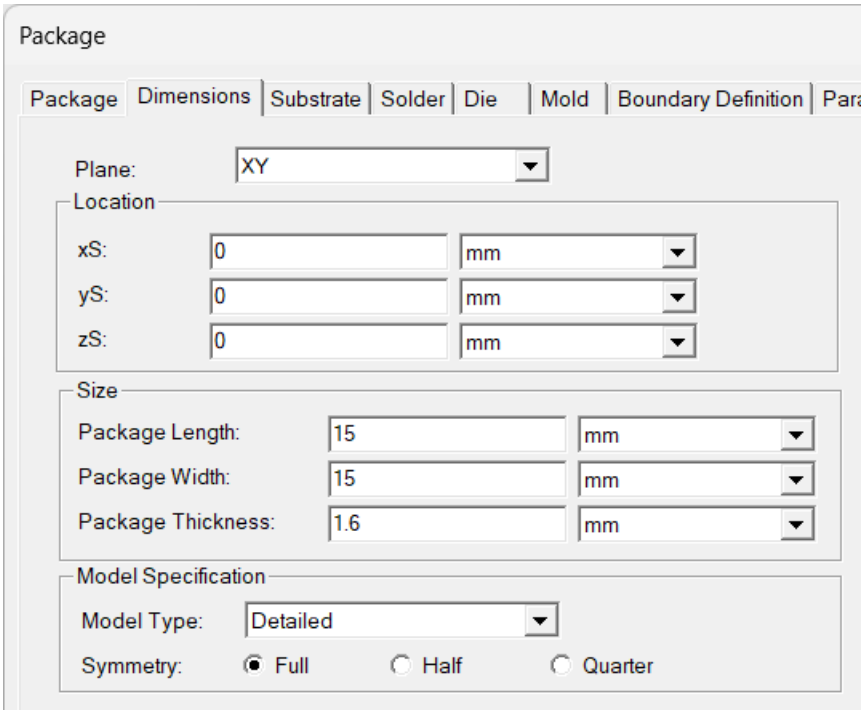
Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Defining Flip-chip Parameters

Flip- chip BGA enables you to create geometry and boundary conditions to represent a flip-chip ball grid array package

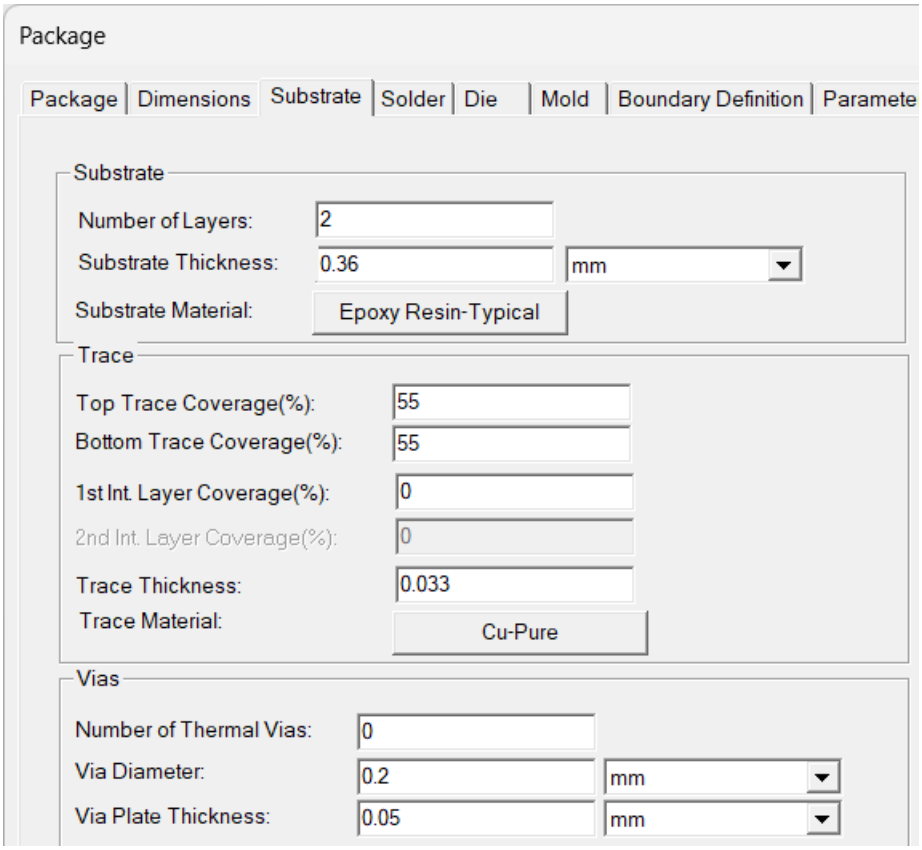
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





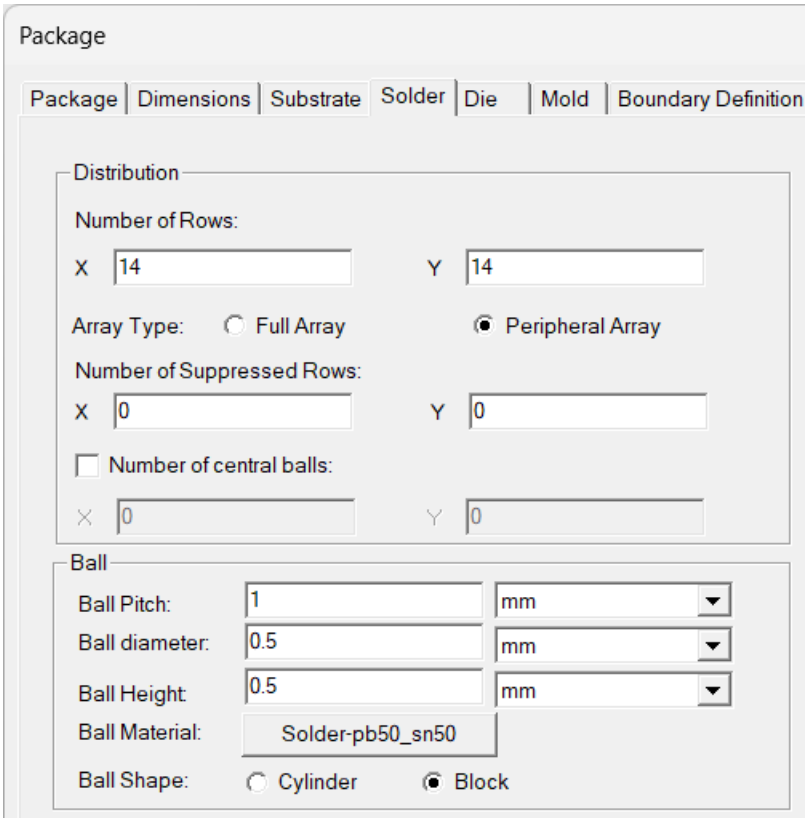
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specs	Under **Model Specs**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



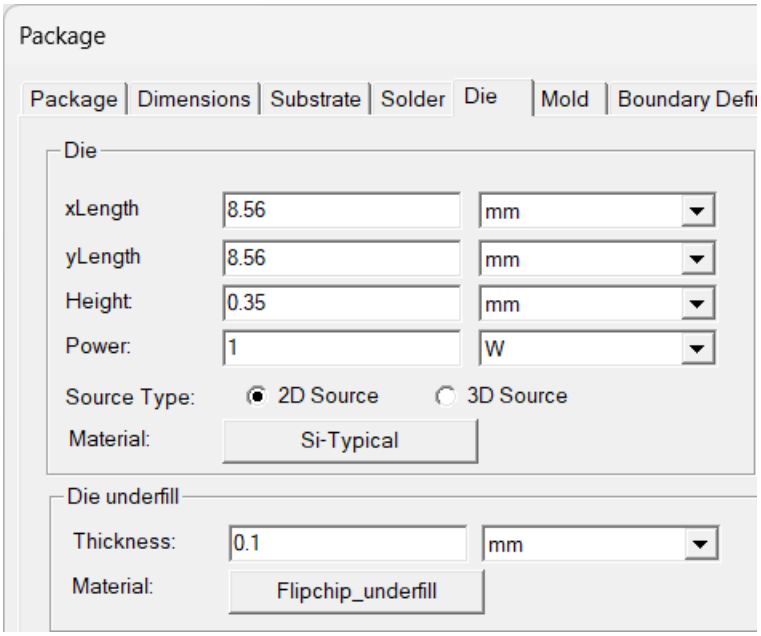
Substrate	Under **Substrate**, specify the **Number of Layers**, **Substrate Thickness**, and **Substrate Material**.
Trace	Under **Trace**, specify the coverage percentage for the top trace, bottom trace, first internal layer, and second internal layer. Also specify the **Trace Thickness** and **Trace Material**.
Vias	Under **Vias**, specify the number of vias and via diameter and thickness.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



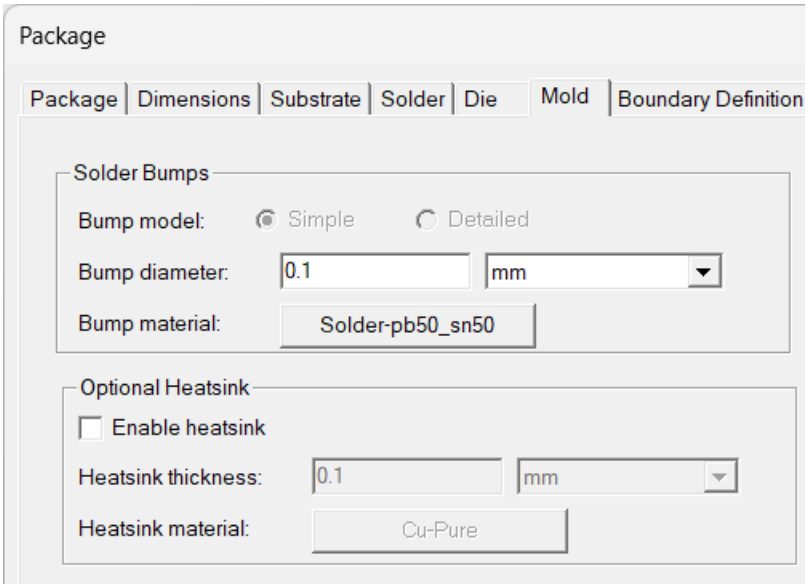
Distribution	Under **Distribution**, enter the number of rows for solder balls, and select the **Array Type**. For a **Peripheral Array**, specify the number of rows to be suppressed rows and number of central balls.
Ball	Under **Ball**, specify the dimensions of the solder balls, including pitch, diameter, and height. Also, specify the material for the solder balls.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



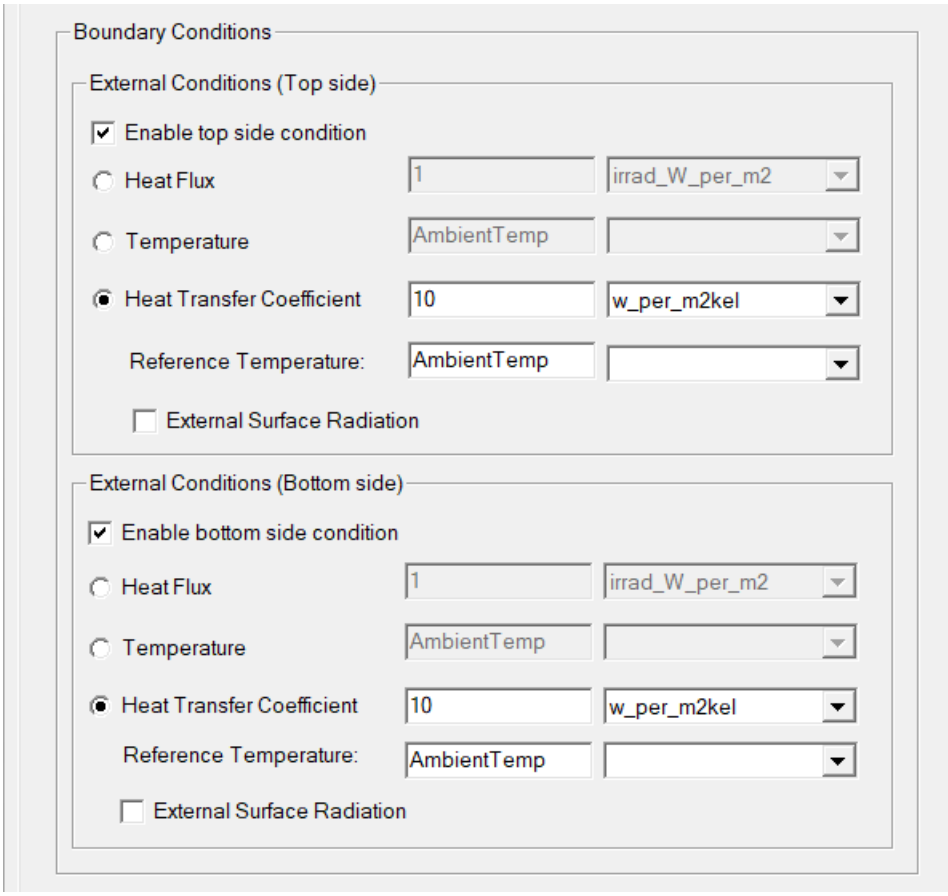
Die	Under **Die**, specify the length of the die in each coordinate direction. Also, specify the die power, source type, and material.
Die Underfill	Under **Die Attach**, specify the bump thickness of the die underfill material and the material.
Heatsink	Under **Optional Heatsink**, specify whether to include a heatsink in the package. If so, specify the heatsink thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Solder Bumps	Under **Solder Bumps**, specify the bump diameter and select the material for the bumps.
Optional Heatsink	Under **Optional Heatsink**, specify whether to include a heatsink in the package. If so, specify the heatsink thickness and material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

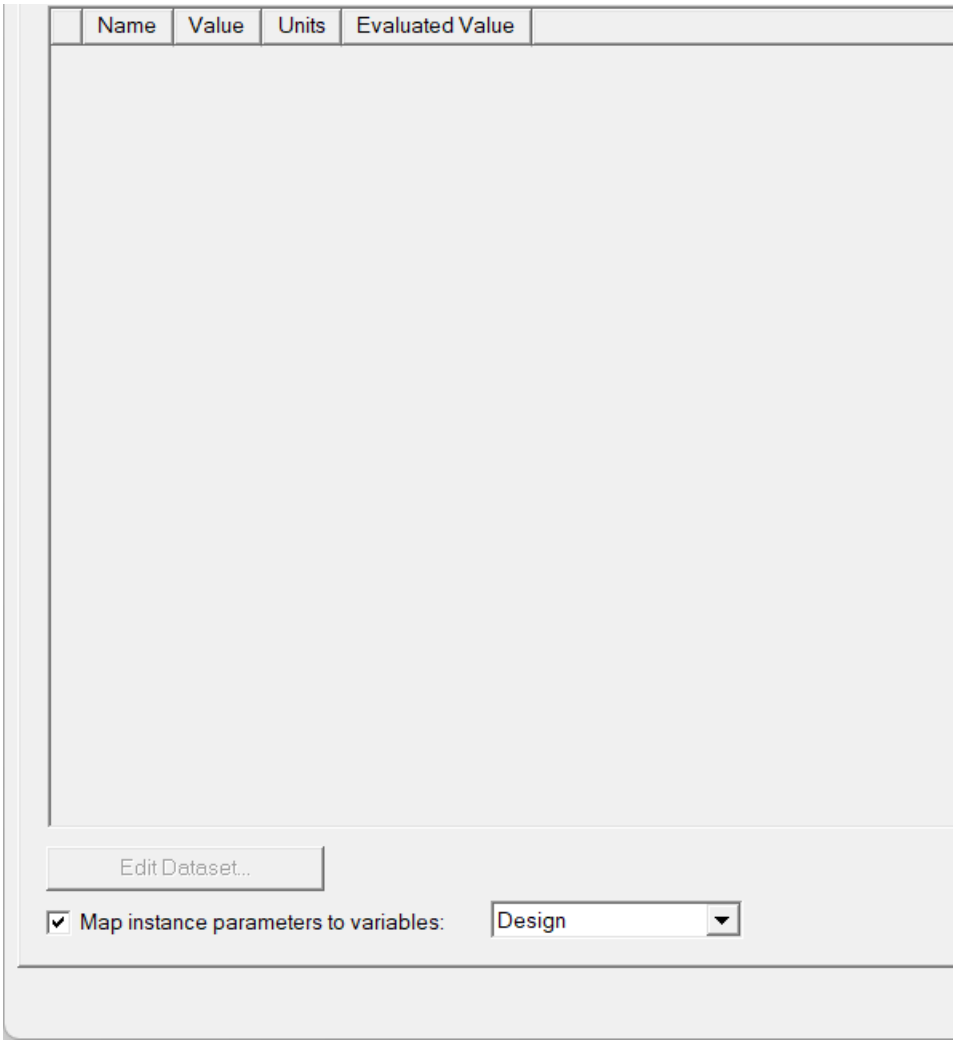


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

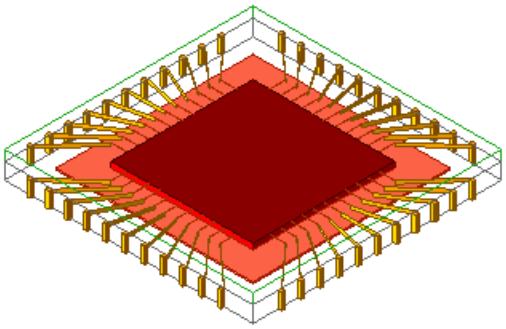


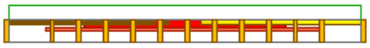
Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

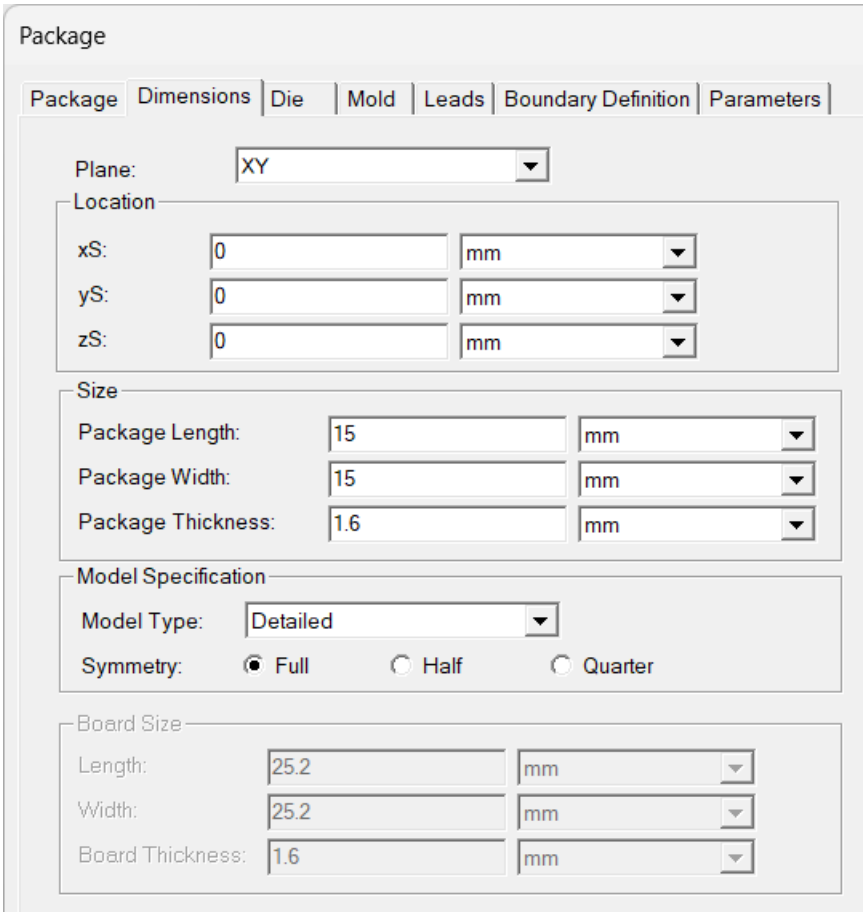
Defining QFP Parameters

QFP enables you to create geometry and boundary conditions to represent a quad flat package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



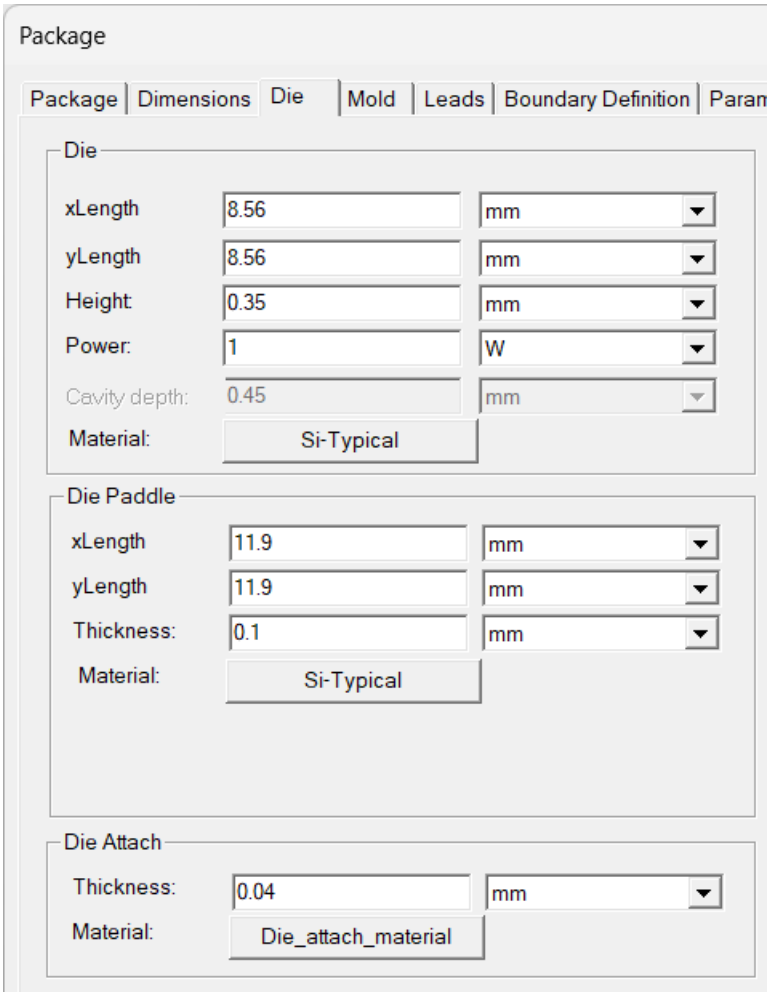




Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Local Origin	Under **Local Origin**, specify the coordinates to define the center from the **Location** of the package is offset. The default setting specifies the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

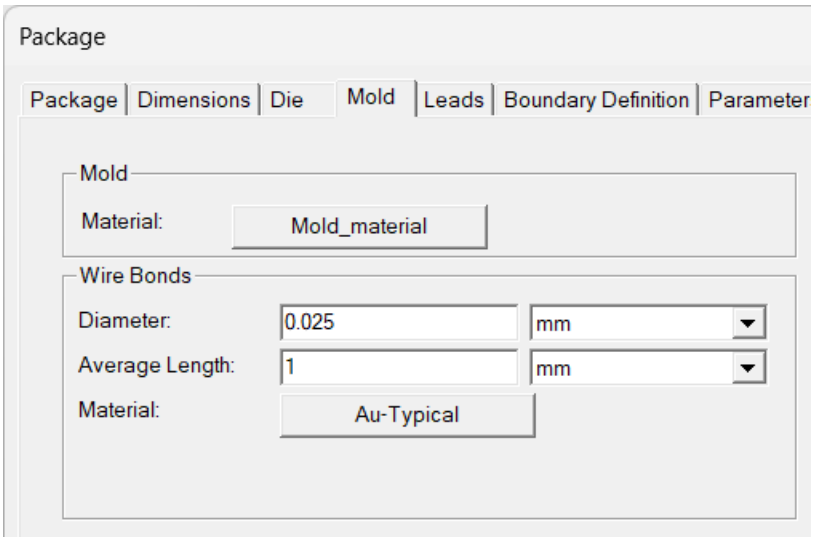
	center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specification	Under **Model Specification**, specify the **Model Type** as **Detailed**, **Compact Conduction Model (CCM)**, **Characterize JC**, or **Characterize JB** and select a **Symmetry**.
Board Size	Under **Board Size**, when the **Model Type** is **Characterize JB**, specify the length of the board in each coordinate direction. The coordinate direction represented by **Board Thickness** depends on the selected **Plane**.



Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.

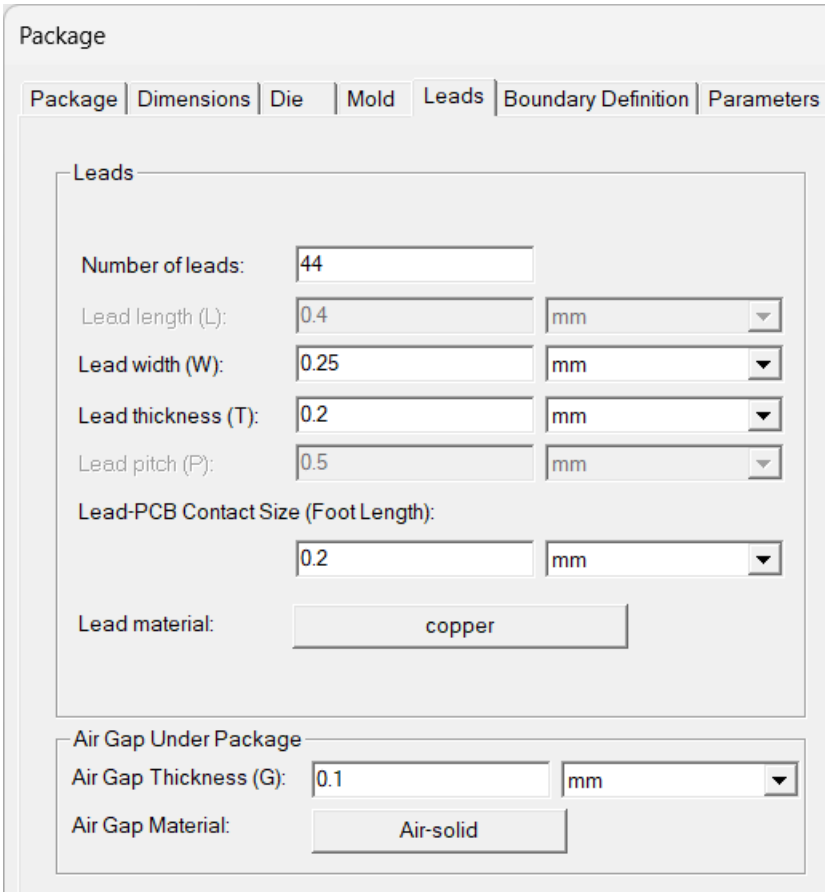
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.



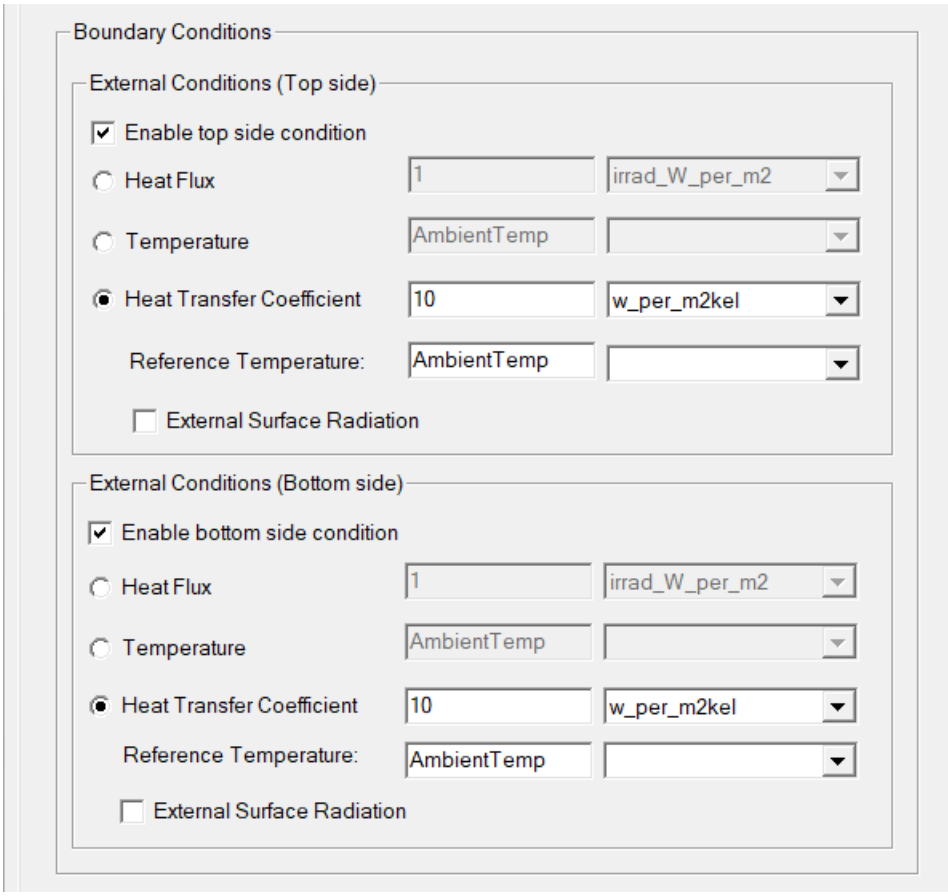
Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads, the lead width and thickness, lead-PCB contact size (foot length), and material.
Air Gap Under Package	Under **Air Gap Under Package**, specify the thickness of the air gap underneath the package. The default material is Air-solid.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

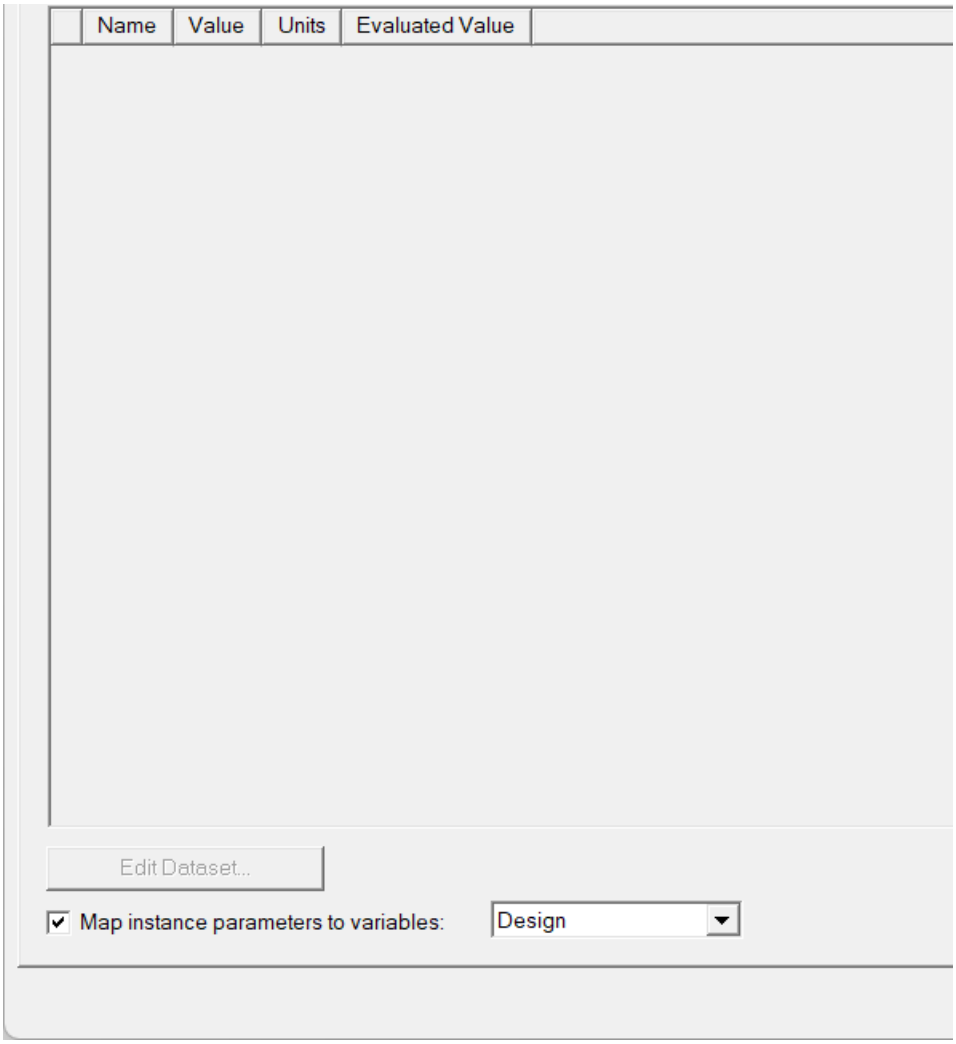


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

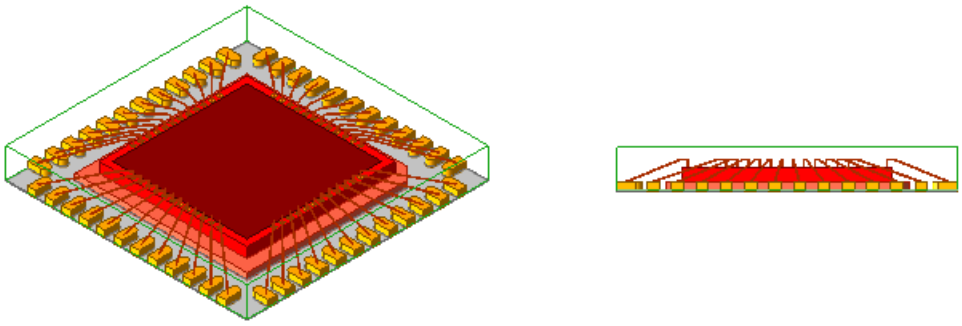


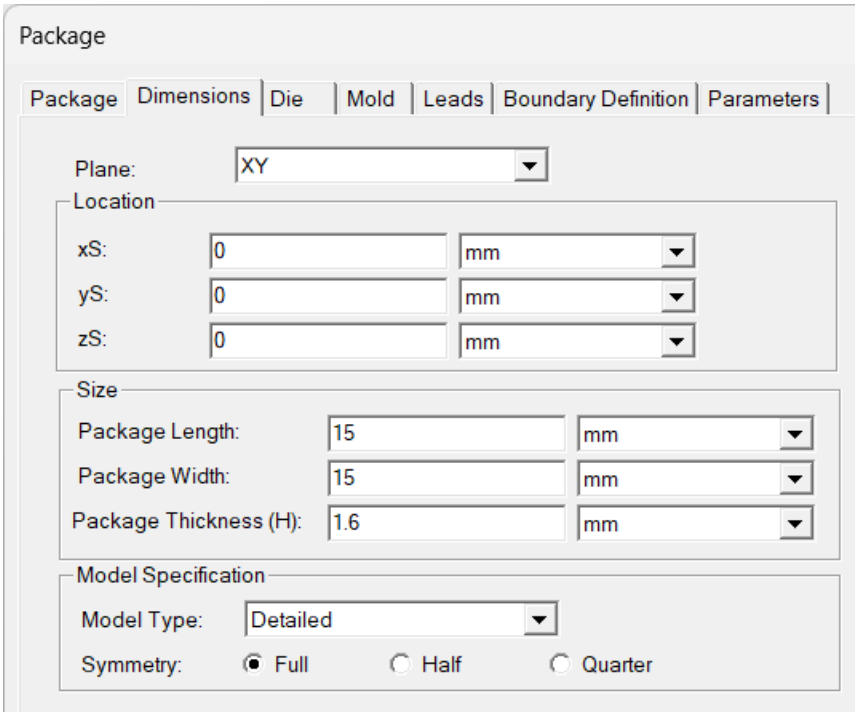
Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Defining QFN Parameters

QFN enables you to create geometry and boundary conditions to represent a quad flat no-lead package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

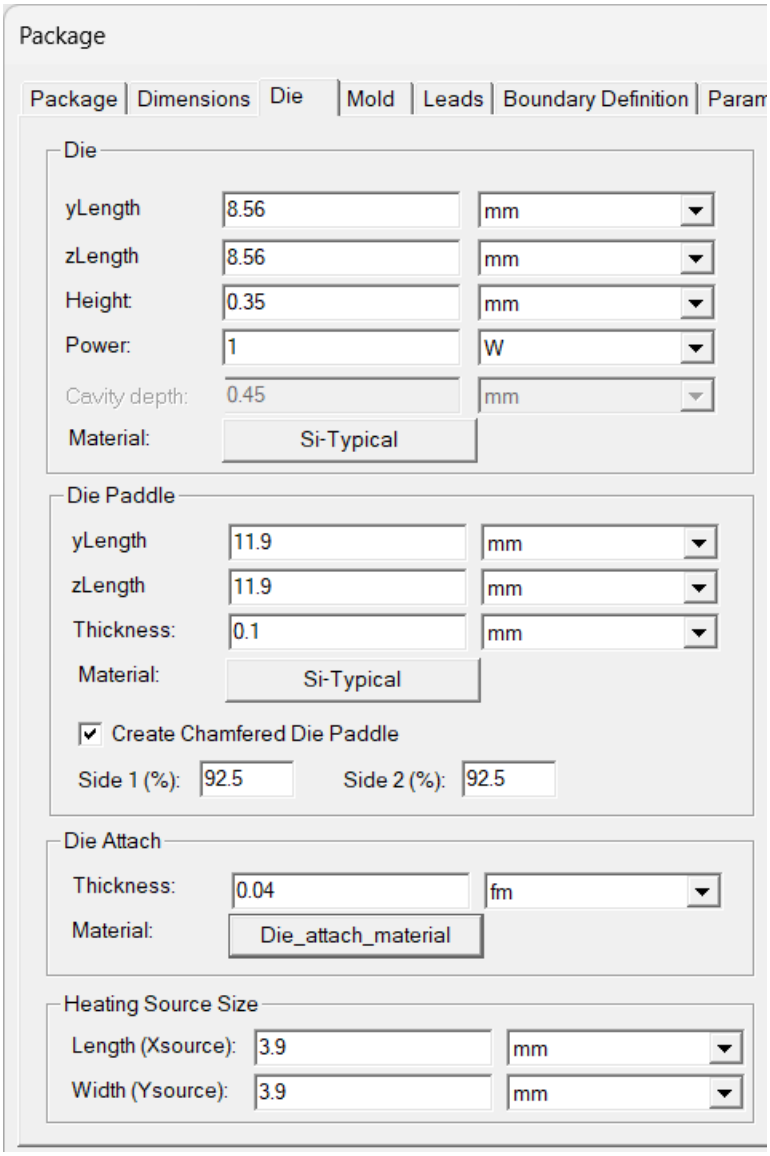




Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specification	Under **Model Specification**, the default specification for the **Model Type** is **Detailed**

.

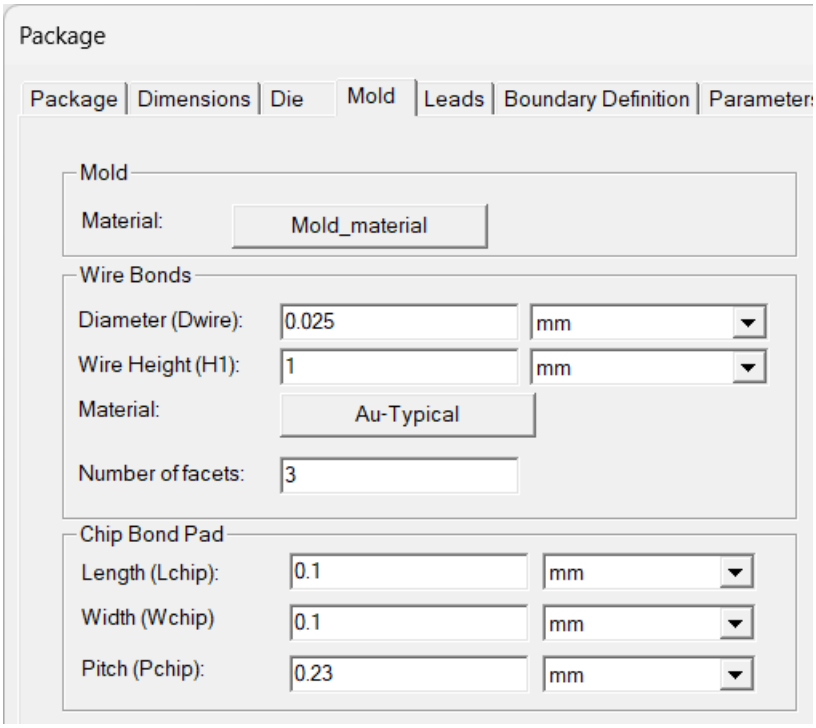
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material. If needed, enable **Create Chamfered Die Paddle** and specify **Side 1 (%)** and **Side 2 (%)** to specify the shape of the die.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

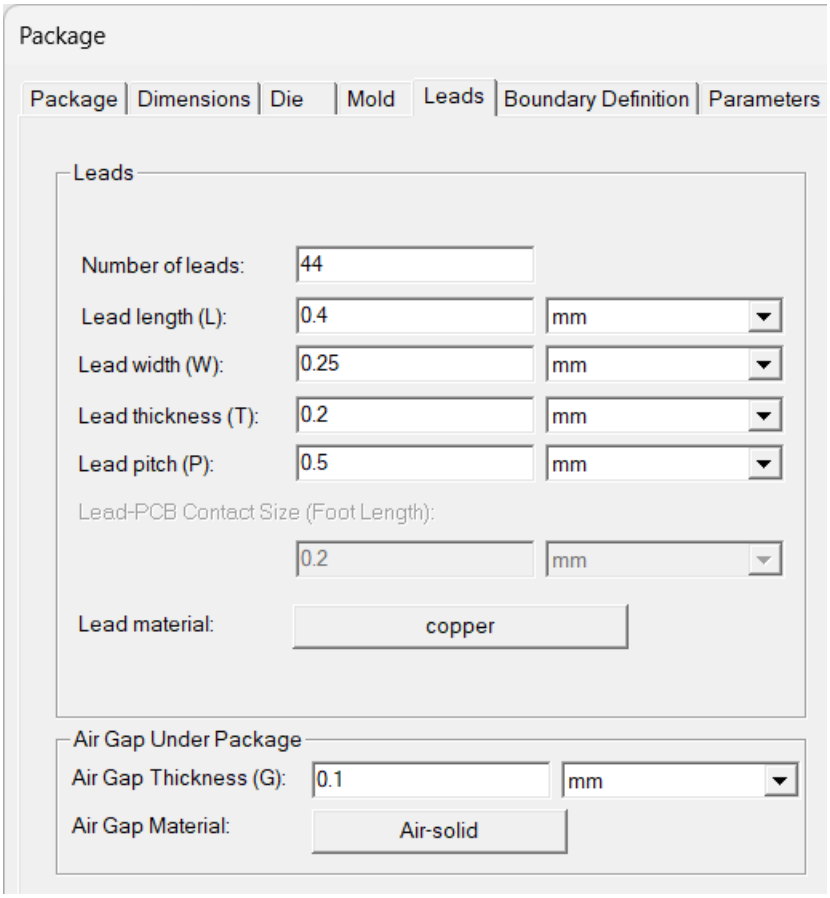
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heating Source Size	Under **Heating Source Size**, specify the length and width of the die.



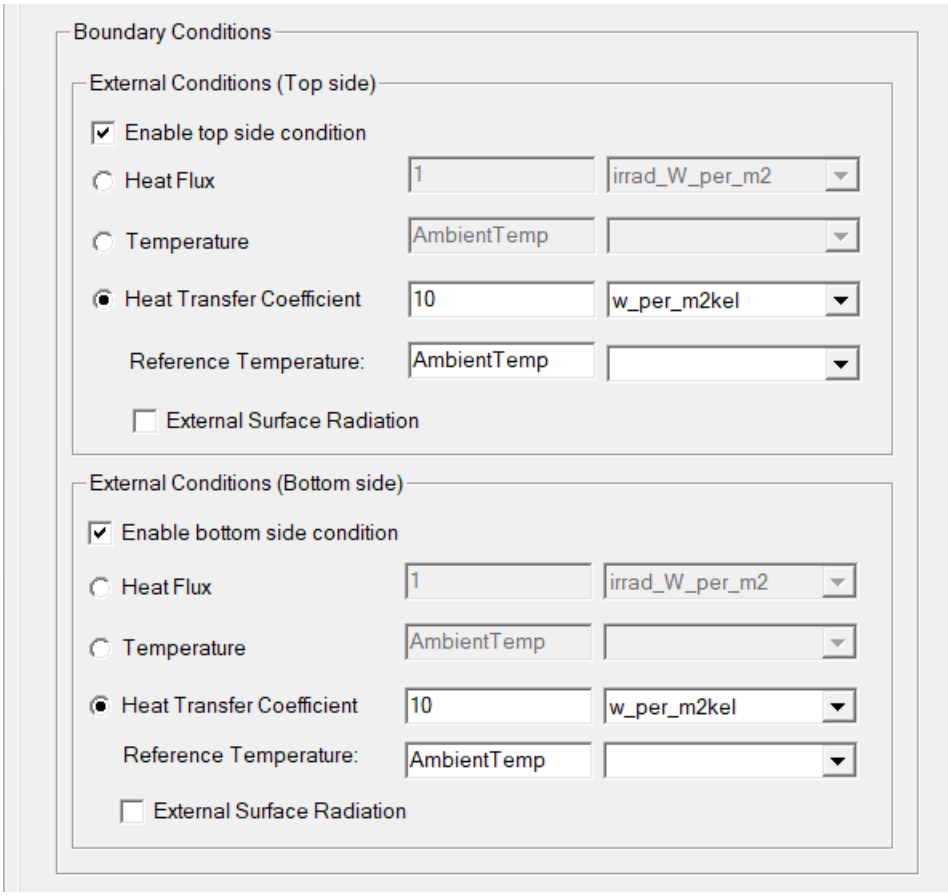
Mold	Under **Mold**, specify the **Material** for the package mold.
Wire	Under **Wire**, specify the wire diameter, number of facets, height, and material.
Chip Bond Pad	Under **Chip Bond Pad**, specify the length, width, and pitch of the chip bond pad.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads. Also specify the lead length, width, thickness, pitch, and material.
Air Gap Under Package	Under **Air Gap Under Package**, specify the thickness of the air gap underneath the package. The default material is Air-solid.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

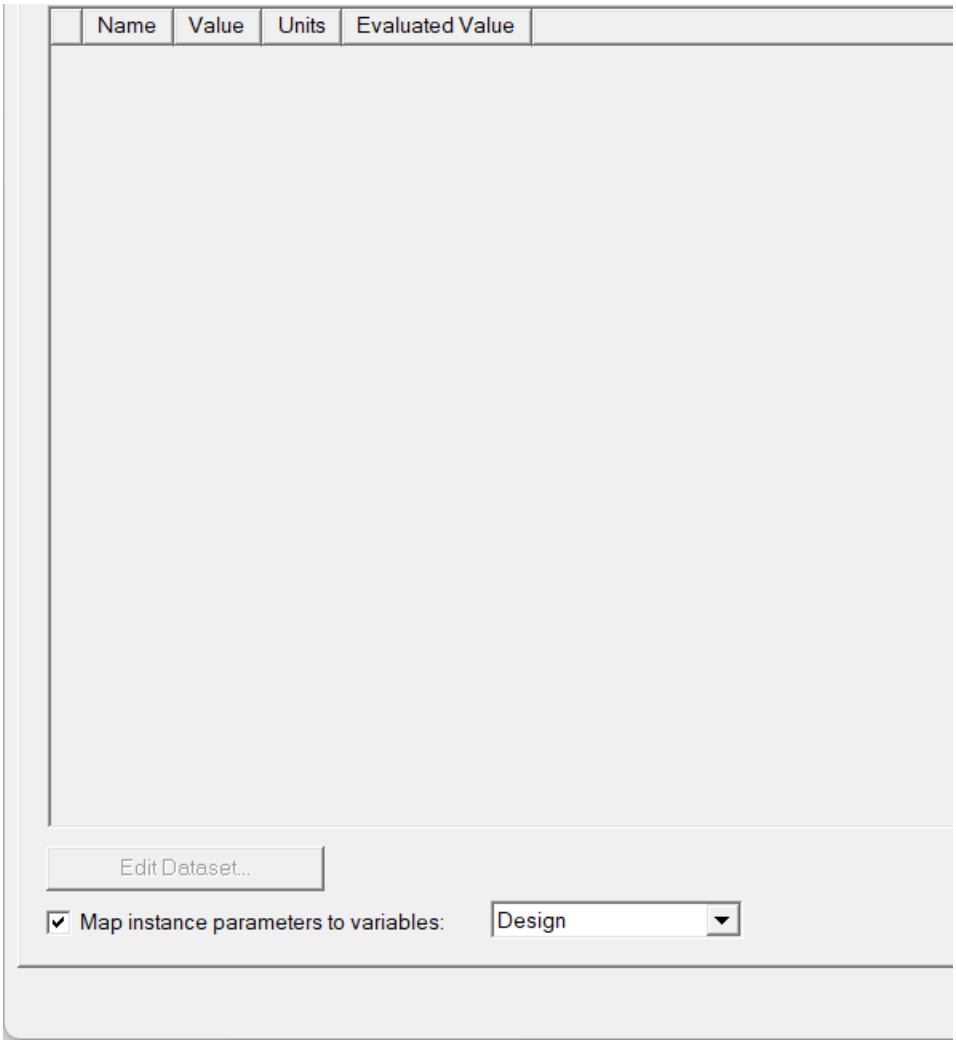


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

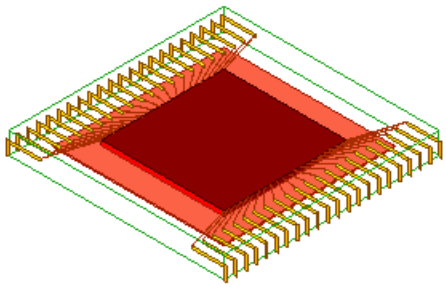


Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

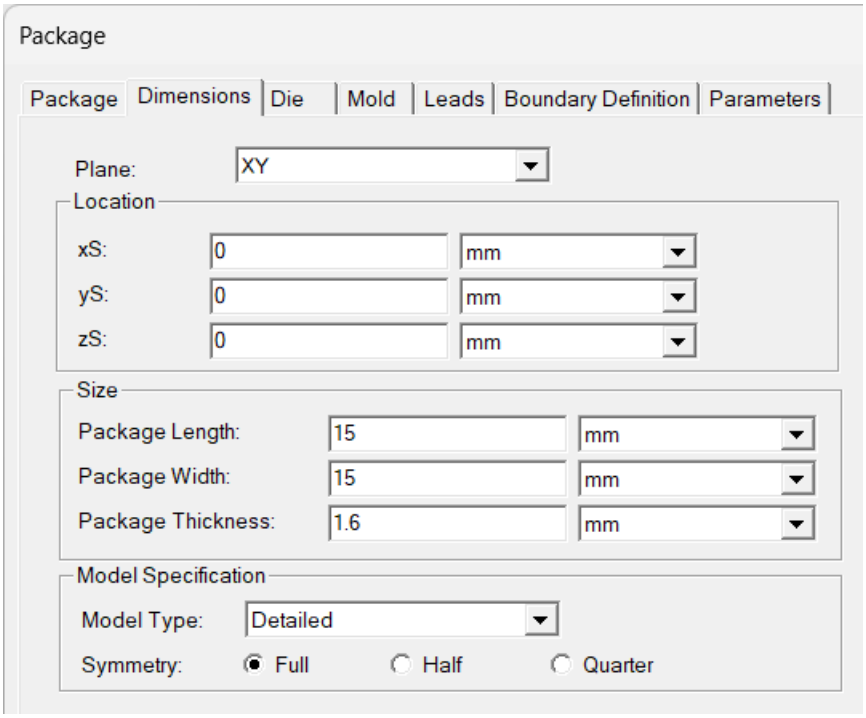
Defining Dual Parameters

Dual enables you to create geometry and boundary conditions to represent a DUAL package.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

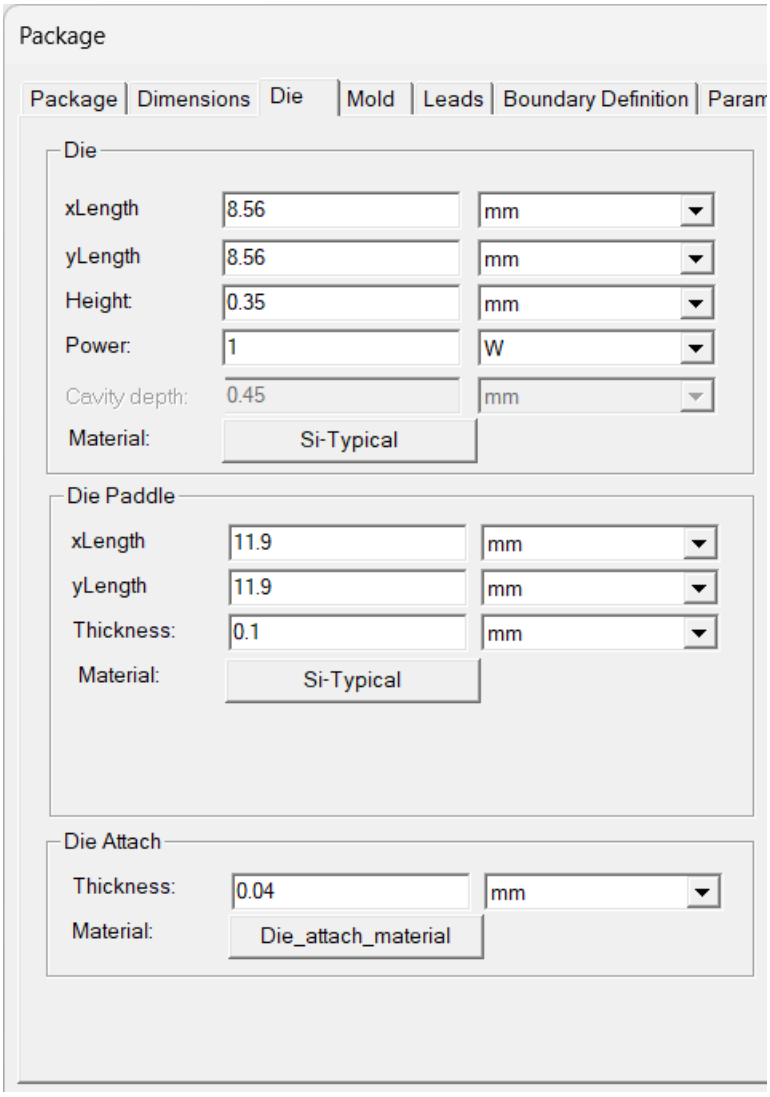






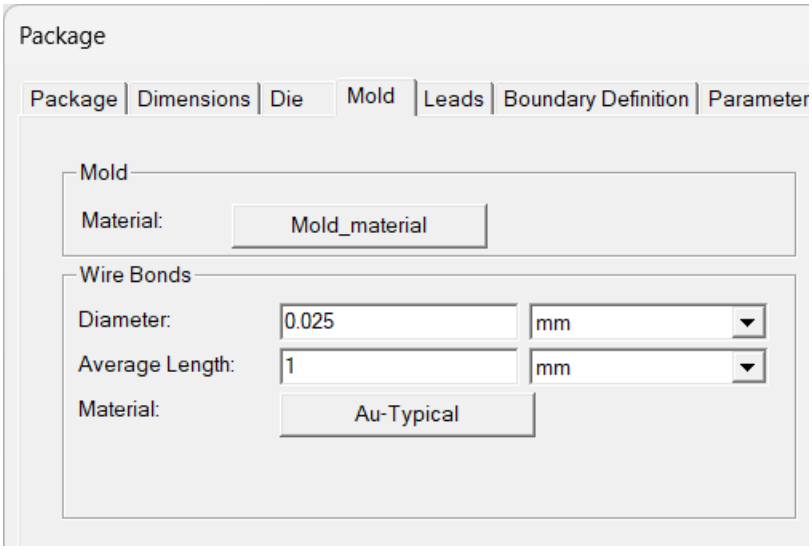
Plane	**Plane** specifies the plane on which to create the package.
Location	Under **Location**, specify the starting coordinates for the package geometry relative to center of the **3D Modeler**.
Size	Under **Size**, specify the length, width, and thickness of the package.
Model Specification	Under **Model Specification**, specify the **Model Type** as **Detailed** or **Compact Conduction Model (CCM)** and select a **Symmetry**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



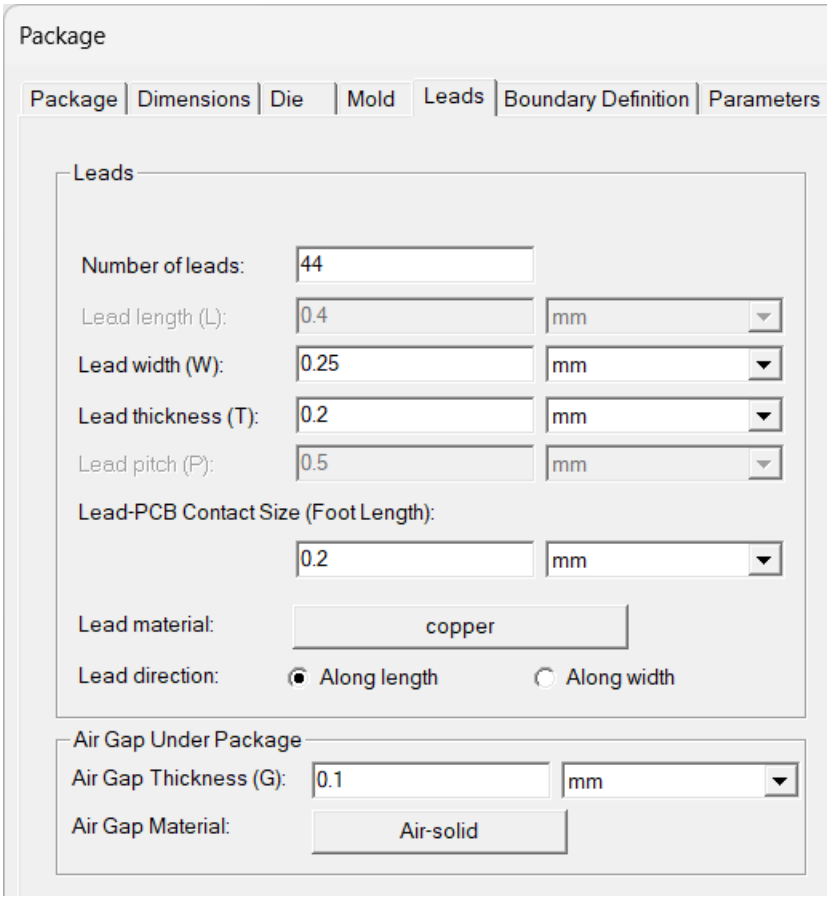
Die	Under **Die**, specify the length of the die in each coordinate direction. The coordinate direction represented by **Height** depends on the selected **Plane**. Also, specify the die power and material.
Die Paddle	Under **Die Paddle**, specify the length of the die paddle in each coordinate direction. The coordinate direction represented by **Thickness** depends on the selected **Plane**. Also, specify the die paddle material.
Die Attach	Under **Die Attach**, specify the thickness of the die attach material and the material.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



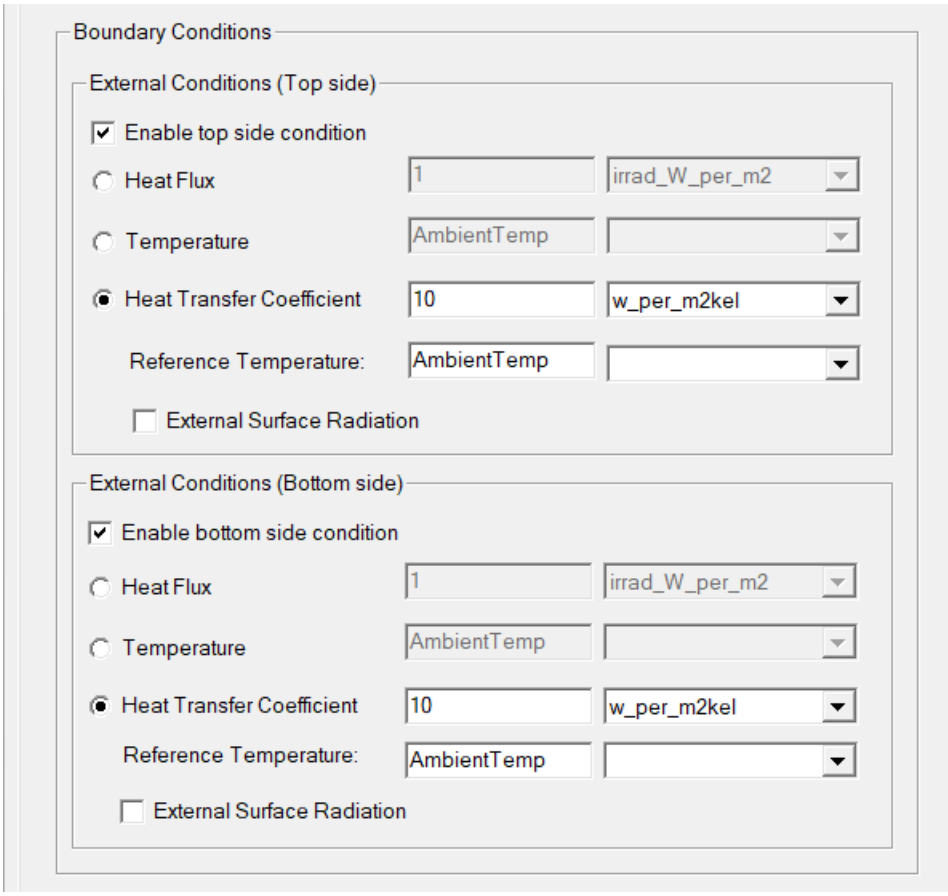
Mold	Under **Mold**, select the material for the mold.
Wire Bonds	Under **Wire Bonds**, specify the diameter, average length, and material of the of the wire bonds.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leads	Under **Leads**, specify the number of leads, the lead width and thickness, lead-PCB contact size (foot length), material, and the direction the leads will point.
Air Gap Under Package	Under **Air Gap Under Package**, specify the thickness of the air gap underneath the package. The default material is Air-solid.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

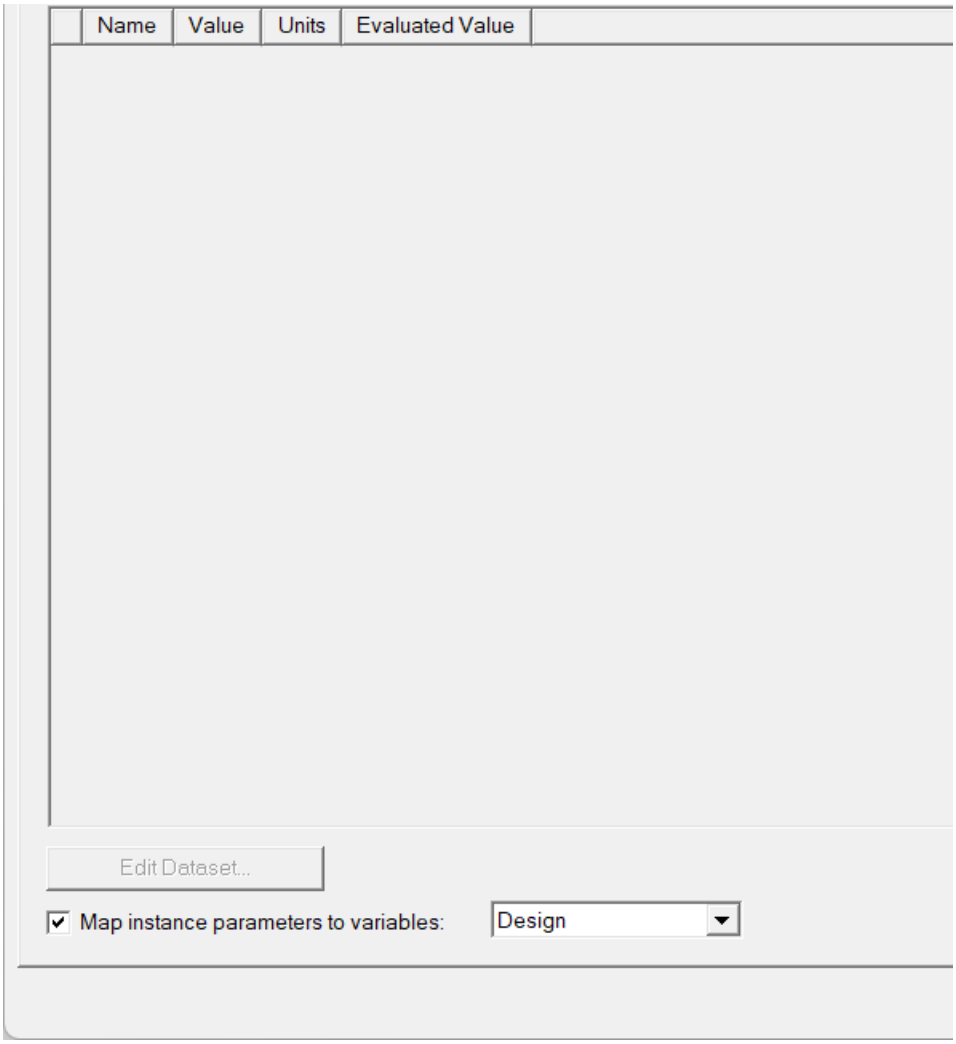


Note:

Settings on the **Boundary Conditions** tab are only applied for conduction-only simulations (i.e., when the **Problem Type** is set to **Temperature** only).

Boundary Conditions	Under **Boundary Conditions**, select **Enable top/bottom side condition** to specify which side(s) of the package are assigned thermal boundary conditions.
External Conditions (**Top** and **Bottom** sides)	When enabled, select **Heat Flux**, **Temperature**, or **Heat Transfer Coefficient** to define the external thermal conditions. For **Heat Transfer Coefficient**, enter a **Reference Temperature** and, if needed, enable **External Surface Radiation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** of the variable is displayed in the Properties window and Properties dialog box when the component is selected.
Value	The **Value** is used in the analysis.
Units	**Units** is the unit of measure for the parameter.
Edit Dataset	Click **Edit Dataset** to input dataset values for the variable.
Map instance parameters to variables	Enable **Map instance parameters to variables** and select either **Design** or **Project** to map the component variable to a design or project variable.

Defining Native Component Instance Parameters

While creating or editing a native component, you can expose component properties as instance parameters of the component. To do this, define component variables in the native component

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

creation dialog box. The component variables created in the native component creation dialog remain internal to the component and are visible as instance parameters in the **Properties** window when the component instance is selected in the **Project Manager** or the history tree. Once a component instance is created, you can assign design or project variables to instance parameters to perform optimetrics, animations, plots, etc.

While inserting a native component for the first time, an option (**Map instance parameters to variables**) is provided to automatically create design or project variables and assign them to instance parameters.

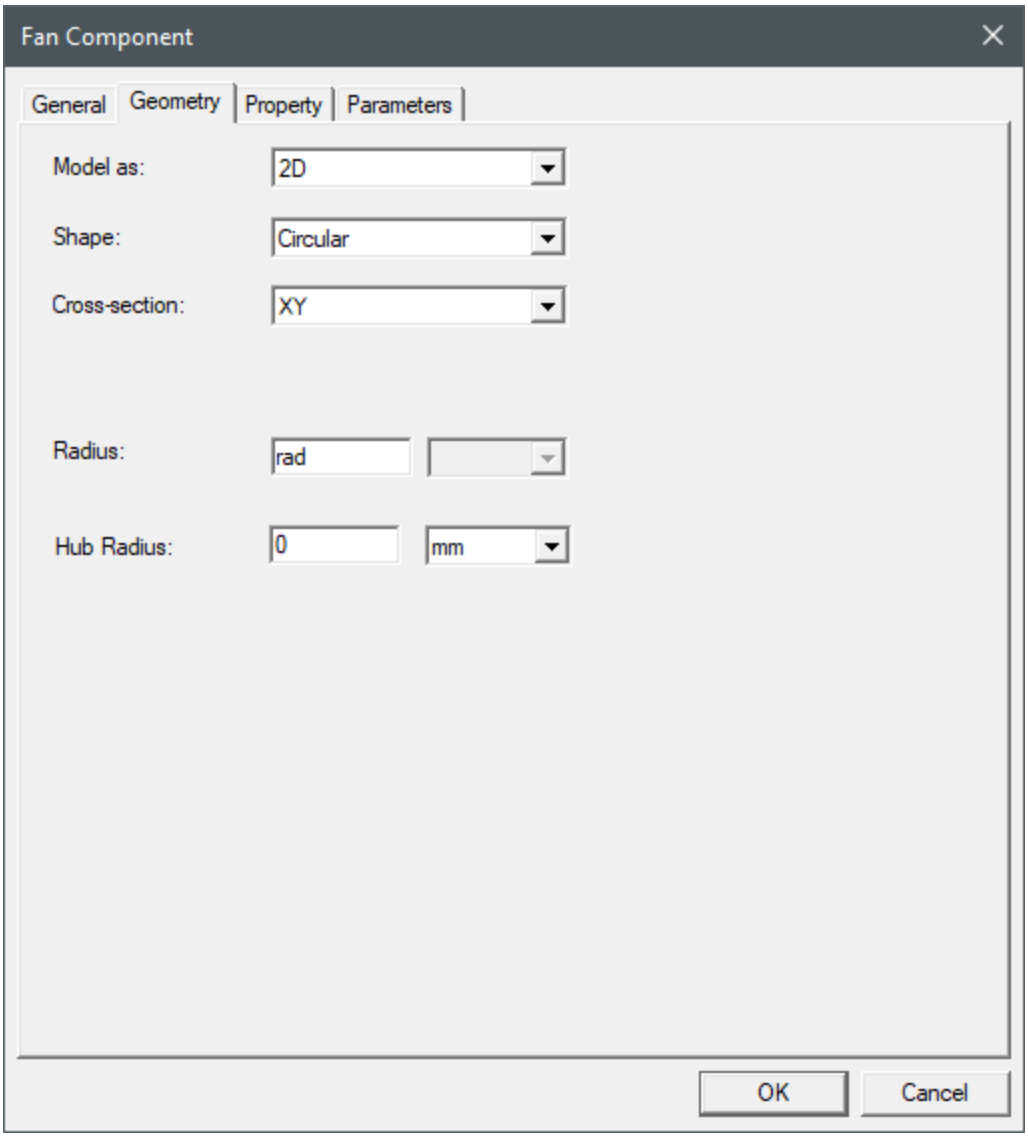
> **Note:** If you select **Map instance parameters to variables**, the design or project variable value is assigned to the instance parameter.

Define an Instance Parameter

The following is a general procedure to define an instance parameter. In this example, a fan component is created with an instance parameter assigned to the radius.

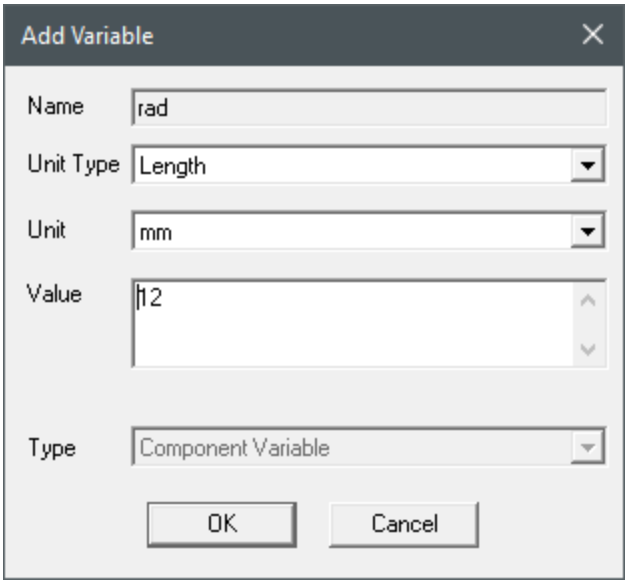
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **Fan**.
2. On **Geometry** tab, enter *rad* in the **Radius** field and press **Enter**.



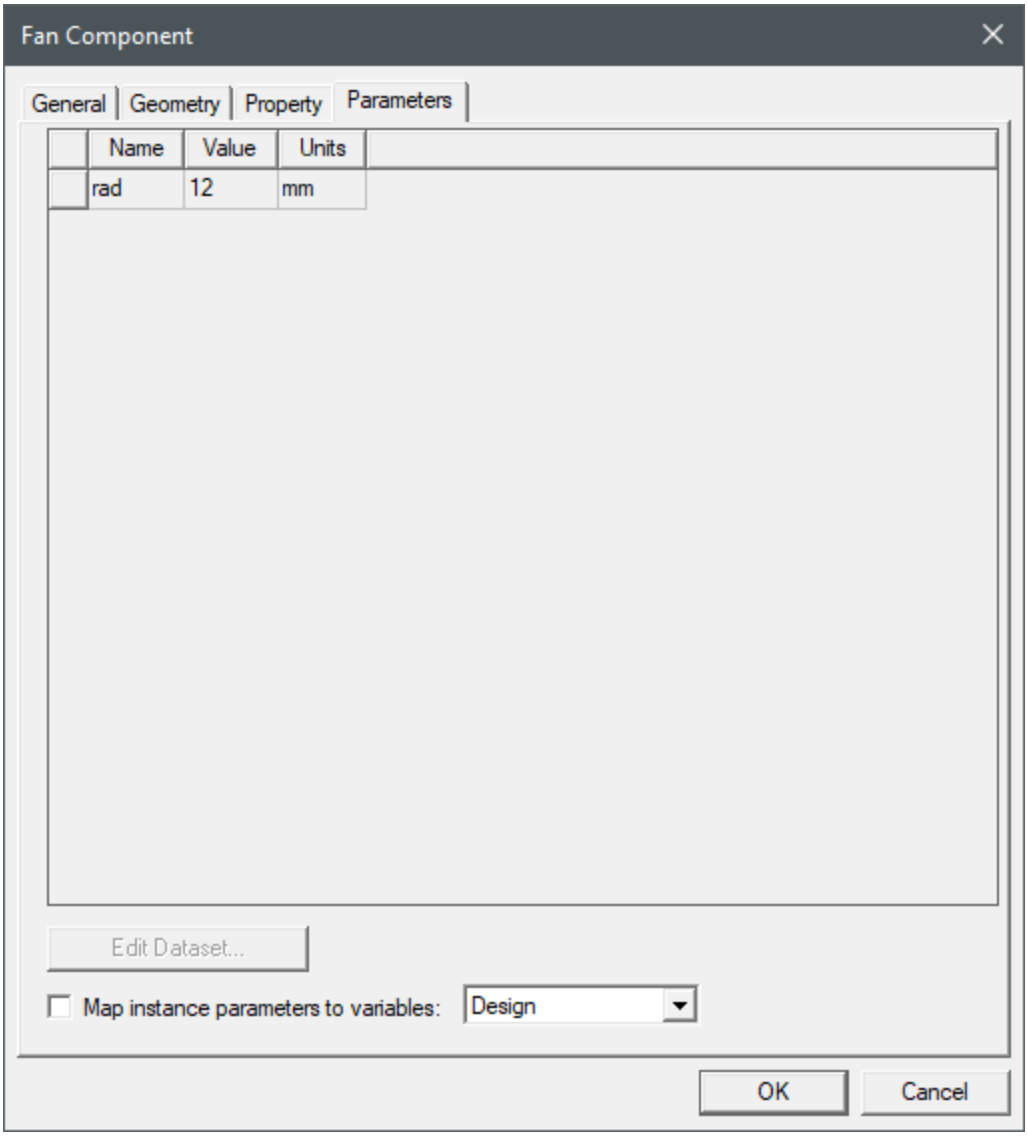
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Add Variable** dialog box, enter *12* for the **Value** and click **OK**.



4. On the **Parameters** tab, note the parameter and definition are displayed. The **Value** can be edited on the **Parameters** tab before creating the component.

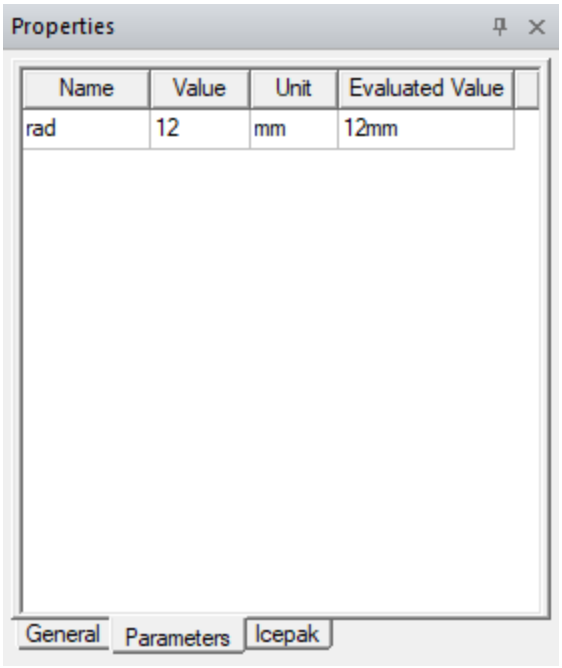
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Deselect the Map instance parameters to variables check box and click **OK** to create the fan.
6. In the **Properties** window, select the **Parameters** tab to view the instance parameter you created. You can edit the **Value** of a parameter by double-clicking it.

Note: After the initial insertion of a native component, the **Map instance parameters to variables** option is not available if you edit the component in its dialog box. You can enter a design or project variable as the **Value** to map an instance parameter to a design or project variable manually.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

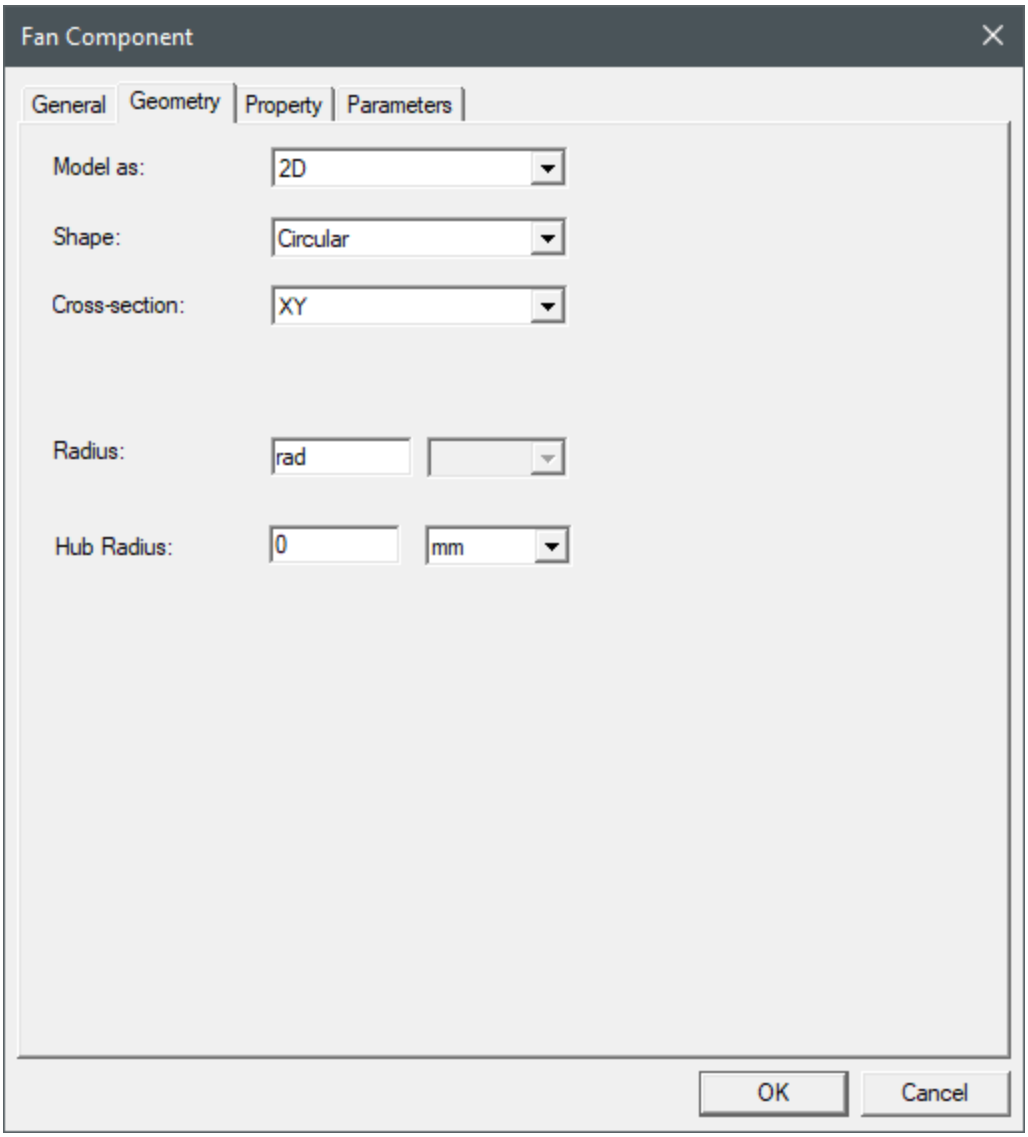


Map an Instance Parameter to a Project or Design Variable

The following is a general procedure to create an instance parameter and map it to a project or design variable. In this example, a fan component is created with a design variable assigned to the instance parameter created for the radius.

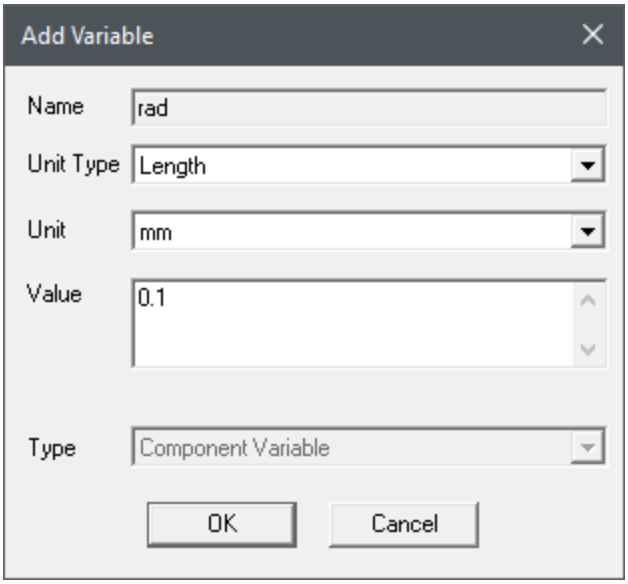
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **Fan**.
2. On **Geometry** tab, enter *rad* in the **Radius** field and press **Enter**.



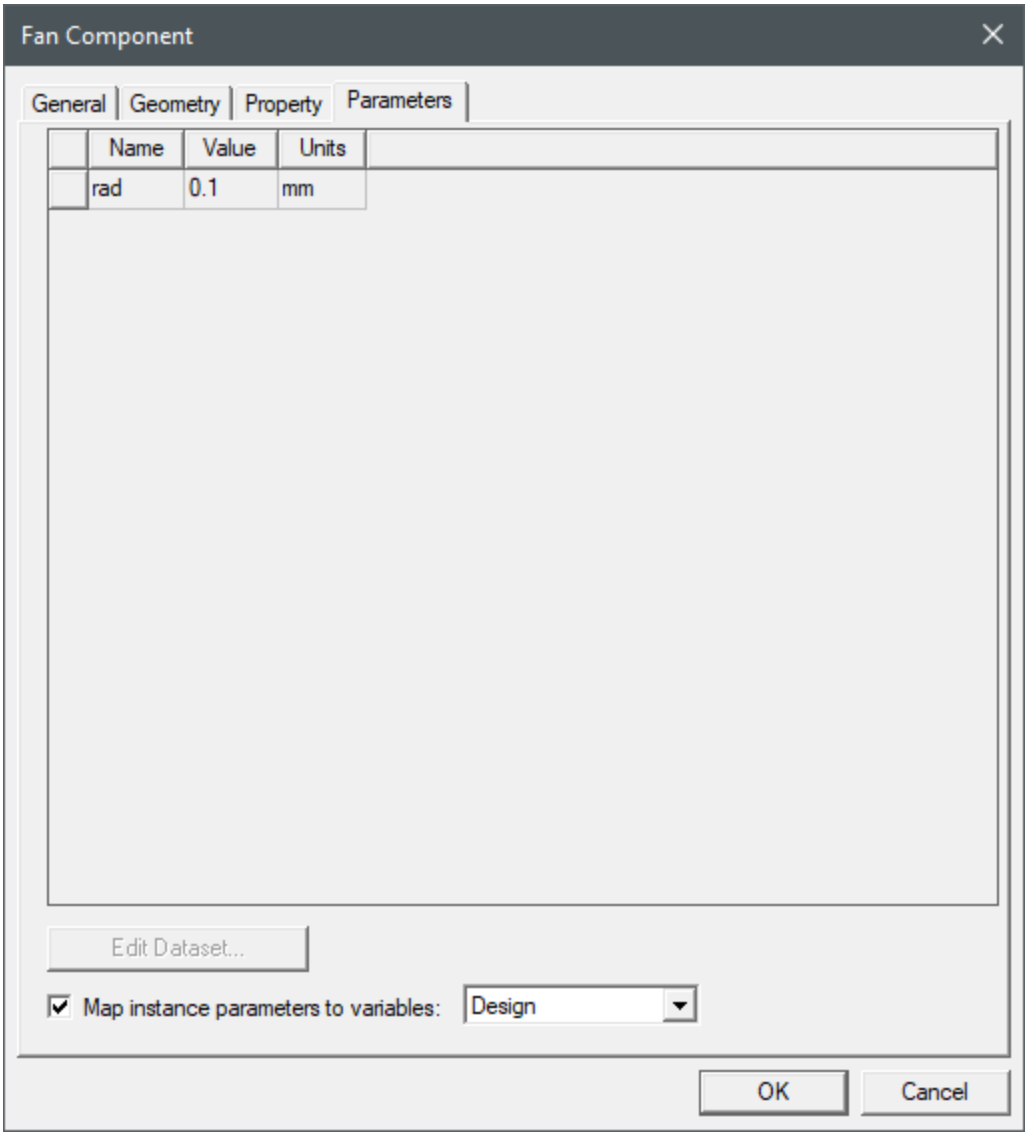
3. In the **Add Variable** dialog box, click **OK**. If a design variable with the same name exists, its value will be assigned to the instance parameter.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



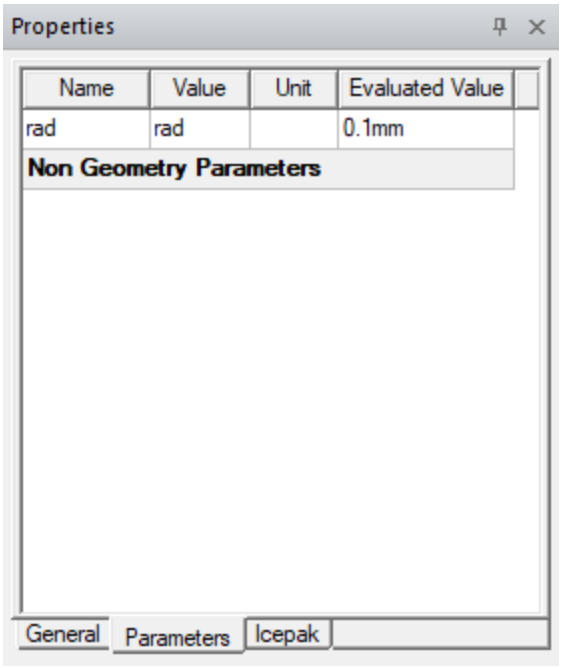
4. On the **Parameters** tab, note the component variable and definition are displayed. The **Value** can be edited on the parameter tab before creating the component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Retain the default values and click **OK** to create the fan.
6. In the **Properties** window, select the **Parameters** tab to view the instance parameter you created. Note that the design variable of the same name is assigned in the **Value** column. You can use design and project variables in optimetrics, animations, plots, etc.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

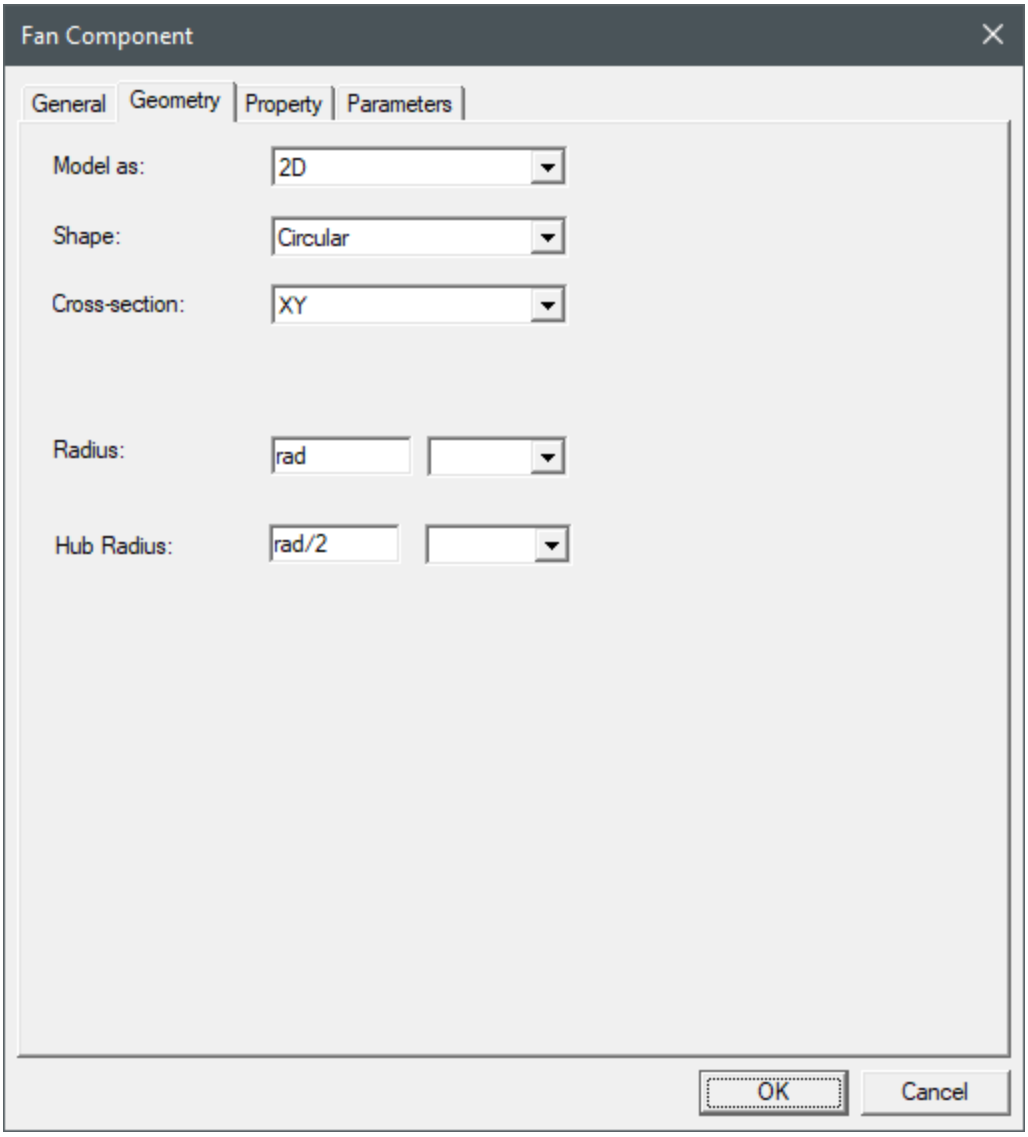


> **Note:** To edit the value of a design variable, select **Design Properties** from the **Icepak** menu. To edit the value of a project variable, select **Project Variables** from the **Project** menu.

Using an Instance Parameter in an Expression

You can use expressions to control multiple properties using a single instance parameter. For example, if you create an instance parameter for a fan **Radius** (*rad*) and the **Hub Radius** is half the **Radius**, you can enter *rad/2* as in the following image.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining Variables for Component Parameters

In Icepak, certain 3D native component parameters can be defined as variables to be used in parametric sweeps.

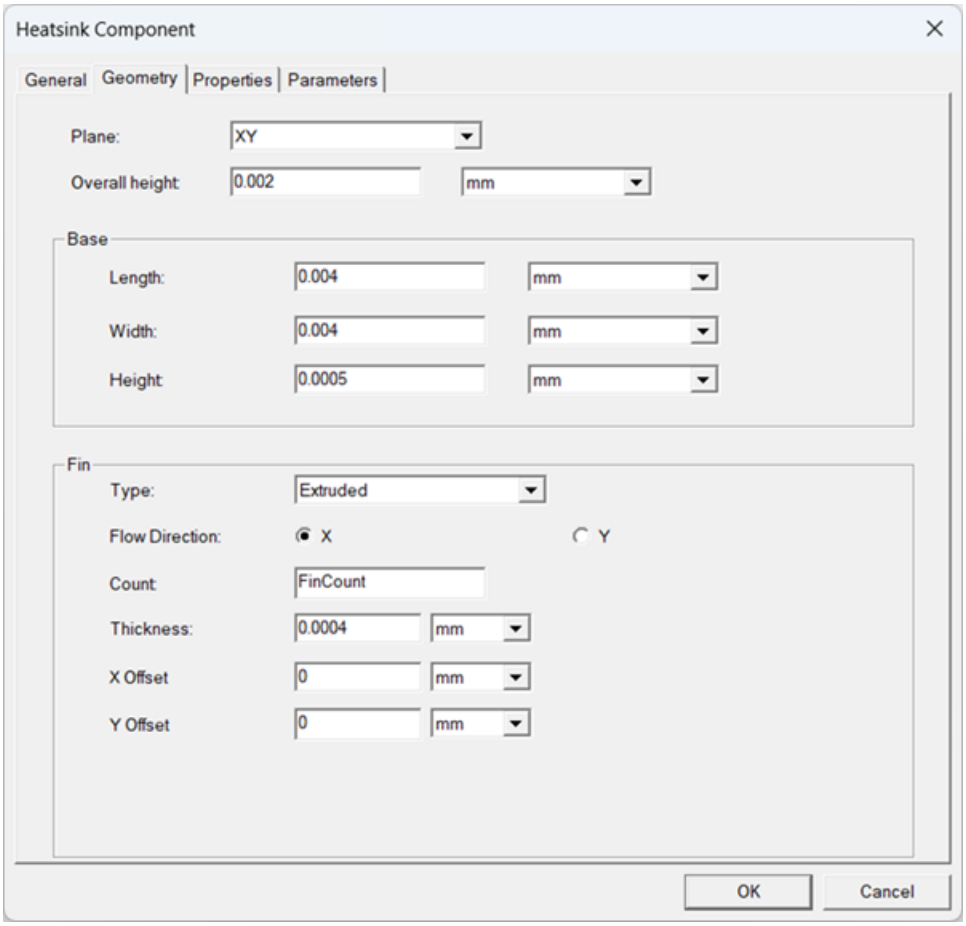
- Heatsink fin **Count**
- Package solderballs/solderbumps (beta)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create a Heatsink with a Fin Count Variable

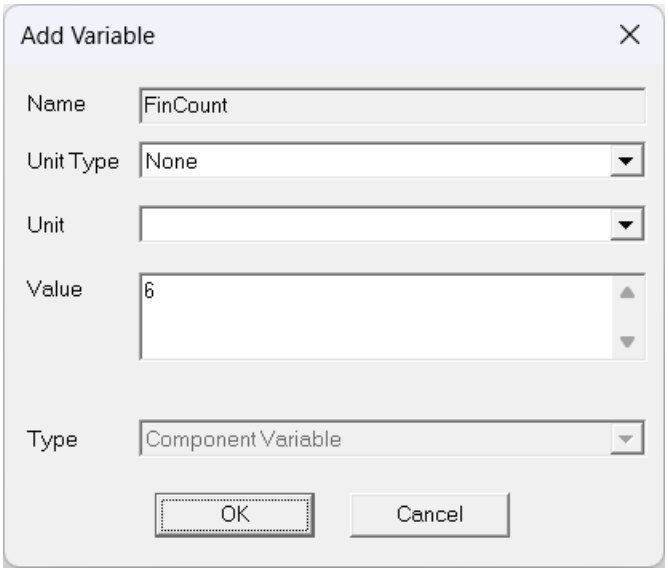
The following is a general procedure to define a variable for a component parameter. In this example, a heatsink component is created with a variable assigned to **Count**.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **Heatsink**.
2. On the **Heatsink Component Geometry** tab under **Fin**, enter *FinCount* as a variable for **Count**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

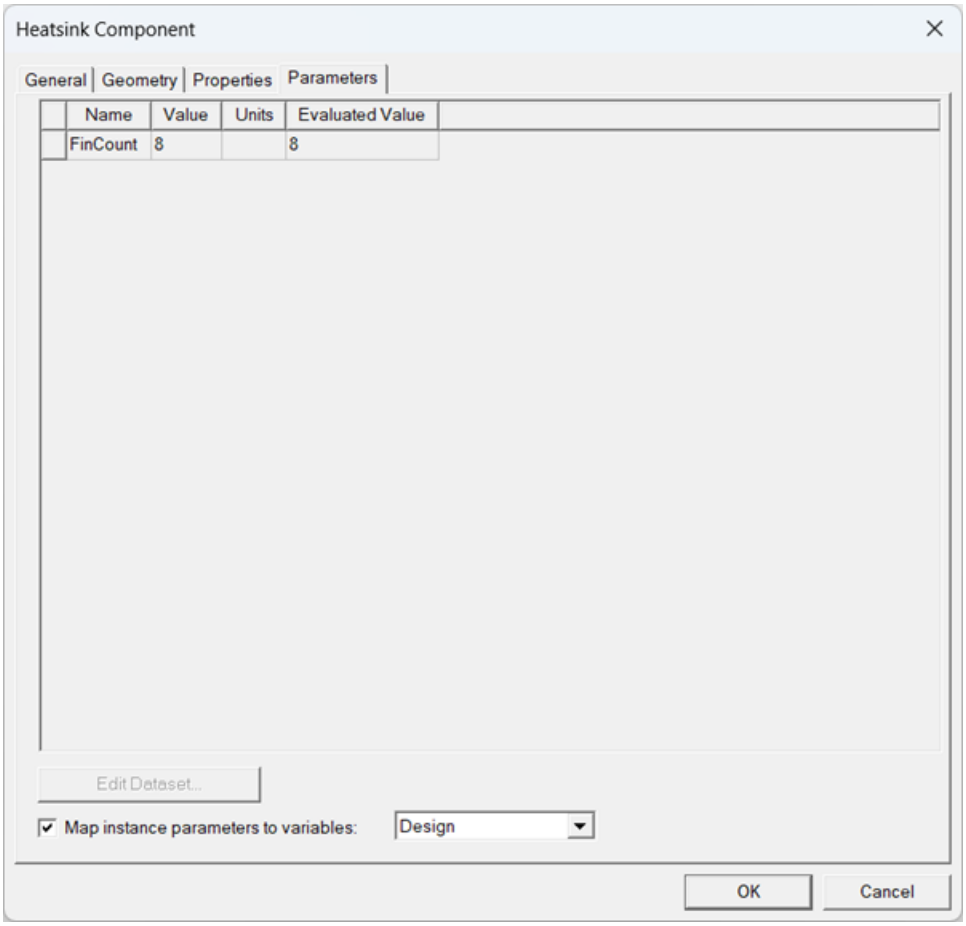
3. In the **Add Variable** dialog box, enter *6* for the **Value** and click **OK**.



4. On the **Parameters** tab, note the parameter and definition are displayed. The **Value** can be edited on the **Parameters** tab before creating the component.

> **Note:**
>
> After the initial insertion of a native component, the **Map instance parameters to variables** option is not available if you edit the component in its dialog box. You can enter a design or project variable as the **Value** to map an instance parameter to a design or project variable manually.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



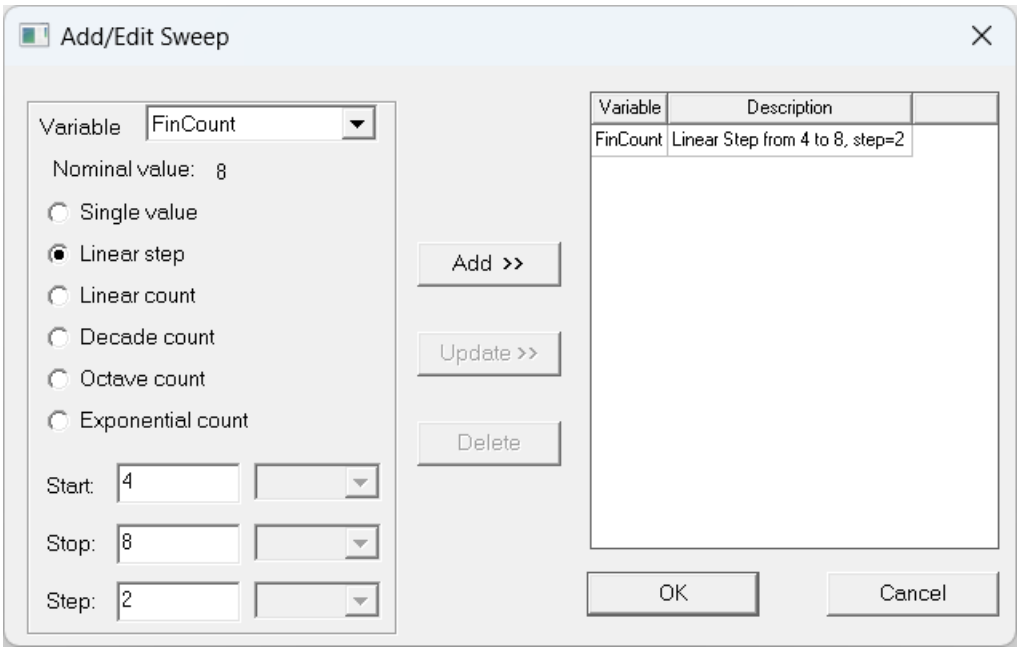
5. Click **OK** to create the heatsink.

Create a Parametric Sweep

The following is a general procedure to create a parametric sweep with a variable for a component parameter. This example contains a variable to evaluate multiple variations featuring a different number of heatsink fins.

1. In the **Project Manager**, right-click **Optimetrics** and select **Add** > **Parametric**.
2. On the **Setup Sweep Analysis** dialog **Sweep Definitions** tab, click **Add**.
3. In the **Add/Edit Sweep** dialog, retain the selection of **Linear step** enter the following to create a parametric sweep featuring variations with 4, 6, and 8 fins:
    a. **Start**: 4
    b. **Stop**: 8
    c. **Step**: 2

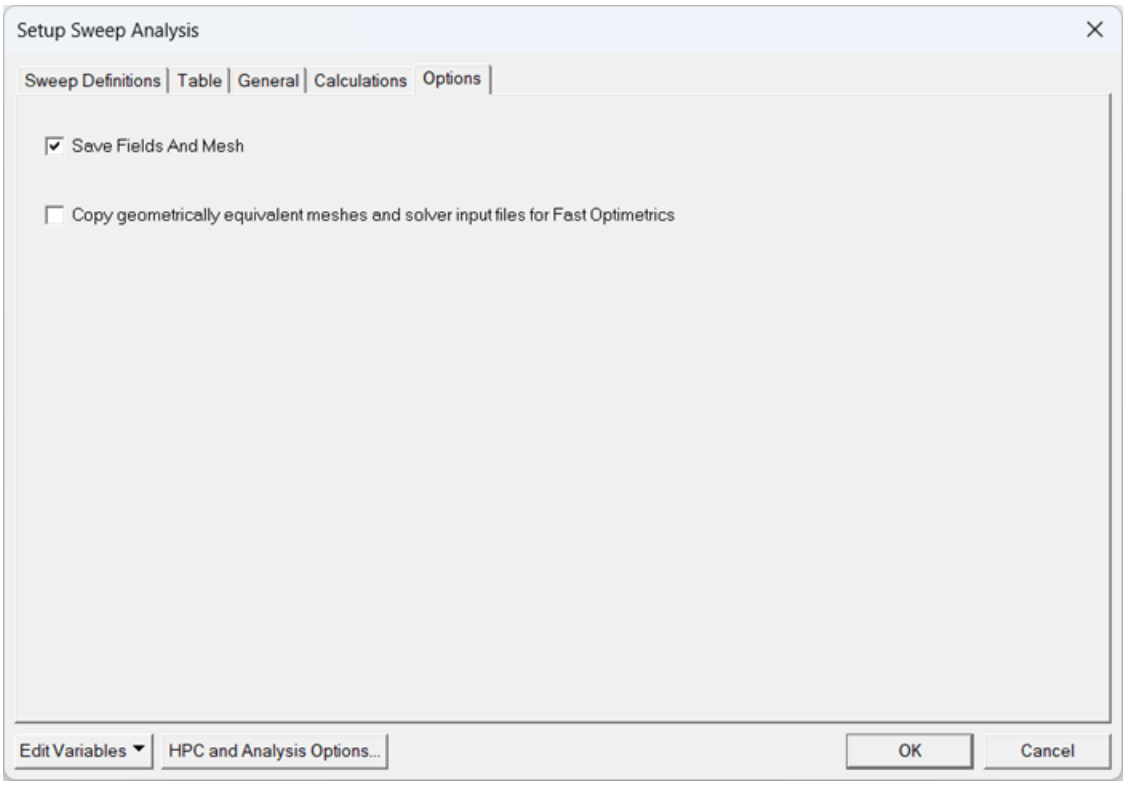
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Add**.
5. On the **Options** tab, enable **Save Fields and Mesh**.

> **Important:**
>
> You must enable **Save Fields and Mesh** to save results for and post-process all variations after solving the parametric sweep.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



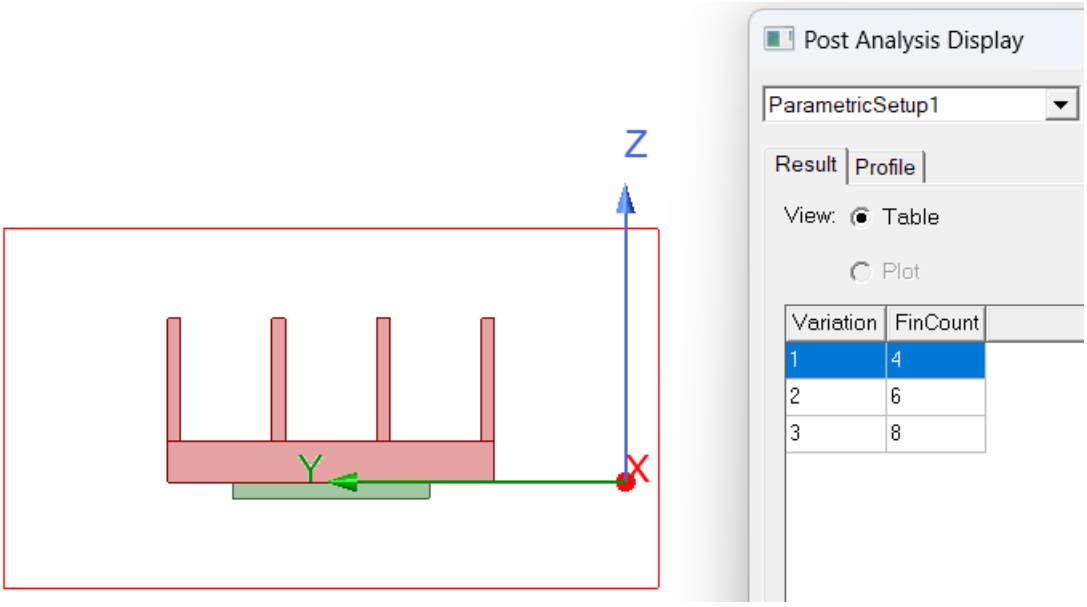
6. Click **OK** to save the parametric sweep.

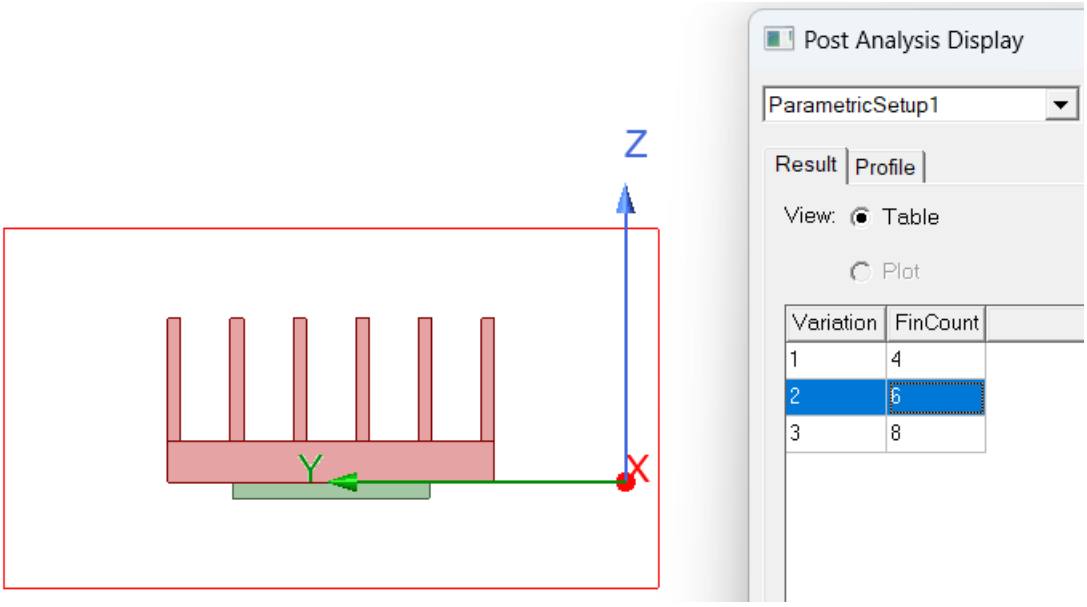
Viewing Design Variations

After the parametric sweep is complete, you can post-process the design. Any field overlays you create can be viewed in each design variation to compare the results.

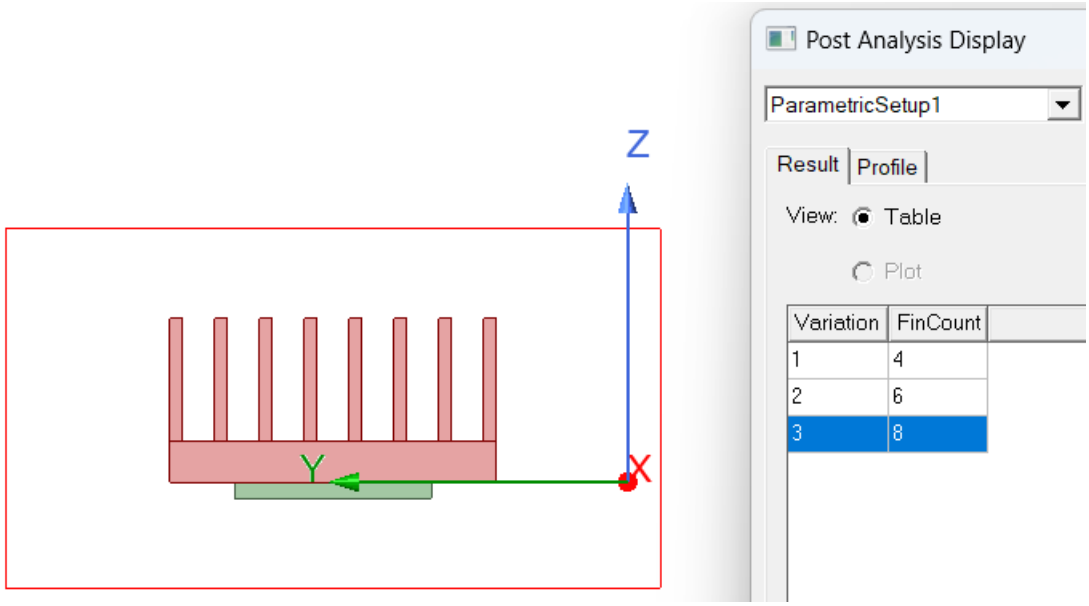
1. In the **Project Manager**, right-click **Optimetrics** and select **View Analysis Result**.
2. On the **Post Analysis Display** dialog **Result** tab, select a variation and click **Apply** to view the different configurations of the component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

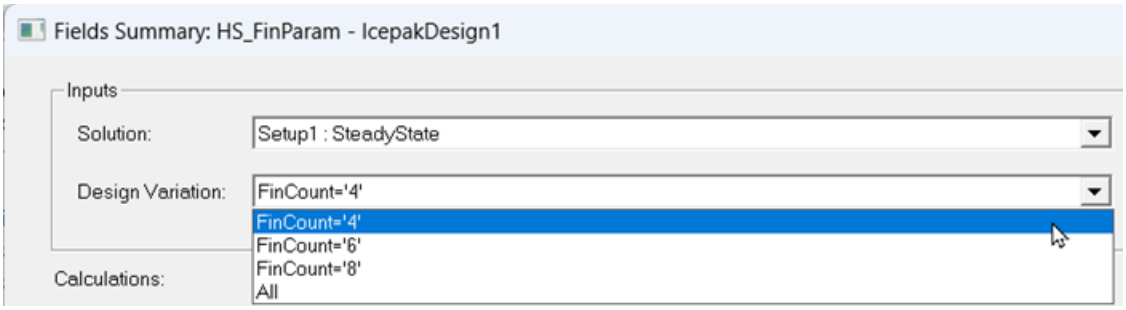




Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Fields Summary** also contains results data for all solved variations.

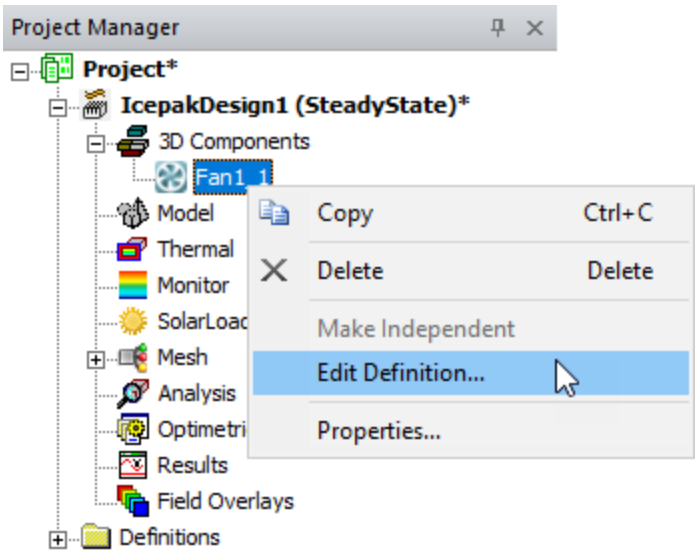


Editing Component Definitions

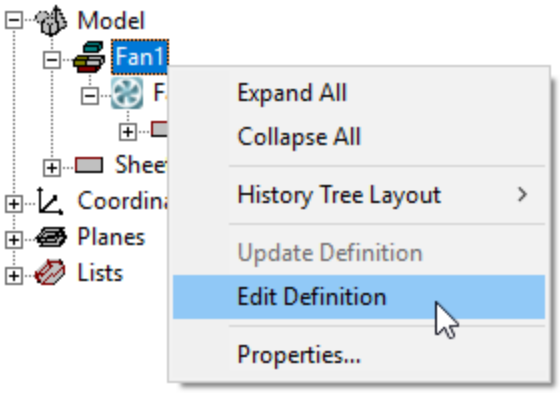
After creating an Icepak component, you can edit the component definition from the Project Manager or the history tree.

- In the **Project Manager**, expand **3D Components**, right-click on the component, and select **Edit Definition**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

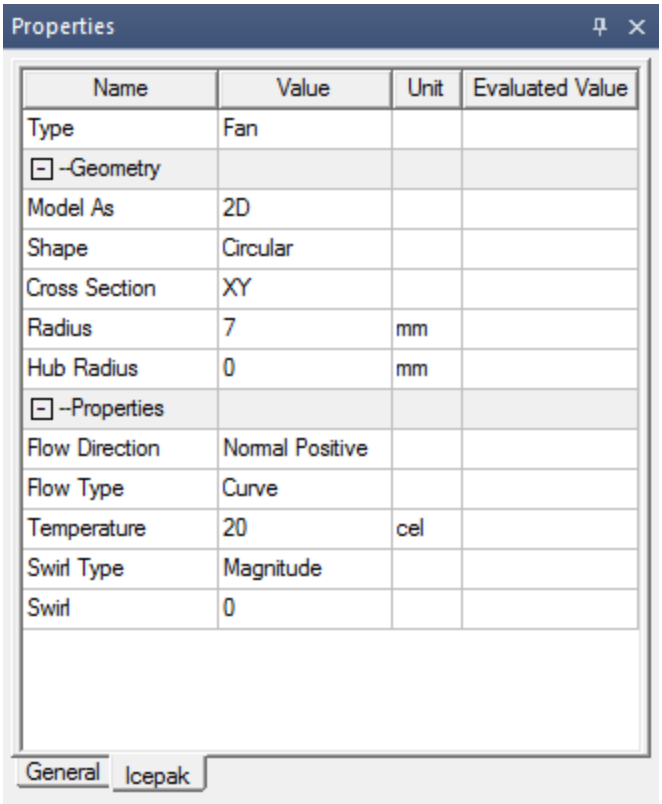


- In the history tree, right-click the component you want to edit and select **Edit Definiton**.



Alternatively, you can edit a component's definition by selecting it in the history tree and changing settings on the in the **Properties** window on the **Icepak** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Not all component settings are editable in the **Properties** window.

Inserting Component Instances

> **Note:** You can not insert an instance of a CTM native component. To create multiple CTMs, you must create multiple CTM native components. See Creating CTMs for how to create a CTM.

After creating an Icepak component, you can use the existing component definition to insert one or more instances of the component in the **3D Modeler**.

1. In the **Project Manager**, right-click **3D Components**.
2. From the right-click menu, hover the cursor over **Insert Instance**. A list of the existing components in the model is displayed.
3. Select the component that you want to create another instance of. A new instance of the component is listed in the **Project Manager**, and its corresponding geometry is listed in the history tree and displayed in the **3D Modeler**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You can create multiple instances of a PCB component and move them using translation and rotation functions. Each instance uses the ECAD data link from the source design. See Inserting Component Instances and Moving and Rotating Objects for more information.

Making Component Instances Independent

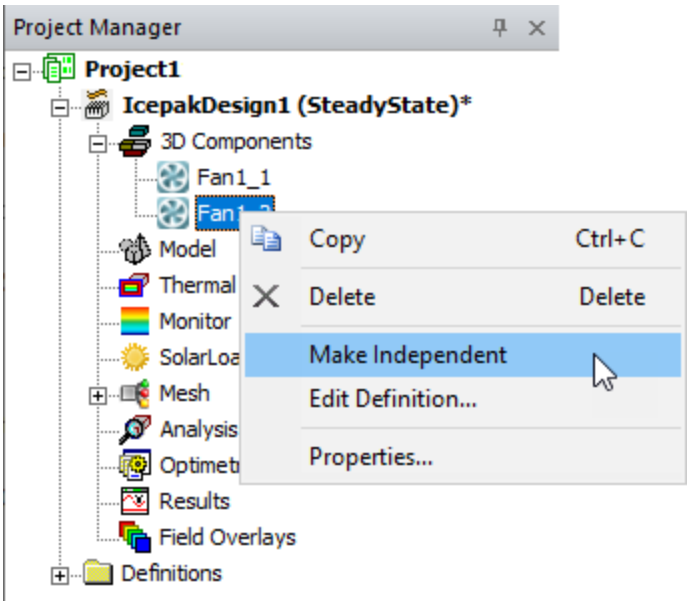
When inserting an instance of a component, it is listed under the original component and assigned the same definition. To create a unique component from a component instance, use the **Make Independent** option in the Project Manager or History Tree. In Icepak designs, **Make Independent** is only available for Icepak native components, (e.g., a Fan, Heatsink, or PCB). **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.

Project Manager

1. In the **Project Manager** under **3D Components**, right-click on a component instance.



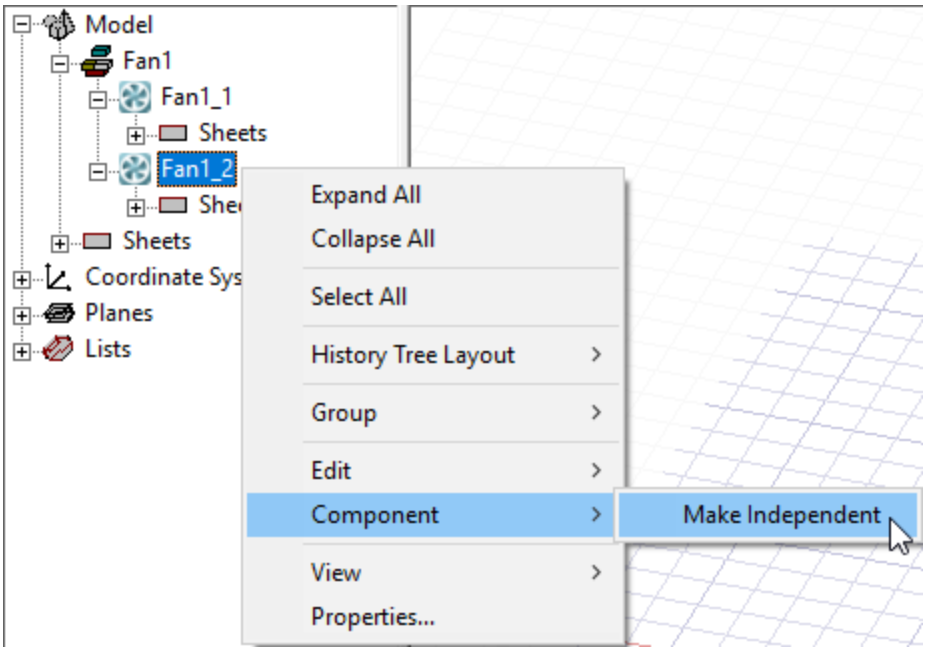
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

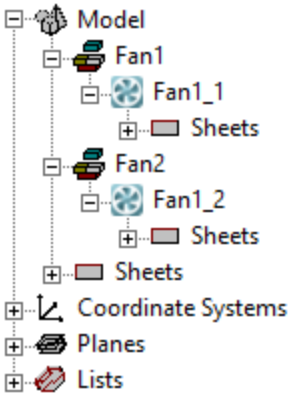
1. In the history tree, right-click on a component instance.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 - Layout Components [Beta]

In Icepak designs, layout components are a beta feature supported for steady-state and transient solution types.

> **Note:**
>
> For information on enabling beta options, see General Options: Desktop Configuration.

Layout components allow you to import HFSS 3D Layout designs into an Icepak design for analysis. Geometry from the HFSS 3D Layout design is accurately modeled in Icepak designs. For example, a printed circuit board (PCB) with multiple layers is modeled with separate geometry for each layer. See Adding a Layout Component for more information.

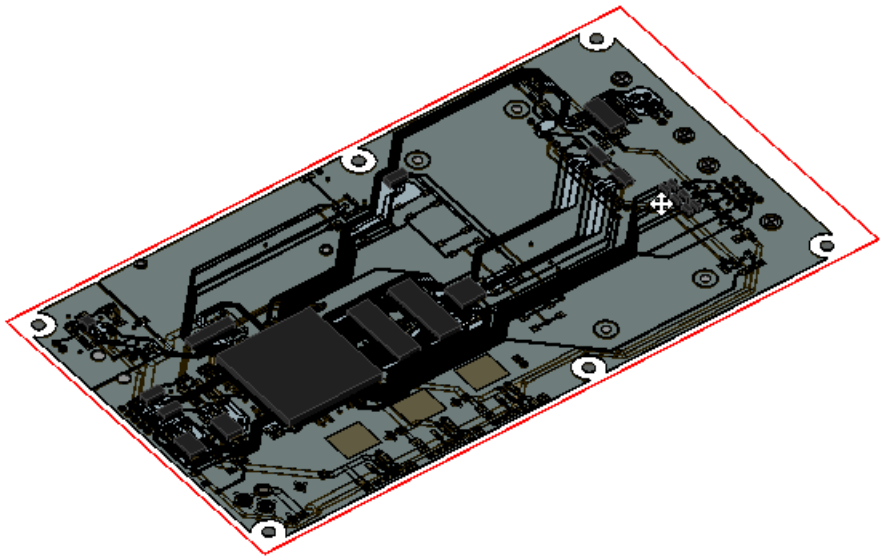
> **Note:**
>
> During the Icepak analysis, the solver uses the layout component's material assignments from the HFSS 3D Layout design. In the Icepak design's History Tree, component geometry is listed under placeholder materials.

Layout components allow you to import power maps from a linked HFSS 3D layout DCIR or AC design. Layout components also allow you to visualize traces on a PCB in a simulation model. The PCB in the image below is a layout component imported from an HFSS 3D Layout DC IR or AC analysis into an Icepak design. See Trace Mapping Using a Layout Component for more information.

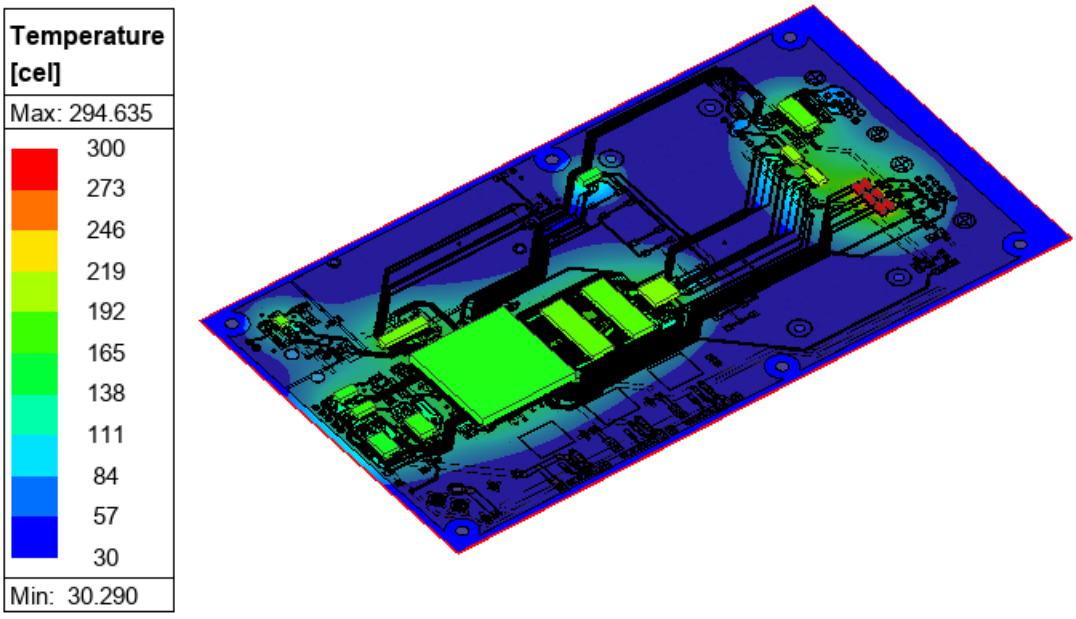
Example

The following figure is a PCB layout component in an Icepak design. The solution type is Steady-State. 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 2026 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:



Related Topics

Adding a Layout Component

Inserting a Layout Component Instance

Editing a Layout Component Definition

Replacing or Editing Layout Components

Layout Component Properties

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

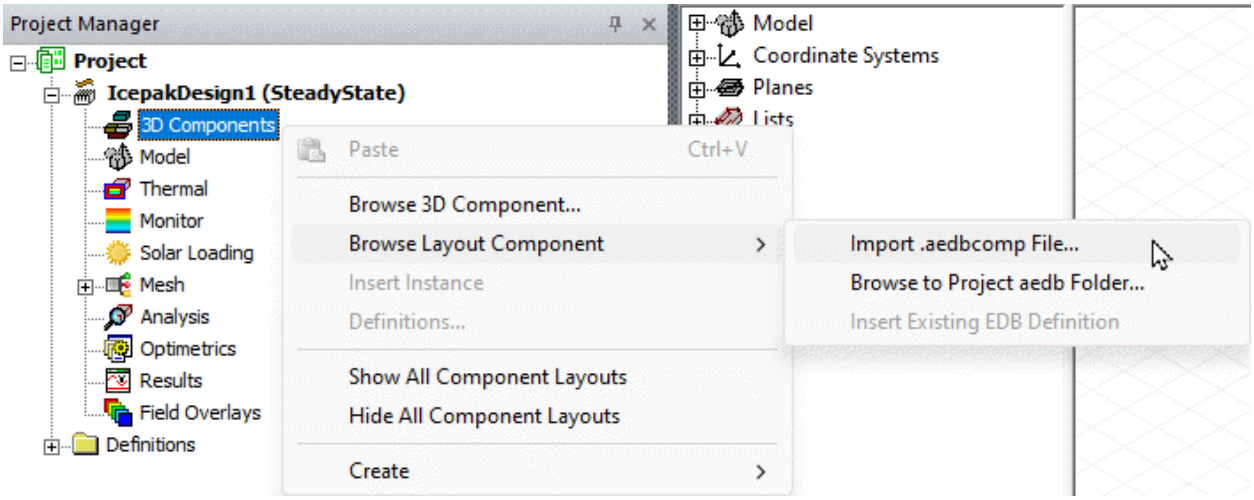
Trace Mapping Using a Layout Component

Adding a Layout Component

Layout components are a beta feature supported for steady-state and transient solution types.

> **Note:**
>
> For information on enabling beta options, see General Options: Desktop Configuration.

1. In the **Project Manager**, right-click **3D Components** and access the **Browse Layout Component** flyout menu:

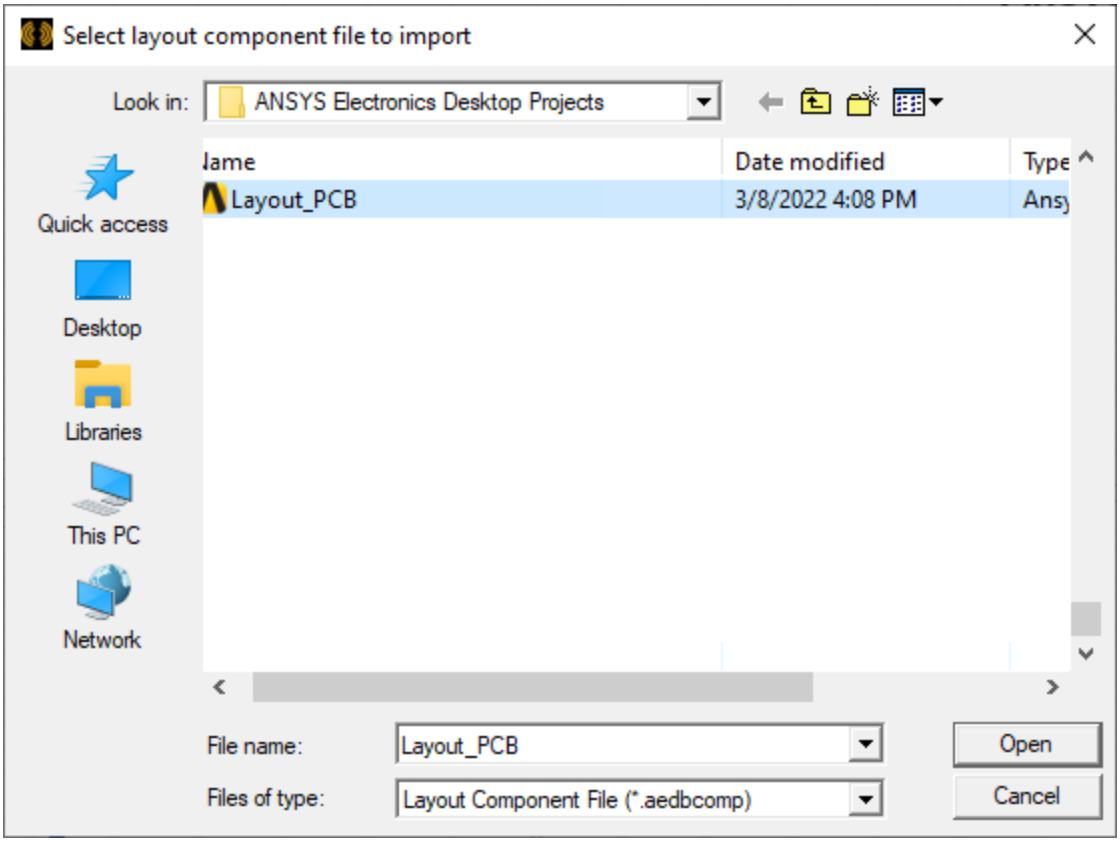


   Then, choose *one* of the following three commands and follow the associated instructions:

- **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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

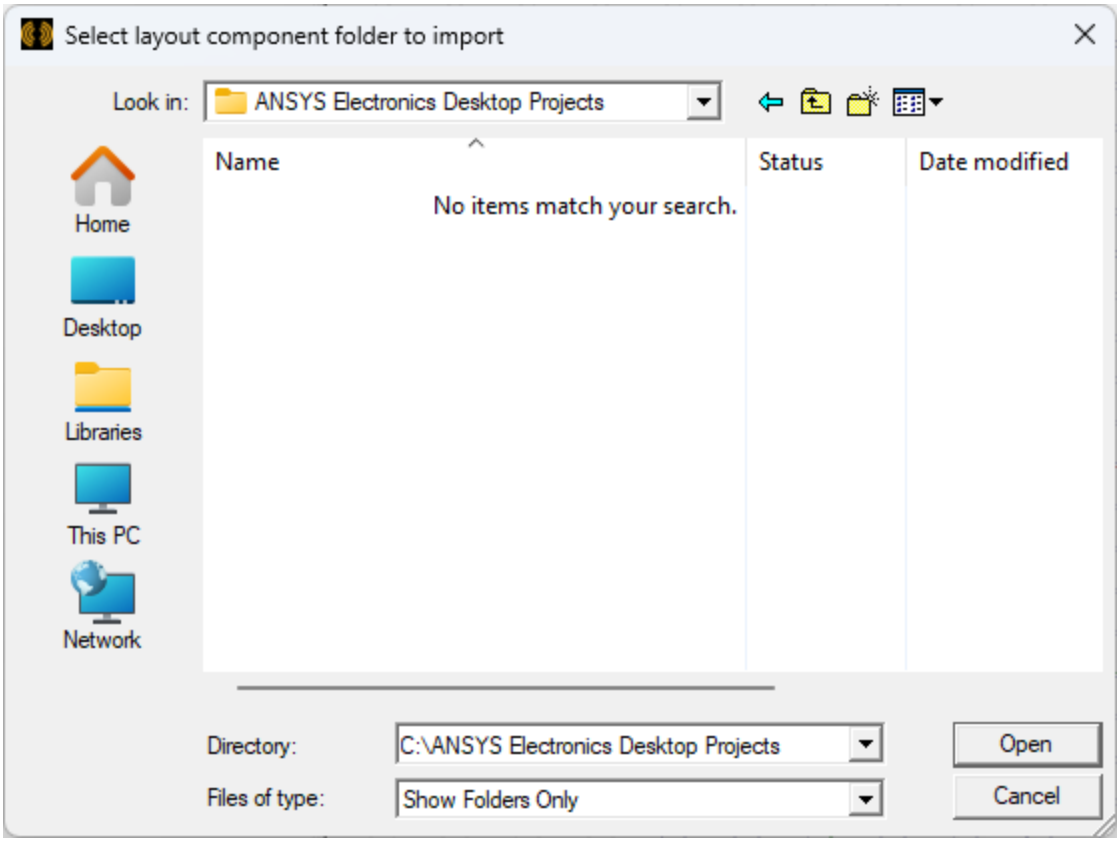


a. Browse to the location of the desired *.aedbcomp file.
b. Select the file.
c. Click **Open** to complete the component source file selection.

- **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.

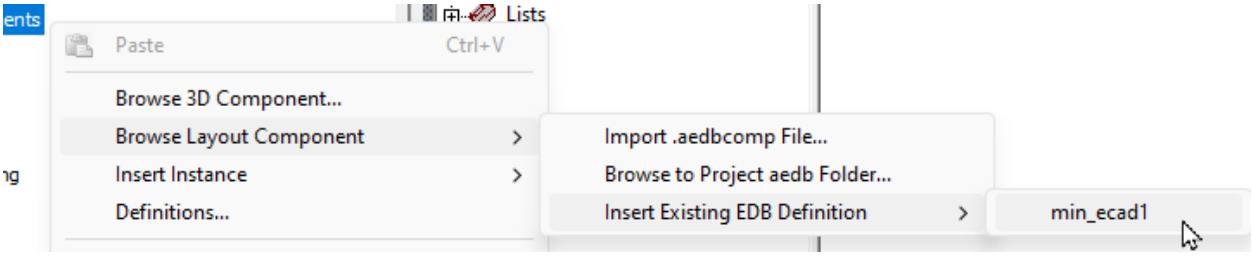
  A **Select layout component folder to import** dialog box appears:

Ansys Electromagnetics Suite 2026 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** shows a list of existing EDB definitions available within the project.



Selecting an .aedbcomp file, .aedb folder, or an existing EDB definition opens the **Insert Layout Component (Beta)** dialog.

The **Insert Layout Component** dialog **General** tab appears:

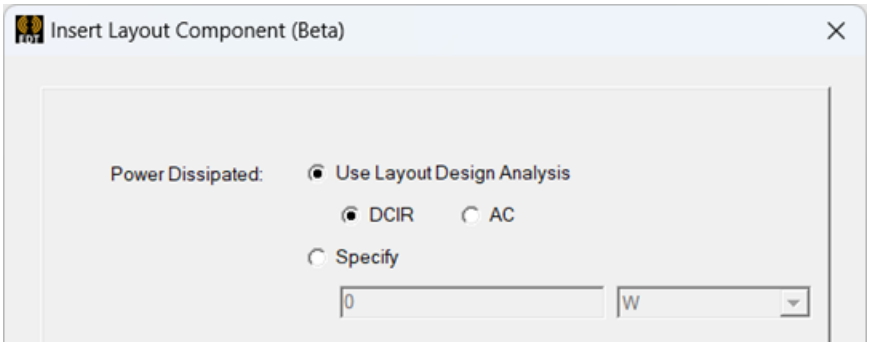
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **General** tab page allows you to specify a target coordinate system. From the **Target Coordinate System** drop-down list, select the coordinate system on which to place the layout component. Relative coordinate systems defined in the target design appear in the drop-down list.

2. On the **Property** tab, select an option to define the **Power Dissipated**. Power dissipated specifies the total power dissipated by the layout component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Use Layout Design Analysis** specifies the power based on the EM loss results in the linked HFSS 3D Layout design.
  - Select **DCIR** or **AC** if the source HFSS 3D Layout design includes a DCIR or AC analysis setup, and you wish to use the results of that solution as a true, nonuniform heat source for the layout component in the Icepak solution. **DCIR** imports power maps from an SIwave DCIR (direct current, internal resistance) solution setup. See Computing DC IR Simulation in HFSS 3D Layout for more information. **AC** imports power maps from an AC Simulation in HFSS 3D Layout.

    Note:

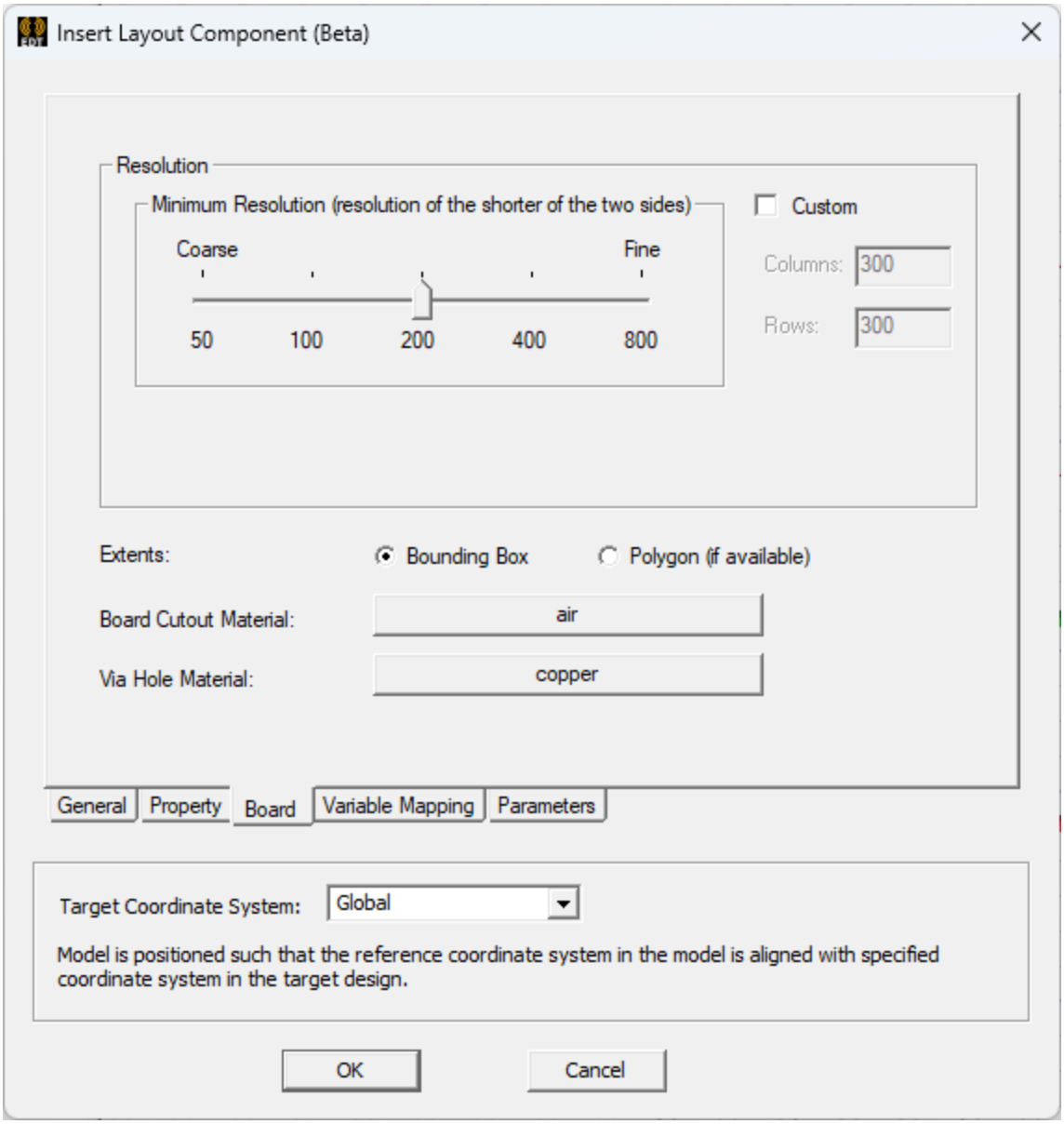
    When a layout component is added to a Icepak design, results from any analysis setups in the source design are *not* imported. The source layout analysis will automatically be run first to obtain the power dissipation values to apply in the Icepak solution. Then the Icepak design will be solved.

    When adding an SIwave DCIR solution setup in the source HFSS 3D Layout design, you do not have to select the option, **Export power distribution for use in Ansys Icepak and IcepakFEA**. This option appears in the *Compute DC Current and Voltage Distribution* dialog box. The process of linking to the source layout automatically handles all required options and parameters.

  - **Specify** defines a uniformly dissipated total power.

3. On the **Board** tab, define the metal fraction, geometry, and material properties of the layout component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 component, whether vertical or horizontal.

     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 component. 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Columns refers to the number of divisions along the horizontal (X direction), and **Rows** refers to the vertical (Y direction) divisions, regardless of which dimension is larger.

See the *"Important"* guidelines in the *Trace Mapping* topic for resolution recommendations.

b. Choose the desired **Extents** option:
   - **Bounding Box:** The bounding box of the source layout design
   - **Polygon (if available):** The largest outline layer polygon of the source layout design

c. Optionally, choose the desired **Board Cutout Material** and **Via Hole Material**.

> **Note:**
>
> **Board Cutout Material** specifies the material assigned to the board cutout geometry, if it exists. Board cutout geometry exists if **Polygon** is selected for **Extents**, and that polygon contains void polygon(s). The cutout geometries are created from those void polygon(s). The default board cutout material is that of the global mesh region if present in the model. Otherwise, the default board cutout material is set to air-solid.
>
> The default material for via holes is copper.

4. What appears on the **Variable Mapping** tab depends on whether variables are defined in the source layout. Choose one of the following two actions:
   - If there are no applicable variables in the source layout, the **Parameters** tab has no variables listed.

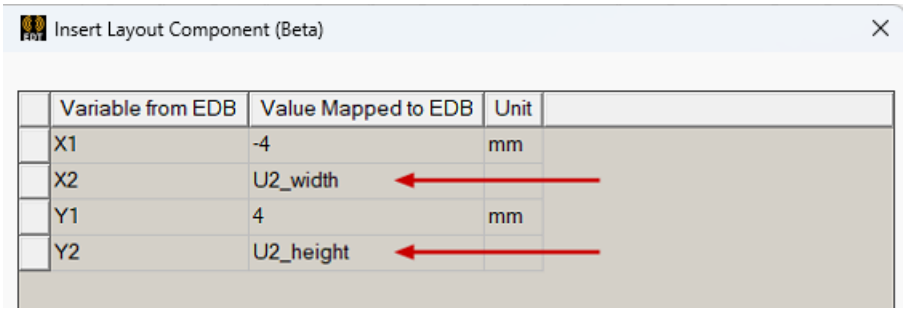
     Skip to step 5.

   - When applicable variables are present in the source layout, the **Variable Mapping** tab, do one of the following:
     a. Optionally, you can substitute local variable names for the numeric values in the **Values Mapped to EDB** column. This 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.

        If you do specify a previously undefined local variable name, the **Add Variable** dialog box will appear when you press **Enter** so that you can define its unit type, units, and initial value in-place.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the following example, two of the four variables in the source ECAD database (X2 and Y2) have had local variables substituted for the original numeric values:



> **Note:**
>
> If you wish to specify a project variable name, you do *not* have to prefix the variable name with a dollar sign ($). On the next tab, you can choose whether to create Design or Project variables, and the name will be automatically prefixed with a dollar sign for the latter type.

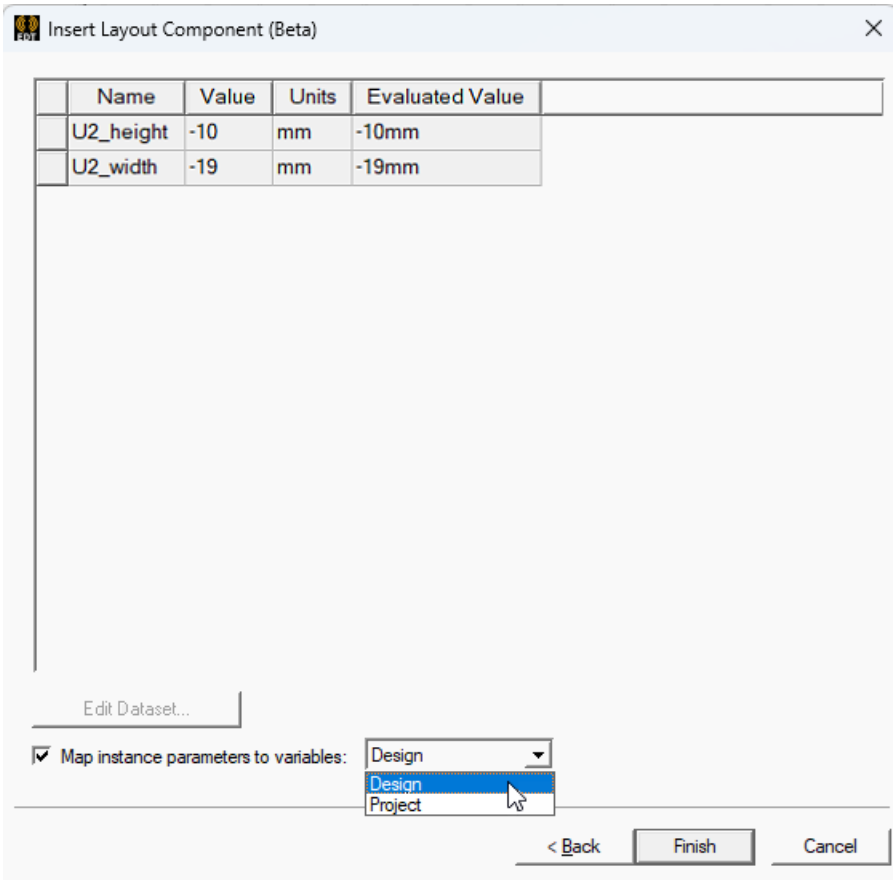
b. On the **Parameters** tab, if you chose not to define local variables on the **Variable Mapping** tab, or if the source design contains no variables, none will be listed on the **Parameters** tab.

   Skip to step 5.

c. Otherwise, when you have defined local variables on the **Variable Mapping** tab, they will be listed on the **Parameters** tab along with their values and units.

   The following figure shows the parameters with two local variables substituted for numeric values.

Ansys Electromagnetics Suite 2026 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 Icepak 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 Icepak design or AEDT project variables, select this option and choose between the two types in the next step.

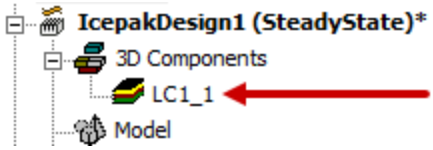
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. If **Map instance parameters to variables"** 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.

5. Click **OK**.

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:



6. 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.

Two-Way Coupling

Two-way coupling can be added to Icepak designs with layout components linked to HFSS 3D Layout designs with AC or DC loss. See Adding Two-Way Coupling to an Icepak Design for more information. Loss data calculated by the source HFSS 3D Layout design is consumed by Icepak when calculating thermal data in the current coupling iteration. The thermal data is then consumed by the source design in the next coupling iteration's calculation. After a two-way coupling simulation is complete, view the coupling iteration results in the **Solution** dialog box **Profile** tab.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Task	Real Time	CPU Time	Memory	Information
Coupling Iteration 1				Solver Iteration: 1
Populate Solver Input	00:00:00	00:00:00	176 M	
Solver Initialization				Real Time: 00:00:11, CPU Time: 00:00:11, Memory: 3.18 G
PCB Loss Group 1				Surface Loss: 2.55 (Watt), Group 1: LC1_000_METAL1_BBox
PCB Loss Group 2				Surface Loss: 0.32746 (Watt), Group 2: LC1_002_METAL2_BBox
PCB Loss Group 3				Volume Loss: 2.6475 (Watt), Group 3: LC1_001_DIELECTRIC1_BBox
				Total Mapped PCB Loss: 5.5249 (Watt)
Solve	00:00:21	00:00:21	3.25 G	
Coupling Iteration 2				Solver Iteration: 75
Populate Solver Input	00:00:00	00:00:00	177 M	
Solver Initialization				Real Time: 00:00:10, CPU Time: 00:00:10, Memory: 3.19 G
PCB Loss Group 1				Surface Loss: 2.8525 (Watt), Group 1: LC1_000_METAL1_BBox
PCB Loss Group 2				Surface Loss: 0.36222 (Watt), Group 2: LC1_002_METAL2_BBox
PCB Loss Group 3				Volume Loss: 2.6162 (Watt), Group 3: LC1_001_DIELECTRIC1_BBox
				Total Mapped PCB Loss: 5.831 (Watt)
Solve	00:00:02	00:00:02	3.24 G	
Coupling Iteration 3				Solver Iteration: 76
Populate Solver Input	00:00:00	00:00:00	177 M	
Solver Initialization				Real Time: 00:00:10, CPU Time: 00:00:10, Memory: 3.2 G
PCB Loss Group 1				Surface Loss: 2.8681 (Watt), Group 1: LC1_000_METAL1_BBox
PCB Loss Group 2				Surface Loss: 0.36379 (Watt), Group 2: LC1_002_METAL2_BBox
PCB Loss Group 3				Volume Loss: 2.6145 (Watt), Group 3: LC1_001_DIELECTRIC1_BBox
				Total Mapped PCB Loss: 5.8465 (Watt)

Note:

You can judge whether you've run a sufficient number of coupling iterations by comparing the total loss magnitudes between the last two iterations (highlighted in the preceding 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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}***
- 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. However, be aware that changing the definition in this manner affects all instances of the component. 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:

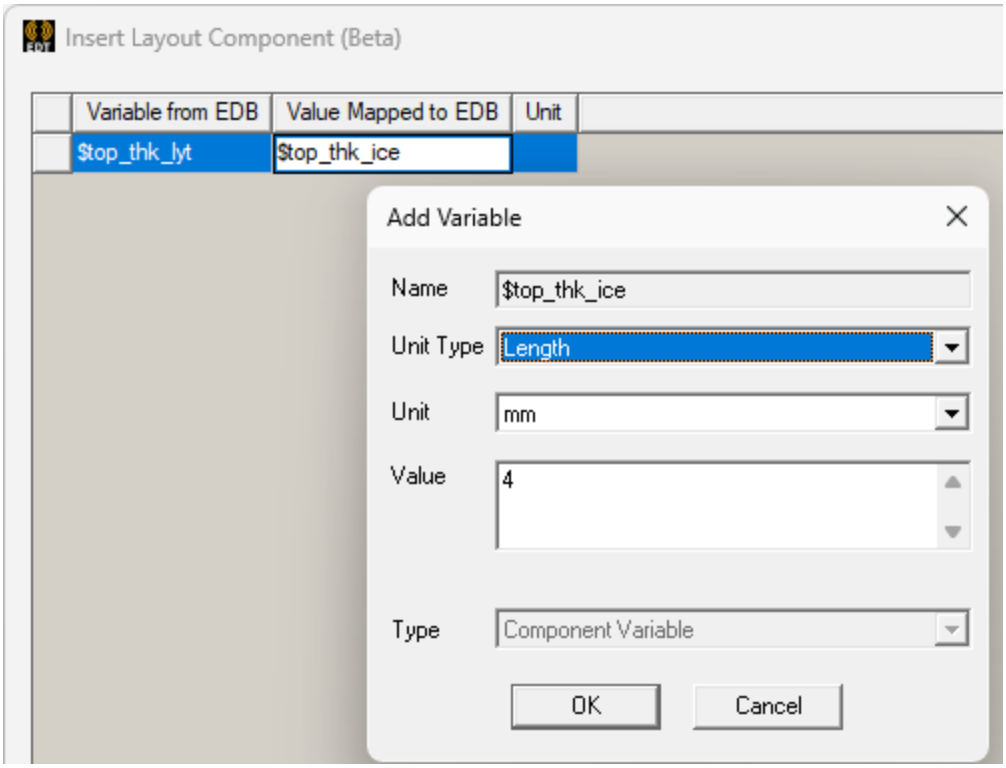
  While adding the layout component to your Icepak design, do the following when you reach the **Variables Mapping** tab of the **Insert Layout Component** dialog box:

  1. In the **Value Mapped to EDB** column, replace one or more numerical values with the name of a local variable.
  2. Press **Enter**.

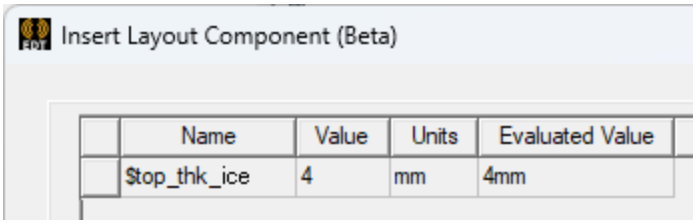
     If the name you specified has not already been defined, the **Add Variable** dialog box will appear.

  3. Select the **Unit Type** and **Unit** from the drop-down menus.
  4. Optionally, change the prepopulated **Value**, which initially equals the value assigned to the associated variable in the source layout.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.
6. Repeat steps 1–4 for any other variables you wish to map between the target and source designs.
7. On the **Parameters** tab of the **Insert Layout Component** dialog box appears, it lists the Icepak design and project variables and the current numerical value of each:



8. 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.

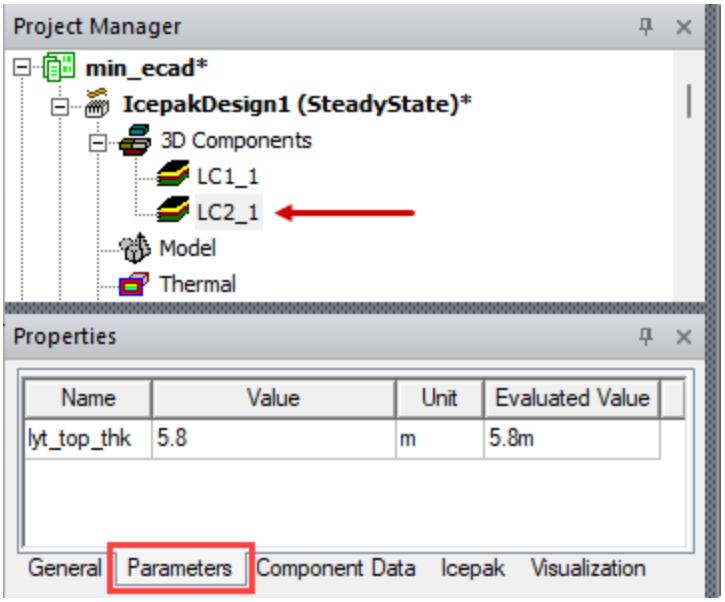
   Regardless of whether you choose to map parameters to local variables or not, each parameter will be exposed as a property of the layout component instance, and the value can be manipulated independently for each instance.

9. Click **OK** to complete the component definition.

- Inserting two or more instances of the layout component:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 so that they don't overlap the first one. 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 *"LC2_1"*).
4. In the **Parameters** tab of the docked **Properties** window, change the **Value** of one or more parameters and press **Enter**

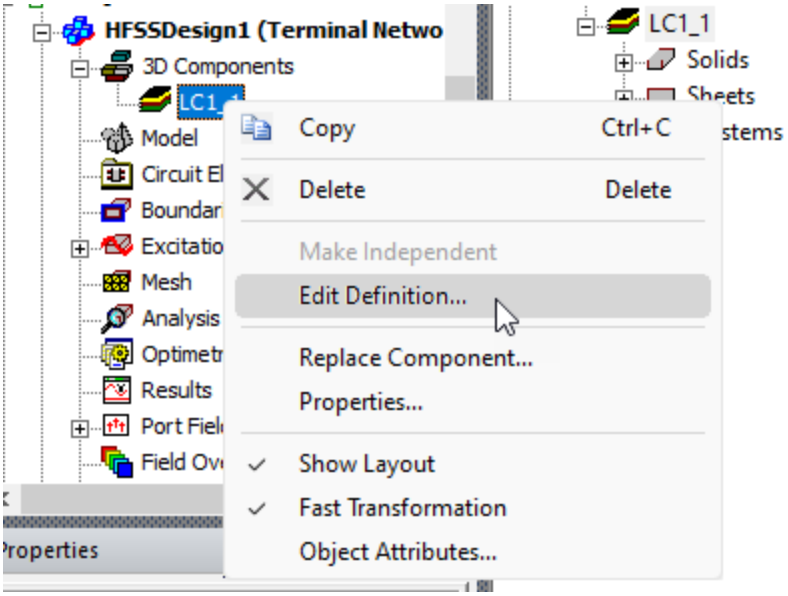


The geometry changes should be immediately visible in the **3D Modeler** window. Position the viewpoint and zoom in/out as required to clearly compare the geometry for each instance.

Editing a Layout Component Definition

All layout components in Icepak designs have a command in the right-click menu called **Edit Definition**.

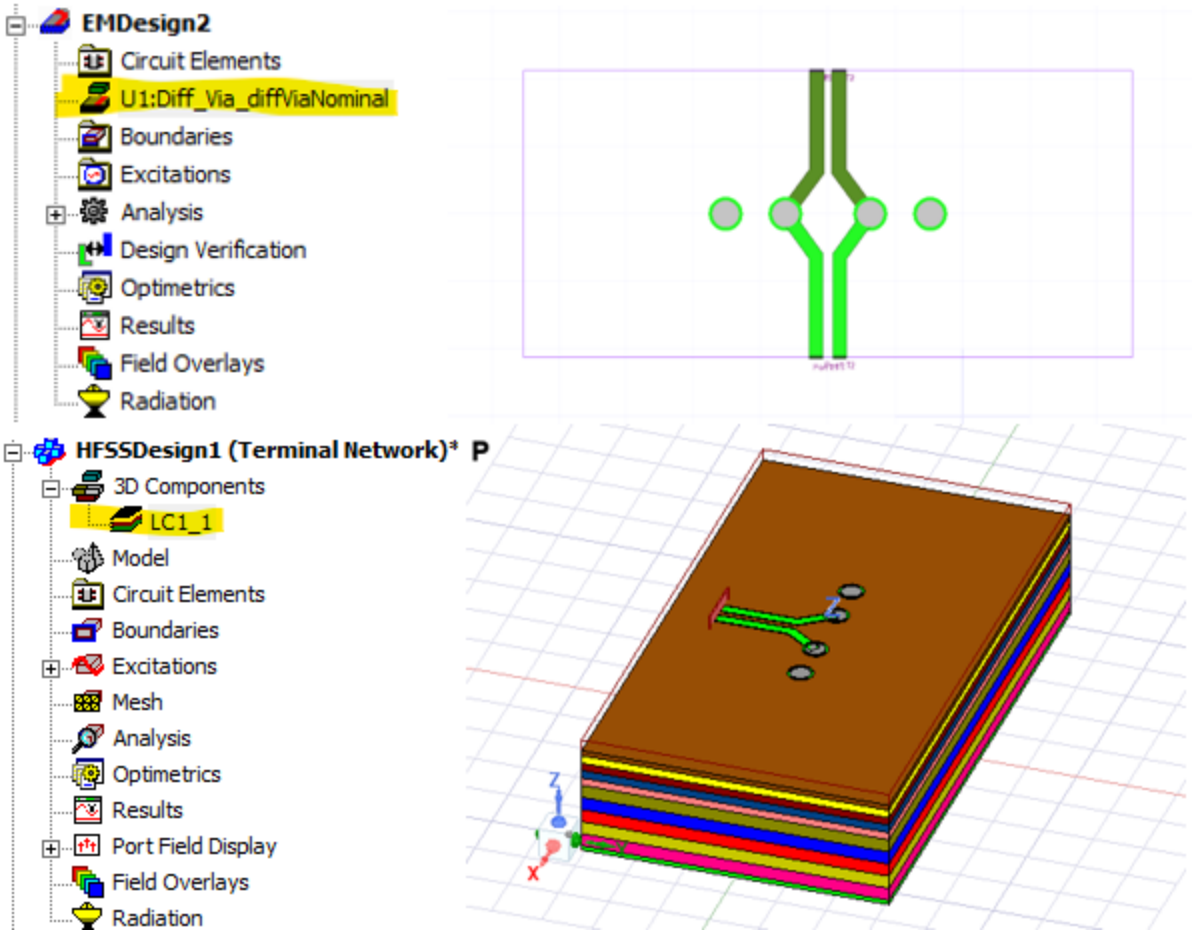
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



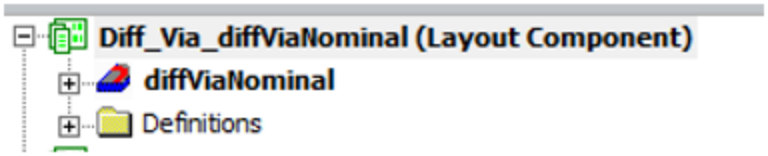
Once clicked, it opens the design containing the EDB definition of the layout component in a separate project to allow for any editing. Once a layout component is edited, either through a parent 3D layout design or through this command, all designs of different types that use this component update it 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”.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

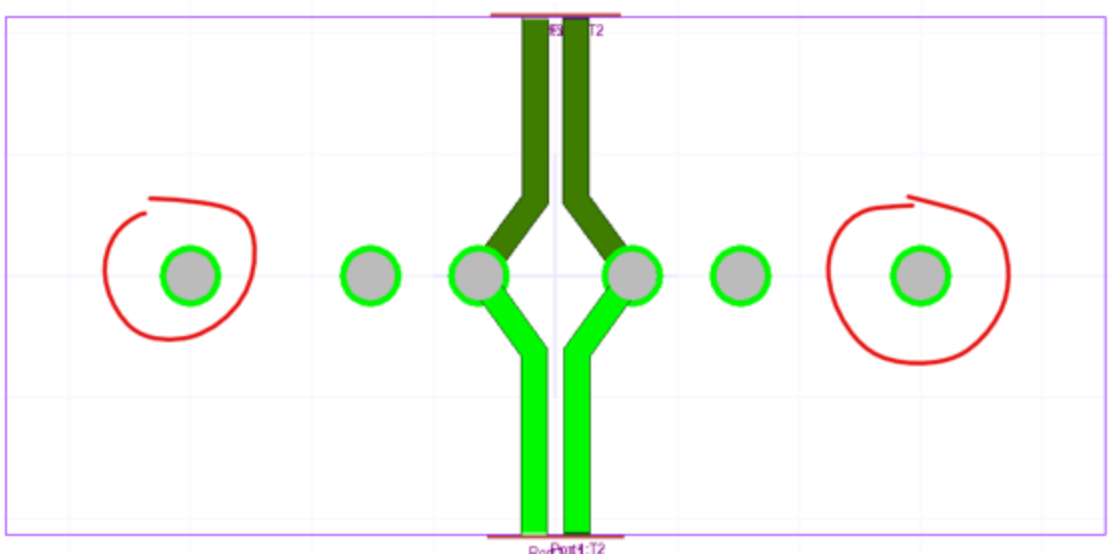


In addition to editing the source layout design using the **Edit Definition** command, you can edit the layout component's local settings, or replace it with a different component, using the **Replace Component** command.



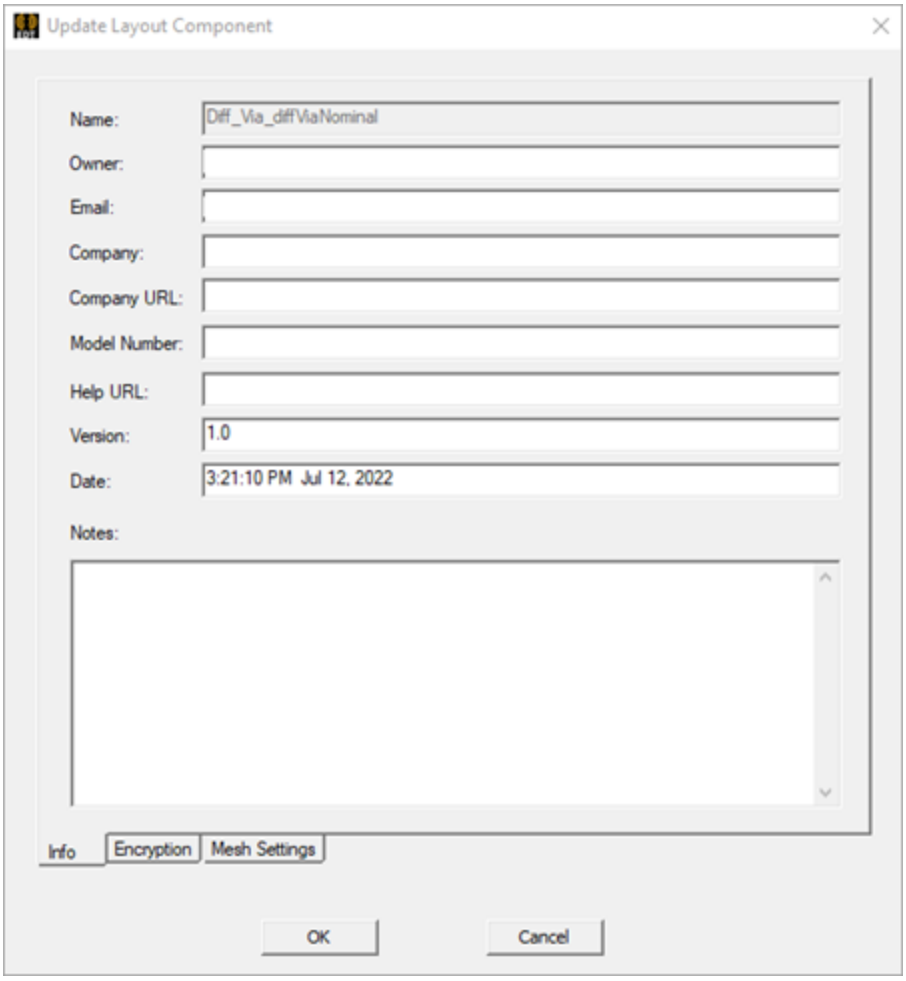
In the source design, you can edit the layout component however you want. This example adds two more vias on the left and right side as shown in the figure below.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



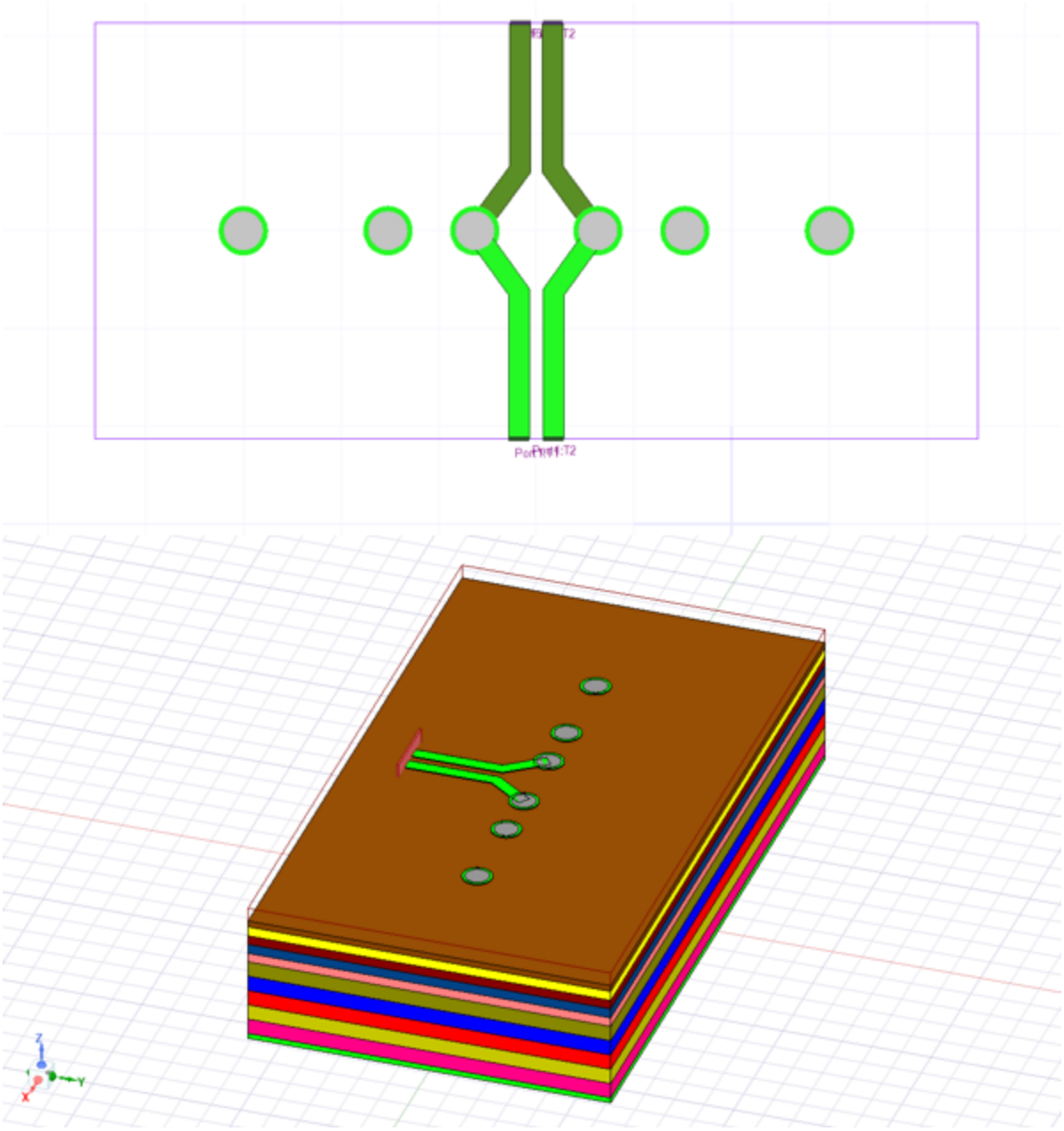
After editing, you can save the source design, and a dialog box opens for you to confirm the changes.

Ansys Electromagnetics Suite 2026 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 Icepak design containing this layout component update to reflect the changes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

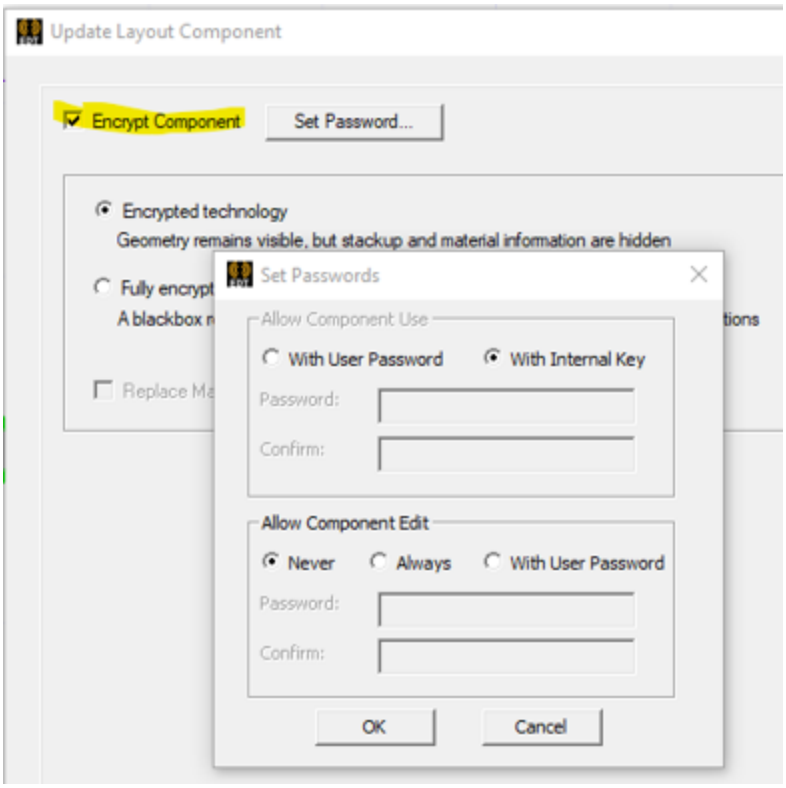


Note:

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 to limit its use and/or limit its editing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Warning:

With the Ansys Electronics Desktop 26r1 release, only HFSS 3D Layout and HFSS support encrypted layout components.

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. And after editing, all encrypted layout components are deleted from any target designs that do not supported encrypted layout components. A warning message appears for each deleted layout component.

Related Topics

Replacing or Editing Layout Components

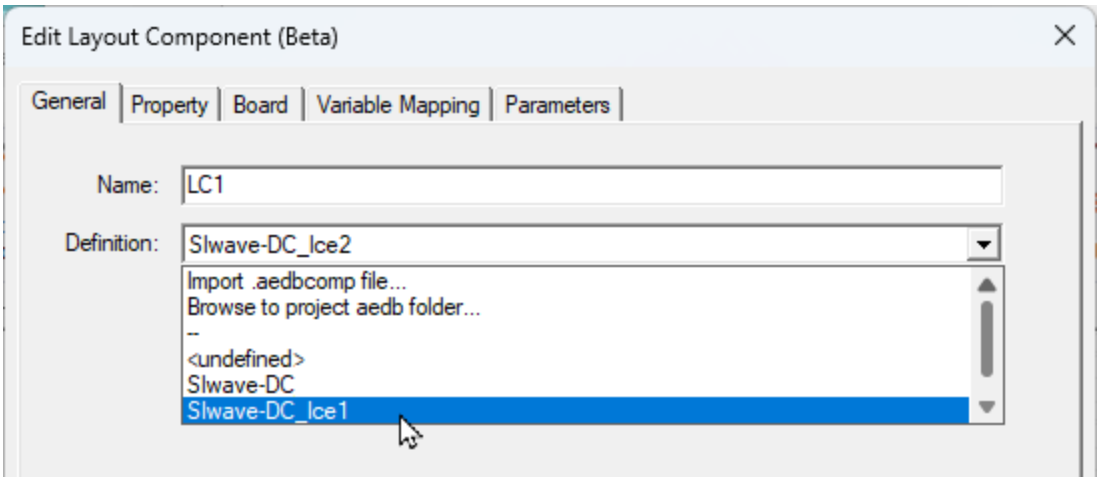
Replacing or Editing Layout Components

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*

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 an Icepak 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 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 Icepak 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit/Replace Procedure

To edit an existing layout component in your Icepak 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
3. Progress through the **Property**, **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 Icepak design, they will be reset to their default configuration whenever the layout component is replaced or regenerated after editing. When this occurs, repeat step 6 in the *Adding a Layout Component* procedure to restore your preferred visualization options.

Layout Component Properties

While you must create or edit the definition of a layout component using the **Insert Layout Component** dialog box (see Adding a Layout Component), you can display its information and

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **Properties** window:

- General Tab
- Parameters Tab
- Component Data Tab
- Visualization Tab

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 Icepak design)
- **Reference Coordinate System:** The coordinate system of the source HFSS 3D Layout design
- **Target Coordinate System:** The coordinate system in the Icepak 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 Icepak model.

The following example shows the appearance of the **Parameters** tab when the layout component has two parameters defined:

The **Name** column lists the name of the variable in the target Icepak 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** tab of the **Insert Layout Component** dialog box. See Adding a Layout Component for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Data Tab

For the layout component beta, this tab does not display any information relevant to the Icepak design.

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:

- **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.

- 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 2026 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 check box 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.

Trace Mapping Using a Layout Component

Trace mapping 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 Icepak design. Use a previously created HFSS 3D Layout design as a source for Adding a Layout Component to your Icepak design. The conducting objects are visualized in the layout component, but they are not directly included as 3D geometry.

Ansys Electromagnetics Suite 2026 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 Icepak 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 Icepak design, but it is used in the layout component solution.

In Icepak designs, cutouts and via holes are voids that are modeled as fluid objects assigned the fluid material specified to the Region object.

The signal layers exists in the Icepak 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 on the board's signal layers. Metal fraction values are also mapped along the full length of vias, through dielectric and signal layers.

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 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.

PCB Grid Resolution

The grid resolution that is used to create the thermal conductivity distribution of the board can be customized. Depending on the trace resolution and the computational costs desired, you can change the values for the columns and rows to receive optimum results. The size of the grid is in parentheses next to the column and row count.

Note: When linked from an HFSS 3D Layout design, circles and arcs in traces are approximated as polygons with a maximum of 12 points in an Icepak design.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following points outline best practices for balancing accuracy and cost of computing the cell by cell orthotropic thermal conductivity:

- For board-level simulations, enter column and row counts that yield at most a grid size equal to the minimum trace width or minimum via diameter. If possible, use column and row counts that yield a grid size equal to half the minimum trace width or half the minimum via diameter.
- For system-level simulations, enter column and row counts to yield at most a grid size that equals four to eight times the minimum trace width or minimum via diameter.
- For all models, the CFD mesh on the board should equal four to eight times the grid size.
- Column and row counts or CFD meshes that significantly deviate from the above guidelines can adversely affect accuracy.

Note: When using the slider bar to set the grid resolution, the Ansys Electronics Desktop application uses the aspect ratio to determine the resolution of the longer side. At low resolution, you may observe a slight grayscale pixel difference and asymmetry at the trace boundaries. Also, you may observe slight differences between using the slider bar and custom grid resolution settings.

Note: You may observe slight differences between Classic Icepak and Icepak in the Electronics Desktop as their metal fraction computation and display methods differ.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - Generating a Mesh

Once you have finished designing your model, you need to generate the computational mesh that is used as the basis of the solution procedure. The mesh consists of discrete elements located throughout the computational region. Within each element, Icepak solves the equations that govern the flow and heat transfer in the model.

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.

For Icepak designs, you must define mesh regions, including the outer geometry representing the cabinet. Within the cabinet geometry, you can select individual geometric objects in the history tree and define them as mesh regions with their own mesh settings.

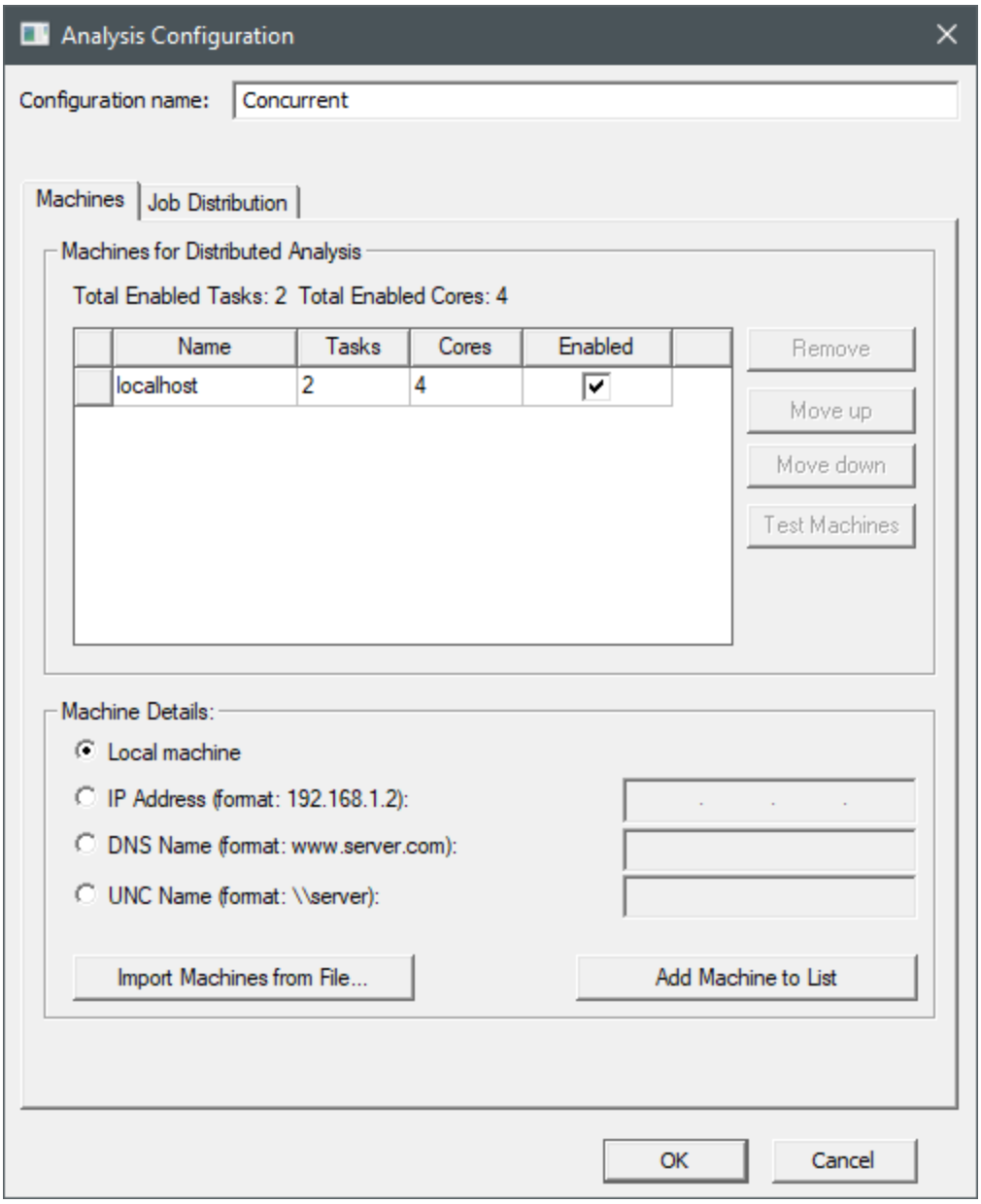
Enabling Concurrent Meshing

Using the Electronics Desktop High Performance Computing options, Icepak can generate mesh for multiple mesh regions concurrently, decreasing meshing time. Use the following procedure to define an analysis configuration for concurrent meshing.

> **Note:** Concurrent meshing is only available for the local machine.

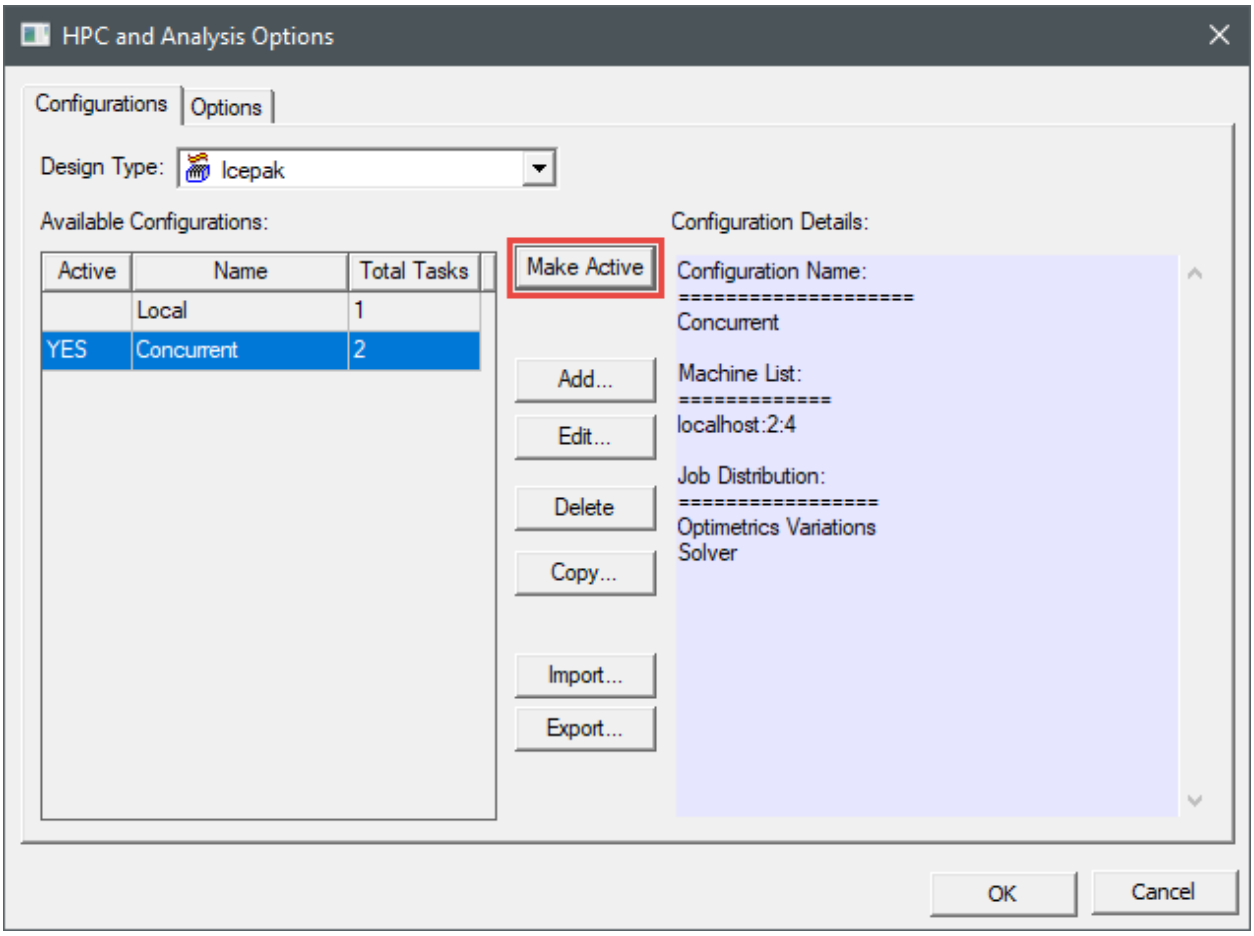
1. On the **Simulation** ribbon, click **HPC Options**.
2. In the **HPC and Analysis Options** dialog box, click **Add**.
3. For **Configuration name**, enter a name for the analysis configuration.
4. On the **Machines** tab, click **Add Machine to List**.
5. Enter a value greater than 1 for **Tasks** and **Cores**.
    - **Tasks** defines how many instances of the Icepak mesher are used.
    - **Cores** defines how many processor cores are used.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Click **OK**.
7. In the **HPC and Analysis Options** dialog box, select the new analysis configuration and click **Make Active** to set the configuration as active for use during meshing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



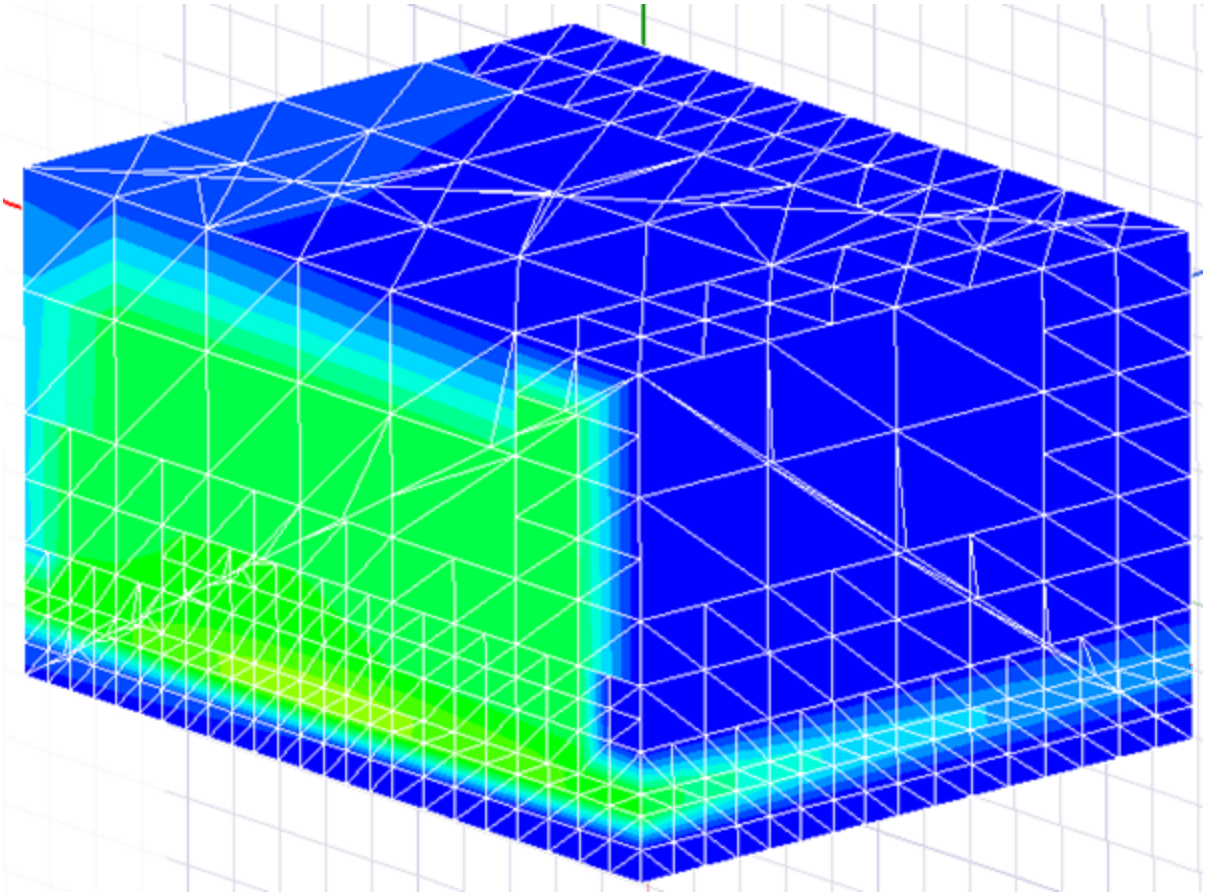
8. Click **OK** to close the **HPC and Analysis Options** dialog box.

Hybrid Mesh

Hybrid mesh instructs the solver to allow hexa and tetra elements to exist in the mesh. Elements other than hexa elements, such as pyramid and prism, are split into tetra elements. Without splitting hexa elements, creating postprocessing objects and using post-processing tools (e.g,. fields calculator or fields summary) is more efficient.

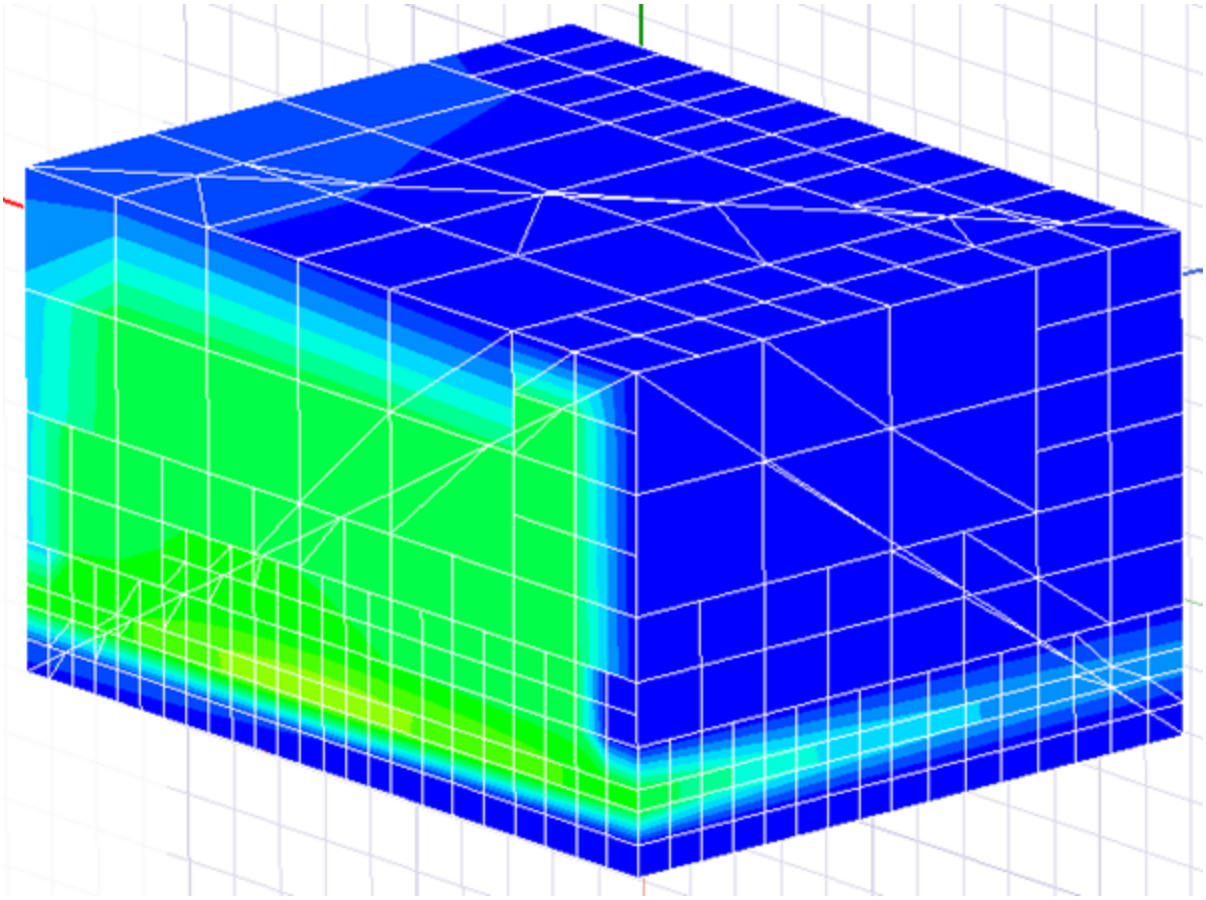
The following images present an example with hybrid mesh disabled.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following images present an example with hybrid mesh enabled.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Quality and Type

The quality of the mesh is one of the most critical aspects of a CFD model. A good mesh is essential for a good solution. A good mesh requires proper resolution, smoothness, low skewness, and an appropriate number of elements. The main requirements can be summarized as follows:

- The mesh must be fine near objects where the gradients of temperature and velocity may be very large (for example, heated blocks or plates, cabinet walls with nearby objects).
- The expansion ratio from one mesh element to the next should be kept in the range between 2 and 5, although in some critical areas a lower value might be better.
- An equilateral element (cube or equilateral tetrahedron) is optimal. Since it is generally not possible to have only optimal elements, you should instead focus on maintaining a low aspect ratio and regular (not skewed) shape for each element. This will reduce the number of long, thin elements and the number of distorted elements, both of which can decrease accuracy and destabilize the solution. Figure 1 shows examples of elements with low and high skew.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

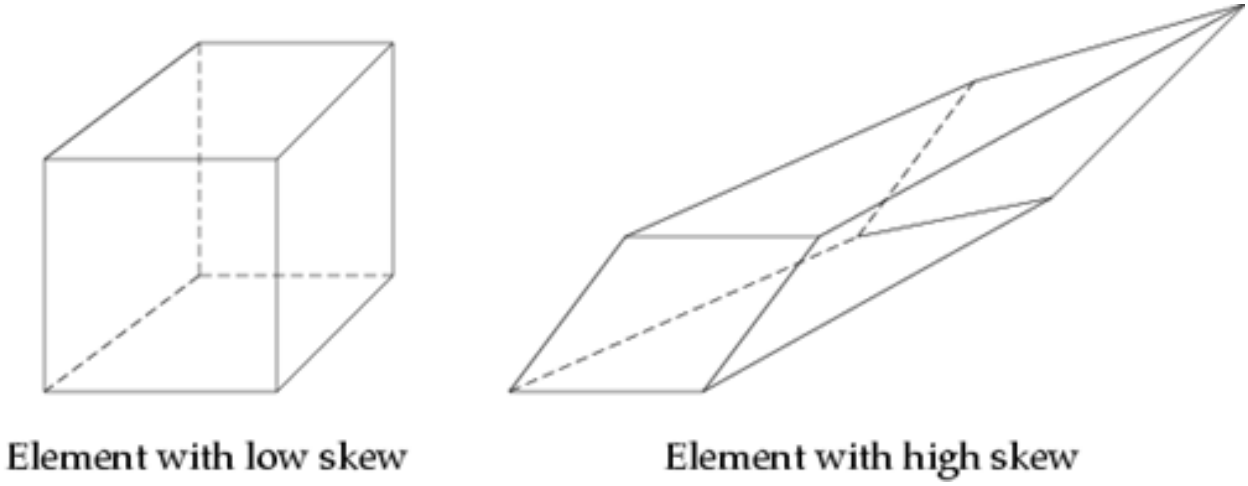


Fig. 1 Elements with Low and High Skew

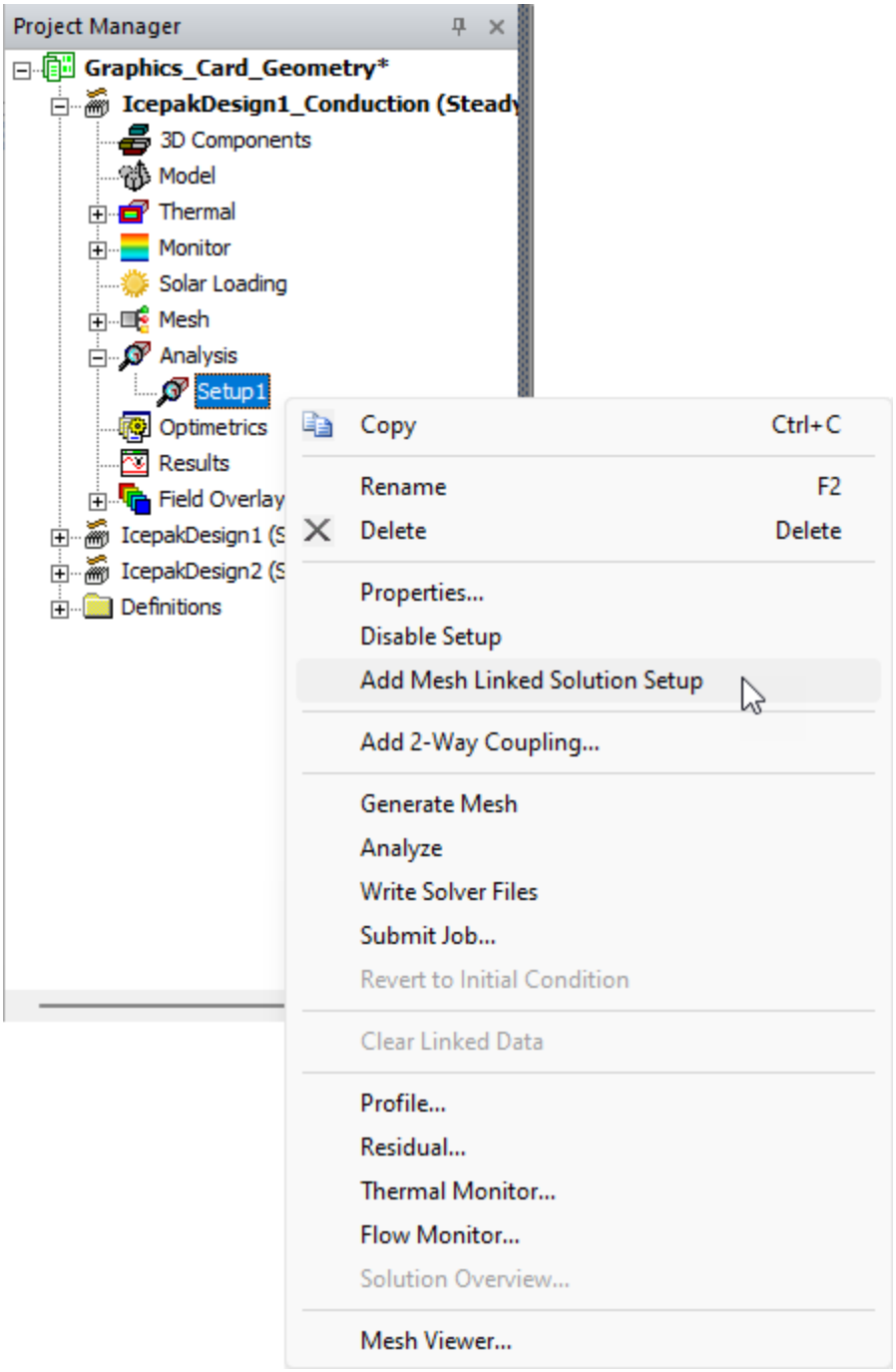
- It is also possible to have non-conformal meshing in a certain region of the model to improve grid quality and/or to reduce the mesh count. A bounding box can be applied to a certain region, and the mesh inside this region need not match the mesh outside the region.
- For an efficient calculation, the mesh should be coarser in areas where the gradients of velocity and temperature are small. Since there are no small changes in flow behavior to be captured, it would be wasteful to have a fine mesh in such regions. The cost of the calculation will be directly proportional to the number of elements in the mesh, so it is best to concentrate the elements where you need them, and reduce the number of elements elsewhere.

Adding a Mesh Linked Solution Setup

An Icepak solution setup can use mesh from an existing Icepak design. You can create a linked solution setup from the right-click menu.

In the **Project Manager** under **Analysis**, right-click on a solution setup and select **Add Mesh Linked Solution Setup**.

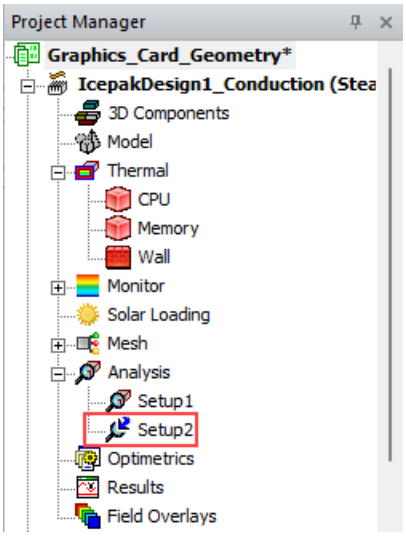
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> A solution setup's right-click menu contains the **Mesh Viewer** option. Click **Mesh Viewer** to display the mesh specific to the selected setup.

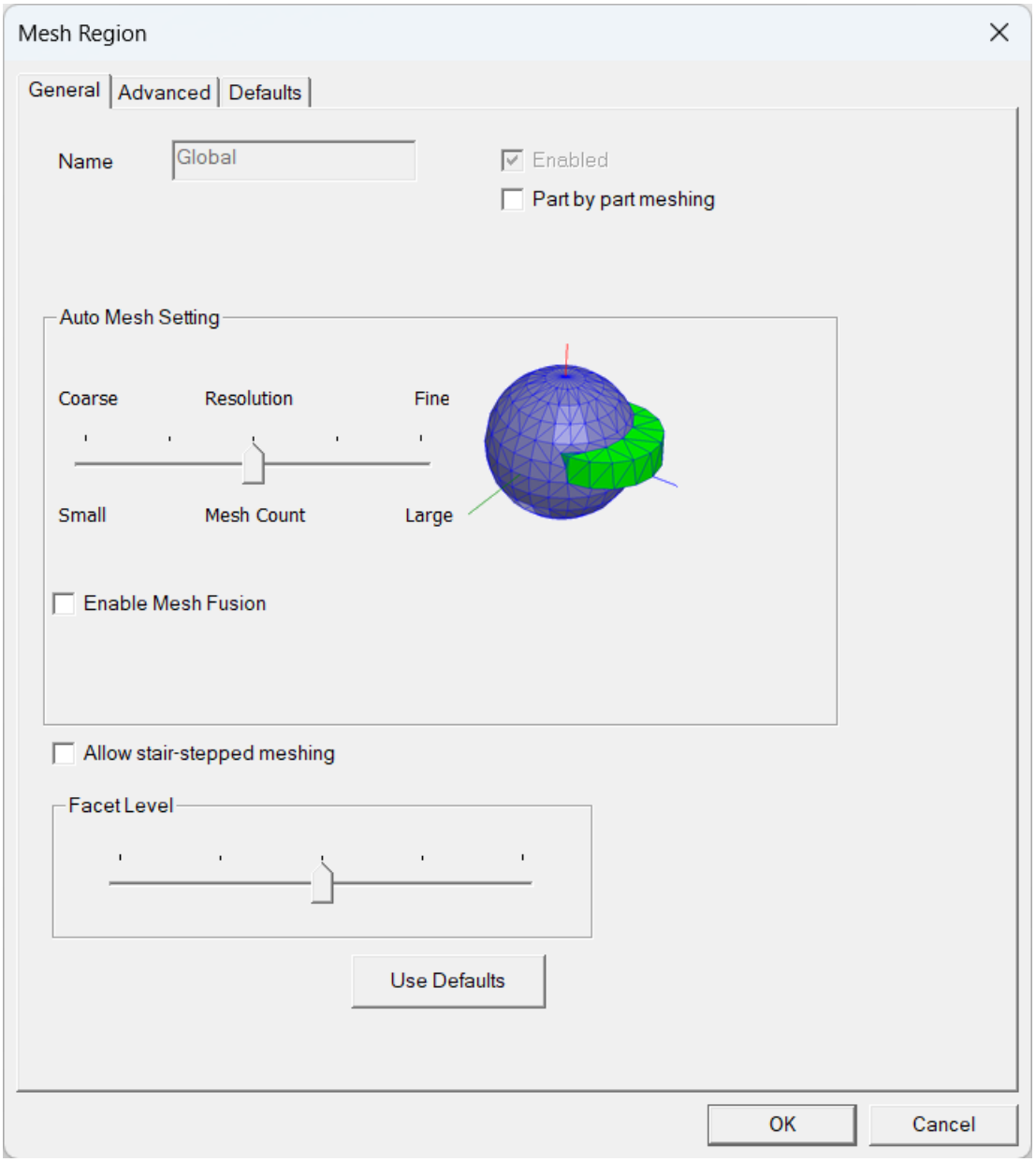
The linked solution setup is displayed in the **Project Manager**. For more information on importing mesh in the **Icepak Solve Setup** dialog box, see Importing Mesh.



Defining Global Mesh Settings

For Icepak models, a region is automatically created surrounding the various components of the model. This is the global computational region. You are able to define mesh settings for the global region to determine mesh size and other advanced settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

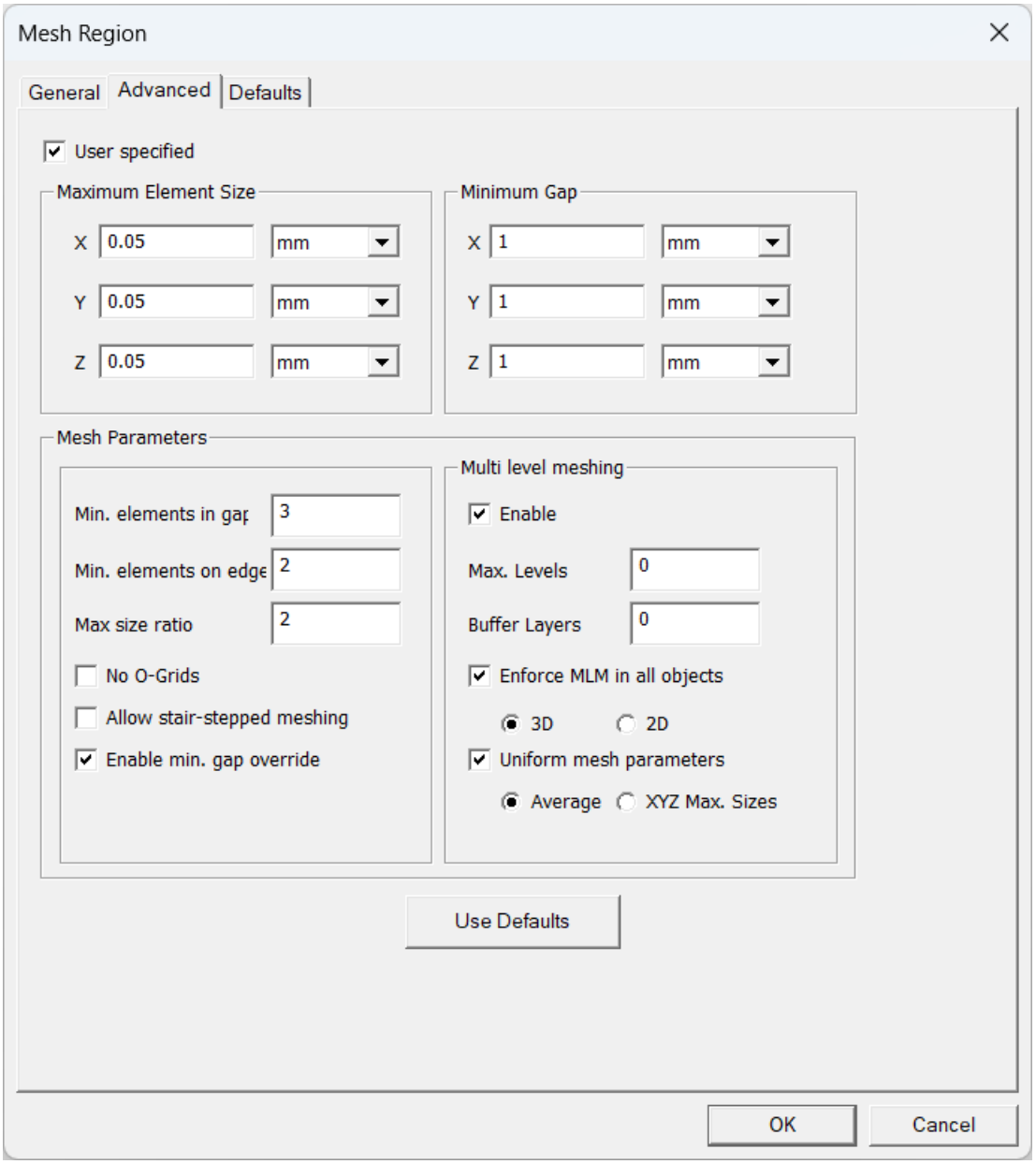


Name	The **Name** for global mesh settings is **Global** by default and is not editable.
Enabled	The global mesh region is enabled by default and cannot be disabled.
Part by part meshing (Beta)	When enabled, **Part by part meshing** meshes each part in the model individually. See Part By Part Meshing for more information.
Auto Mesh Setting	
Auto Mesh Setting	The **Auto Mesh Setting** slider includes a visual representation of the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	resolution you choose as ranging from coarse resolution with a small mesh size through a five-position scale to a fine resolution with a large mesh size. Each slider position automatically sets unique values for the following settings found on the **Advanced** tab: **Maximum Element Size**, **Min. elements in gap**, **Min. elements on edge**, **Max size ratio, Multi level meshing Max. Levels**, and **3D Multi level mesh**. Slider settings 3, 4, and 5 also enable per-object mesh levels.
Enable Mesh Fusion	**Enable Mesh Fusion** is used to automatically group and mesh geometries in a mesh region that uses the mesh slider bar. When enabled, it divides the geometries into sub-domains, which are then meshed separately using automatically selected meshing methods to yield high quality meshes.
Note: Mesh fusion can be enabled or disabled on a design level in the Global **Mesh Region** dialog box. It can be enabled or disabled for individual mesh regions on the **General** tab of the **Mesh Region** dialog box. See Mesh Fusion for more information.	
Allow stair-stepped meshing	**Allow stair-stepped meshing** disables projection and decomposition during 3D cut cell meshing. The boundary mesh will not be conformal, and the resulting mesh will only approximate the shapes of your geometries. When enabled, stair-stepped meshing is applied to the global mesh region and all mesh regions within it. Stair-stepped meshing is not applied when **User specified** is enabled on the **Advanced** tab. Stair-stepped meshing should only be used if resolving the shape of the geometry is not important.
Facet Level	The **Facet Level** slider has a five-position scale to define the facet level of CAD objects in the design. Facet count increases as the slider position increases from 1 to 5. That is, slider setting 1 results in the fewest facets, and setting 5 results in the most facets. As the slider setting increases, the meshing of rounded corners and edges increases in resolution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Element Size	
X, **Y**, **Z**	**Maximum Element Size** specifies the desired maximum element length in each coordinate direction. Typical values are about 1/20 of the region dimensions in the corresponding directions.
Minimum Gap	
X, **Y**, **Z**	**Minimum Gap** specifies the minimum distances separating objects in your model in the X, Y, and Z coordinate directions. Minimum gap influences geometry and meshing.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• Geometry: If the gap between geometries is smaller than 10% of the minimum gap setting, the gap is closed (that is, one geometry is snapped to the match the adjacent geometry). • Meshing: Icepak HD Mesher calculates a meshing tolerance based on the minimum gap, domain dimensions, the smallest geometric feature, and other factors. If the tolerance is calculated such that it is smaller than the geometric gap, the gap is meshed.
Mesh Parameters	
Min. elements in gap	**Min. elements in gap** specifies the minimum number of elements between adjacent objects.
Min. elements on edge	**Min. elements on edge** specifies the minimum number of elements on each edge of each object.
Max size ratio	**Max size ratio** specifies the maximum ratio of the sizes of adjacent elements (for the whole model).
No O-Grids	**No O-Grids** (**Mesher-HD** and **Hexa Unstructured** only) specifies whether or not objects will have O-grids around them. This option is disabled by default, indicating that Icepak will generate O-grids around all objects, including those that contain other objects.
Allow stair-stepped meshing	**Allow stair-stepped meshing** disables projection and decomposition during 3D cut cell meshing. The boundary mesh will not be conformal, and the resulting mesh will only approximate the shapes of your geometries. Stair-stepped meshing should only be used if resolving the shape of the geometry is not important.
Enable min. gap override	Icepak computes the minimum size of bounding boxes of all objects in each coordinate direction. If this computed value is smaller than the specified **Minimum Gap** value, Icepak uses the smaller value when **Enable min. gap override** is enabled.
Multi level Meshing (MLM)	
Max. Levels	**Max levels** specifies the maximum number of levels that may be required to achieve the desired mesh refinement based upon curvature/proximity. For a given coarse mesh, the desired proximity/curvature based refinement may be achieved even before reaching the max levels.
Buffer Layers	**Buffer Layers** is used in conjunction with multi-level meshing. The default value is zero, which means refinement is not propagated to adjacent layers. When you set this value to a number greater than zero, such as 1 or 2, refinement is propagated to additional mesh layers, which helps in generating better meshes in the region of under-resolved geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

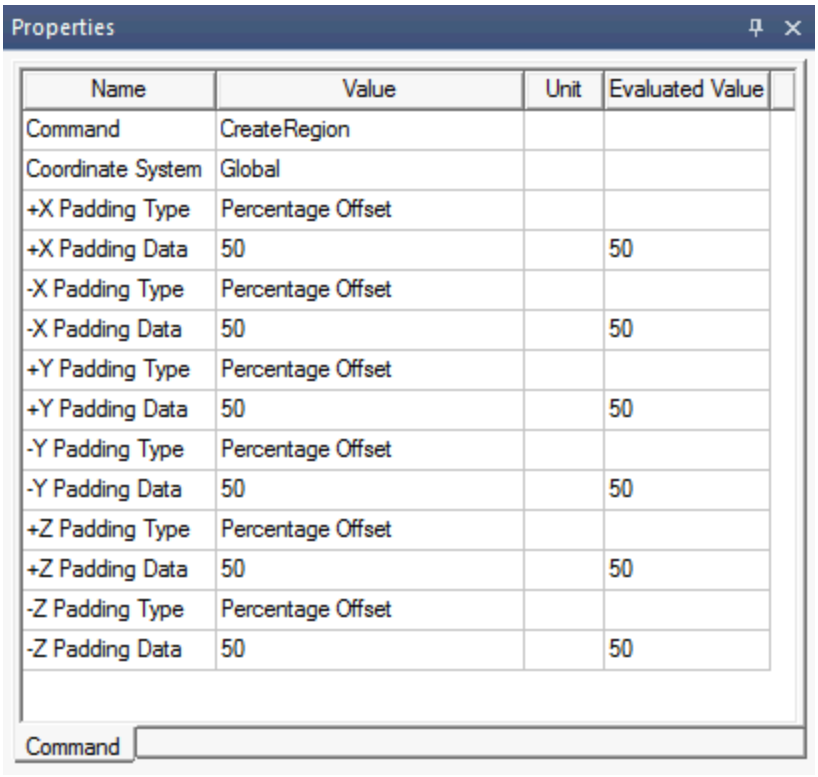
Enforce MLM in all objects	**Enforce MLM in all objects** starts with a coarse background mesh and then refines the mesh in the planar directions to resolve fine-level features. You can select to enforce multi-level meshing in **3D** or **2D** objects.
Uniform mesh parameters	**Uniform mesh parameters** has two options: **Average** and **XYZ Max. Sizes**. **Average** creates a uniform mesh with the same mesh size in all coordinate directions. The mesh size is computed by averaging the specified max element sizes.**XYZ Max. Sizes** creates a uniform mesh in each coordinate direction using the corresponding max element size. The resulting mesh has constant spacing in each coordinate direction but the spacing can be different if the max element sizes are different. Using uniform mesh parameters results in lower mesh count and improved quality. It is recommended to use uniform meshing parameters when multi-level meshing is used. If this option is enabled, 3D cut cells and stair-step meshing ignores the object-based parameters to keep the mesh uniform and generates better quality meshes.

You can adjust the global region padding by selecting the region in the history tree and editing the **+/-X**, **Y**, and **Z Padding Data** in the **Properties** window. Click the Value field for **+/-X**, **Y**, and **Z Padding Data** to select from the following padding methods:

- **Percent Offset**
- **Transverse Percentage Offset**
- **Absolute Offset**
- **Absolute Position**

> **Note:**
>
> For **Absolute Offset** and **Absolute Position**, ensure the **Unit** is set to the appropriate unit of measure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To define global mesh settings:

1. From the **Icepak**>**Mesh** menu, select **Edit Global Region**.

   > **Note:**
   >
   > Alternatively, access the **Mesh** dialog box by right-clicking **Mesh** in the **Project Manager** and selecting **Edit Global Region**.

2. If the simulation is conduction-only, select **Part by part meshing** to mesh parts individually. Part by part meshing is a beta option.
3. In the **Mesh Region** dialog box on the **General** tab, use the **Auto Mesh Setting** slider to select a mesh resolution between **Coarse** and **Fine**.
4. If needed, select **Enable Mesh Fusion**.
5. If needed, select **Allow stair-stepped meshing**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Use the **Facet Level** slider to define the facet level of CAD objects.

   > **Note:**
   >
   > **Facet Level** settings are not applied to CAD objects imported from .stl or .tzr files.

7. On the **Advanced** tab, if needed, select the **User specified** check box to enable and define the following settings:
   - **Maximum Element Size**
   - **Minimum Gap**
   - **Mesh Parameters**
     - **Min. elements in gap**
     - **Min. elements on edge**
     - **Max Size Ratio**
     - **No O-Grids**
8. If needed, select **Enable** under **Multi level meshing** and define the following settings:
   - **Max levels**
   - **Buffer Layers**
   - **Enforce MLM in all objects**
   - **Uniform mesh parameters**
9. If needed, select the mesh region in the history tree and edit the padding values in the **Properties** window.
10. Click **OK**.

Part By Part Meshing (Beta)

Part by part meshing instructs the solver to mesh each model part individually, eliminating the need for individual mesh regions and thereby simplifying the meshing process.

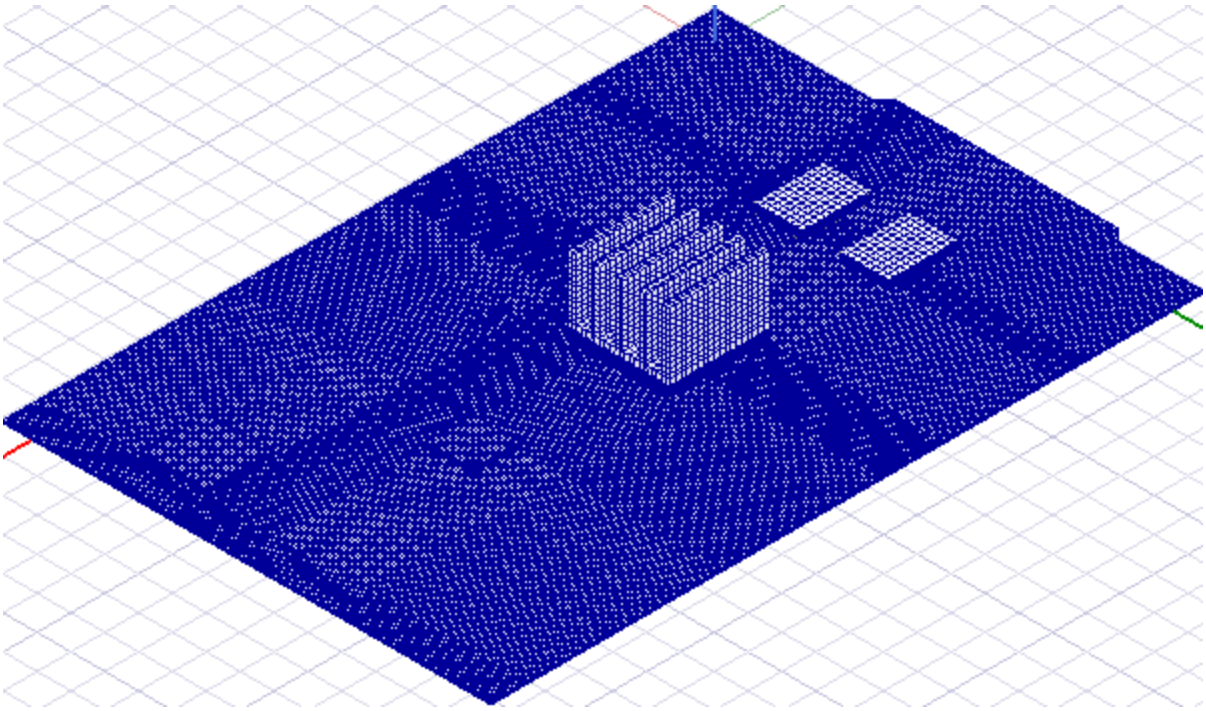
> **Note:** Part by part meshing is currently a beta feature. For information on enabling beta options, see General Options: Desktop Configuration.

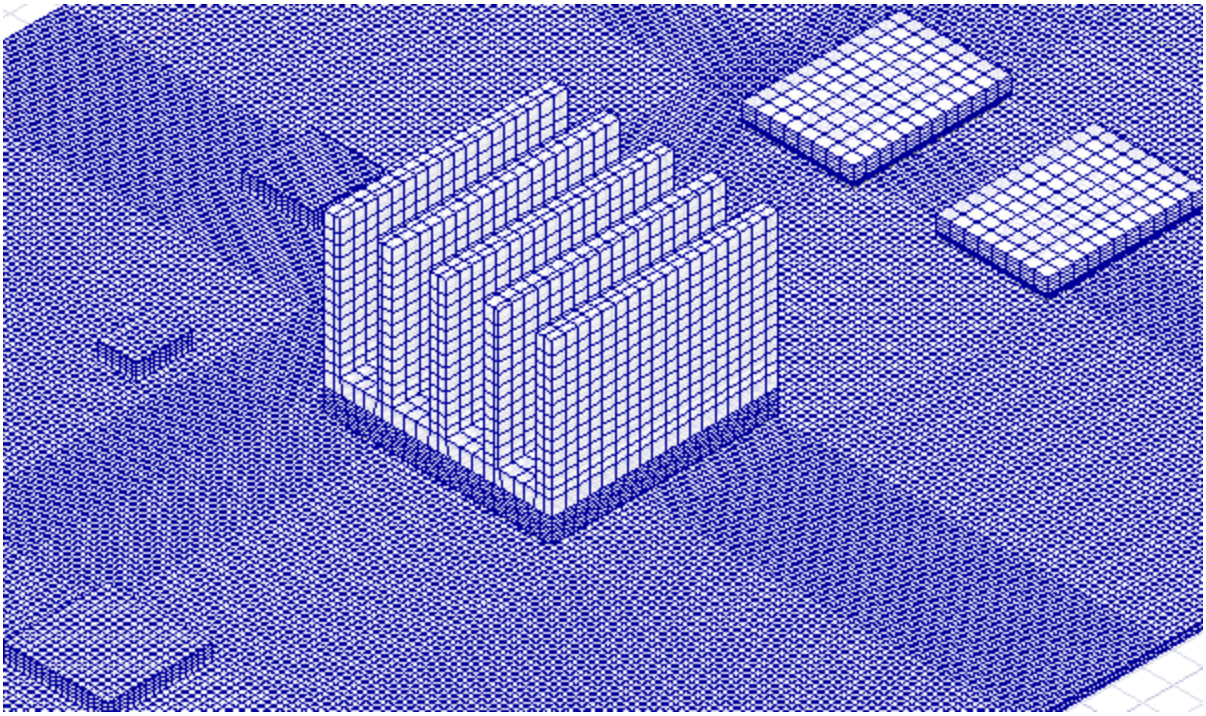
> **Important:** Part by part meshing should be used only on conduction-only simulations.

> **Note:** Since each part is meshed individually, the mesh may not be conformal at the interface between two parts that are in contact unless a boundary condition has been specified at that interface.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following images show mesh for a PCB and components with part by part meshing enabled.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

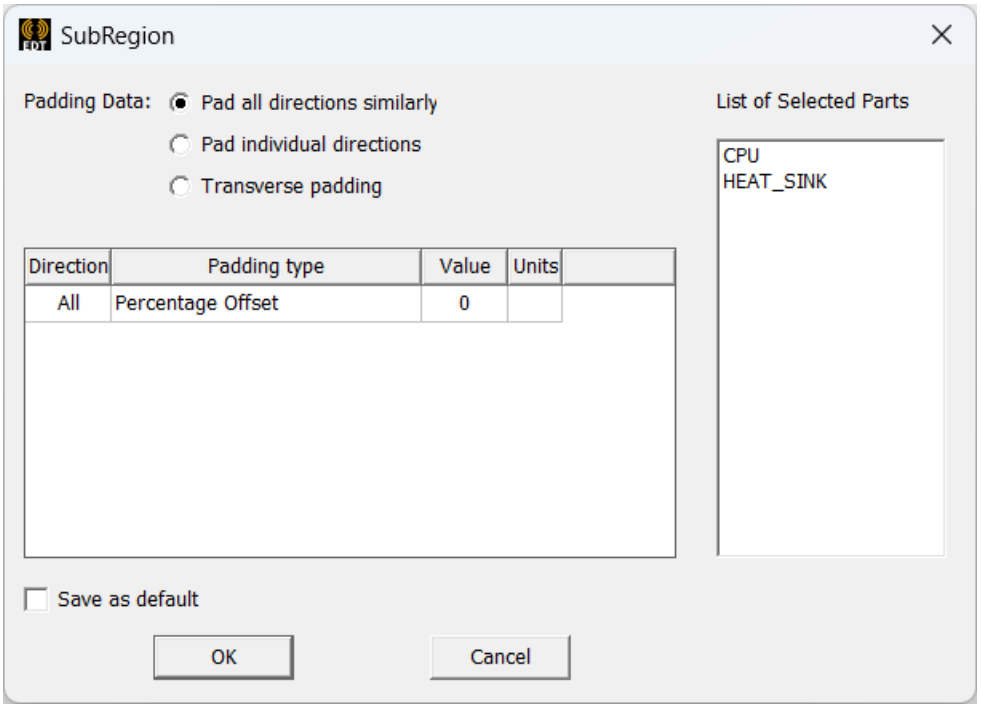
Assigning Mesh Regions

In Icepak, a mesh region can be defined around particular geometry by selecting the geometry in the history tree or **3D Modeler** window, right-clicking, and selecting **Assign Mesh Region**. Prior to defining mesh settings, you must first define the padding settings. Padding allows you to expand or shrink a mesh region in each coordinate direction. Each mesh region face can be adjusted outwardly or inwardly. Specify a positive value to expand the mesh region or a negative value to shrink it.

> **Note:** A mesh region cannot be shrunk to a volume of zero or expanded outside the outer Region (computational domain).

A non-model sub-region is created using the current coordinate system. This region can be meshed independently of the global mesh outside it. At the interface with the global mesh region, there is a many-to-one transition between the mesh cells inside the mesh region and those outside it in the global mesh region, and a non-conformal interface is created at the boundary between the meshes. This allows for the embedding of a fine mesh within the mesh region around geometry where it is needed and for a coarse mesh in the rest of the model.

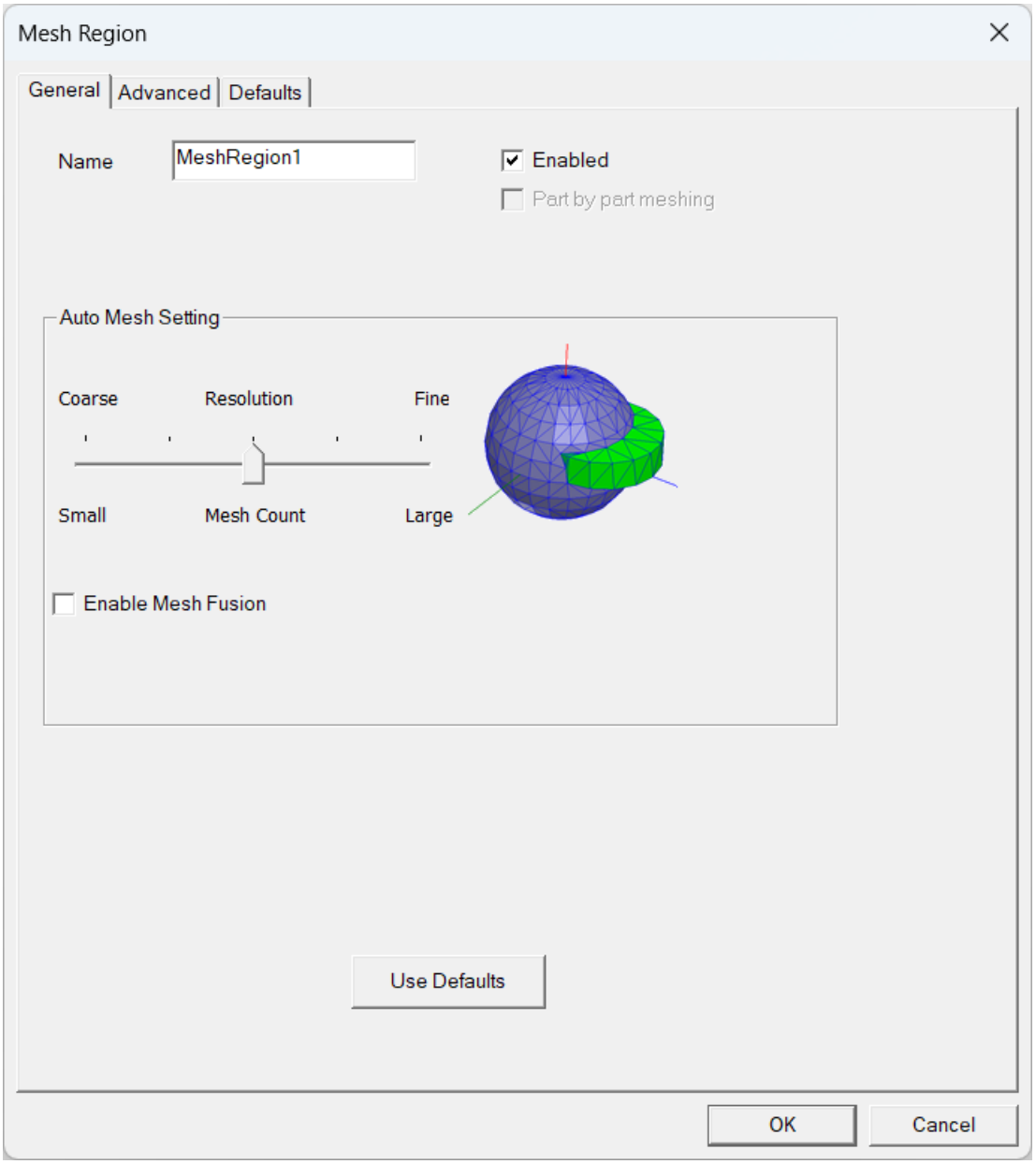
> **Note:** A mesh region can be further refined by assigning a mesh operation. See Mesh Parameters for Mesh Regions for more information.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Padding Data	
Pad all directions similarly	Select **Pad all directions similarly** to specify padding as 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**, also specify the **Units**.
Pad individual directions	Select **Pad individual directions** to specify padding for each coordinate direction individually. In this case, you can specify the **Padding type** by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.
Transverse padding	Select **Transverse padding** to specify padding for each coordinate direction by a percentage of the other two coordinate direction's diagonal length. For example, the X direction is padded using the specified percentage of the diagonal length of the other two directions (Y and Z).
List of Selected Parts	**List of Selected Parts** displays all pieces of geometry that will be enclosed within the mesh region.

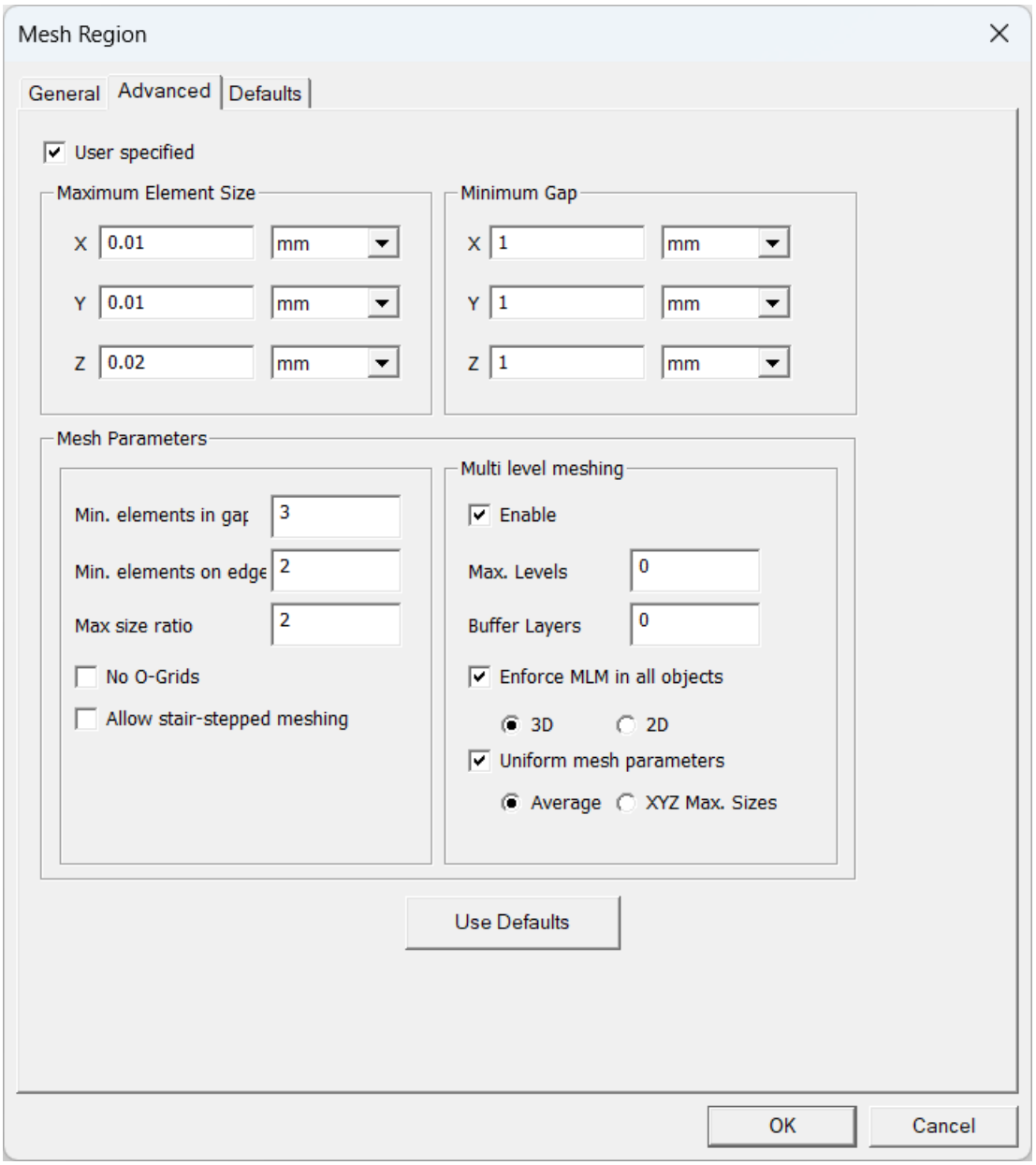
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** of the mesh region appears in the **Project Manager** window and **3D Modeler** window.
Auto Mesh Setting	
Resolution/Mesh Size	The **Auto Mesh Setting** slider includes a visual representation of the resolution you choose as ranging from coarse resolution with a small mesh size through a five-position scale to a fine resolution with a large mesh size. Each slider position automatically sets unique values for the following settings found on the **Advanced** tab: **Maximum Element Size**, **Min. elements in gap**, **Min.**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	elements on edge, **Max size ratio, Multi level meshing Max. Levels**, and **3D Multi level mesh**. Slider settings 3, 4, and 5 also enable per-object mesh levels.
Enable Mesh Fusion	**Enable Mesh Fusion** is used to automatically group and mesh geometries in a mesh region that uses the mesh slider bar. When enabled, it divides the geometries into sub-domains, which are then meshed separately using automatically selected meshing methods to yield high quality meshes.
Note: Mesh fusion can be enabled or disabled on a design level in the Global **Mesh Region** dialog box. It can be enabled or disabled for individual mesh regions on the **General** tab of the **Mesh Region** dialog box. See Mesh Fusion for more information.	



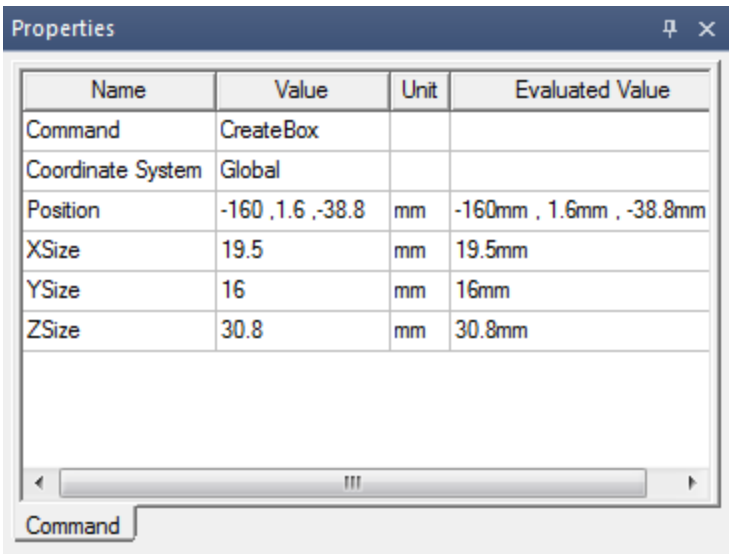
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Element Size	
X, **Y**, **Z**	**Maximum Element Size** specifies the desired maximum element length in each coordinate direction. Typical values are about 1/20 of the region dimensions in the corresponding directions.
Minimum Gap	
X, **Y**, **Z**	**Minimum Gap** specifies the minimum distances separating objects in your model in the X, Y, and Z coordinate directions. Minimum gap influences geometry and meshing. • Geometry: If the gap between geometries is smaller than 10% of the minimum gap setting, the gap is closed (that is, one geometry is snapped to the match the adjacent geometry). • Meshing: Icepak HD Mesher calculates a meshing tolerance based on the minimum gap, domain dimensions, the smallest geometric feature, and other factors. If the tolerance is calculated such that it is smaller than the geometric gap, the gap is meshed.
Mesh Parameters	
Min. elements in gap	**Min. elements in gap** specifies the minimum number of elements between adjacent objects.
Min. elements on edge	**Min. elements on edge** specifies the minimum number of elements on each edge of each object.
Max size ratio	**Max size ratio** specifies the maximum ratio of the sizes of adjacent elements (for the whole model).
No O-Grids	**No O-Grids** (**Mesher-HD** and **Hexa Unstructured** only) specifies whether or not objects will have O-grids around them. This option is disabled by default, indicating that Icepak will generate O-grids around all objects, including those that contain other objects.
Allow stair-stepped meshing	**Allow stair-stepped meshing** disables projection and decomposition during 3D cut cell meshing. The boundary mesh will not be conformal, and the resulting mesh will only approximate the shapes of your geometries. Stair-stepped meshing should only be used if resolving the shape of the geometry is not important.
Multi level Meshing (MLM)	
Max. Levels	**Max levels** specifies the maximum number of levels that may be required to achieve the desired mesh refinement based upon curvature/proximity. For a given coarse mesh, the desired proximity/curvature based refinement may be achieved even before reaching the max levels.
Buffer Layers	**Buffer Layers** is used in conjunction with multi-level meshing. The

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	default value is zero, which means refinement is not propagated to adjacent layers. When you set this value to a number greater than zero, such as 1 or 2, refinement is propagated to additional mesh layers, which helps in generating better meshes in the region of under-resolved geometry.
Enforce MLM in all objects	**Enforce MLM in all objects** starts with a coarse background mesh and then refines the mesh in the planar directions to resolve fine-level features. You can select to enforce multi-level meshing in **3D** or **2D** objects. When **2D** is enabled, if necessary, select a coordinate direction to enforce the multi-level meshing in the specified direction. Multi-level mesh refinement is then constrained to the 2D plane.
Uniform mesh parameters	**Uniform mesh parameters** has two options: **Average** and **XYZ Max. Sizes**. **Average** creates a uniform mesh with the same mesh size in all coordinate directions. The mesh size is computed by averaging the specified max element sizes.**XYZ Max. Sizes** creates a uniform mesh in each coordinate direction using the corresponding max element size. The resulting mesh has constant spacing in each coordinate direction but the spacing can be different if the max element sizes are different. Using uniform mesh parameters results in lower mesh count and improved quality. It is recommended to use uniform meshing parameters when multi-level meshing is used. If this option is enabled, 3D cut cells and stair-step meshing ignores the object-based parameters to keep the mesh uniform and generates better quality meshes.

You can manually adjust the region size by selecting the mesh region in the history tree and editing the **Position**, **XSize**, **YSize**, and **ZSize** values. in the **Properties** window.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To assign a mesh region:

> **Note:** To assign a mesh region to an inclined non-model box, you must first create a relative coordinate system. Then create the non-model box on the relative coordinate system, and assign a mesh region to it.

1. Select an object in the history tree or **3D Modeler** window.
2. Right-click in the **3D Modeler** window and select **Assign Mesh Region**.
3. In the **SubRegion** dialog box, select a **Padding Data** option and define the following padding settings:
    - **Padding type**
    - **Value**
    - **Units**
4. In the **Mesh** dialog box on the **General** tab, use the **Auto Mesh Setting** slider to select a mesh resolution between **Coarse** and **Fine**.
5. If needed, select **Enable Mesh Fusion**.
6. From the **Coordinate System** drop-down list, select the coordinate system on which to create the mesh region.
7. On the **Advanced** tab, if needed, select the **User specified** check box to enable and define the following settings:
    - **Maximum Element Size**
    - **Minimum Gap**
    - **Mesh Parameters**
        - **Min. elements in gap**
        - **Min. elements on edge**
        - **Max Size Ratio**
        - **No O-Grids**
8. If needed, select **Enable** under **Multi level meshing** and define the following settings:
    - **Max levels**
    - **Buffer Layers**
    - **Enforce MLM in all objects**
    - **Uniform mesh parameters**
9. If needed, select the mesh region in the history tree and edit the padding values in the **Properties** window.
10. Click **OK**.

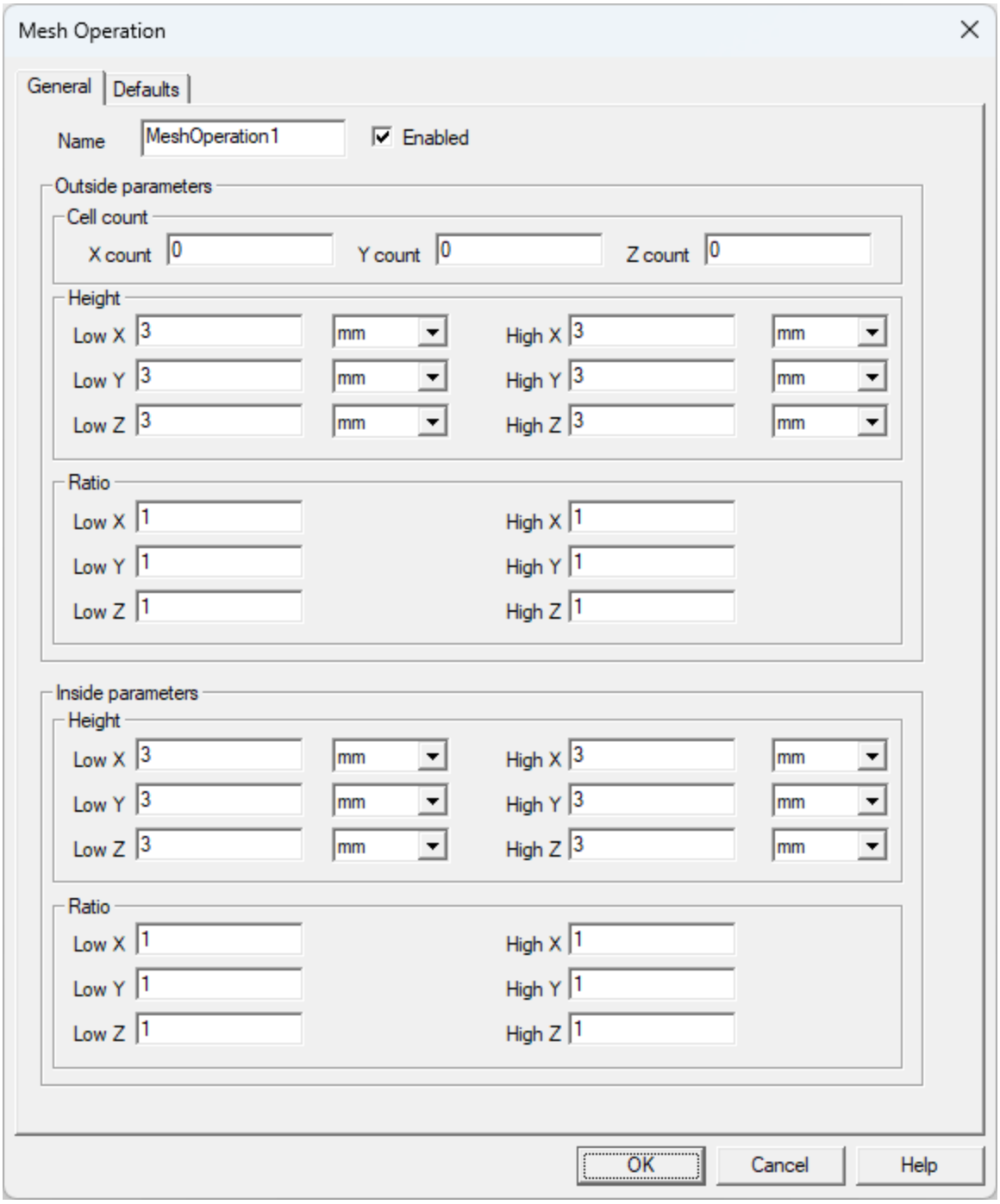
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Parameters for Mesh Regions

To assign a mesh operation with mesh parameters on a mesh region, select the box that is assigned the mesh region in the History tree or **3D Modeler** window and assign a mesh operation. Alternatively, right-click the mesh region under **Mesh** in the **Project Manager**, click **Select Assignment**, and then assign a mesh operation. On the **General** tab, specify **Outside parameters** and **Inside parameters** for the mesh region.

Mesh parameters for mesh regions apply to the low and high X, Y, and Z faces on the outside and inside faces of the mesh region. You can also specify the cell count for the outer faces of the mesh region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Outside Parameters: Cell Count	
X count	**X count** specifies the number of elements along the X axis of the outer mesh region face.
Y count	**Y count** specifies the number of elements along the Y axis of the outer

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	mesh region face.
Z count	**Z count** specifies the number of elements along the Z axis of the outer mesh region face.
Outside Parameters: Height	
Low X, Y, and Z	**Low X, Y, and Z** specifies the height of the first element on the low side surface of the mesh region in the direction normal to that surface, pointing outside of the mesh region.
High X, Y, and Z	**High X, Y, and Z** specifies the height of the first element on the high side surface of the mesh region in the direction normal to that surface, pointing outside of the mesh region.
Outside Parameters: Ratio	
Low X, Y, and Z	**Low X, Y, and Z** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the low side surface on the outside of the mesh region. Together with the height, this parameter controls mesh grading and density near the low side surface on the outside of the mesh region.
High X, Y, and Z	**High X, Y, and Z** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the high side surface on the outside of the mesh region. Together with the height, this parameter controls mesh grading and density near the high side surface on the outside of the mesh region.
Inside Parameters: Height	
Low X, Y, and Z	**Low X, Y, and Z** specifies the height of the first element on the low side surface of the mesh region in the direction normal to that surface, pointing inside of the mesh region.
High X, Y, and Z	**High X, Y, and Z** specifies the height of the first element on the high side surface of the mesh region in the direction normal to that surface, pointing inside of the mesh region.
Inside Parameters: Ratio	
Low X, Y, and Z	**Low X, Y, and Z** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the low side surface on the inside of the mesh region. Together with the height, this parameter controls mesh grading and density near the low side surface on the inside of the mesh region.
High X, Y, and Z	**High X, Y, and Z** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the high side surface on the inside of the mesh region. Together with the height, this parameter controls mesh grading and density near the high side surface on the inside of the mesh region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Fusion

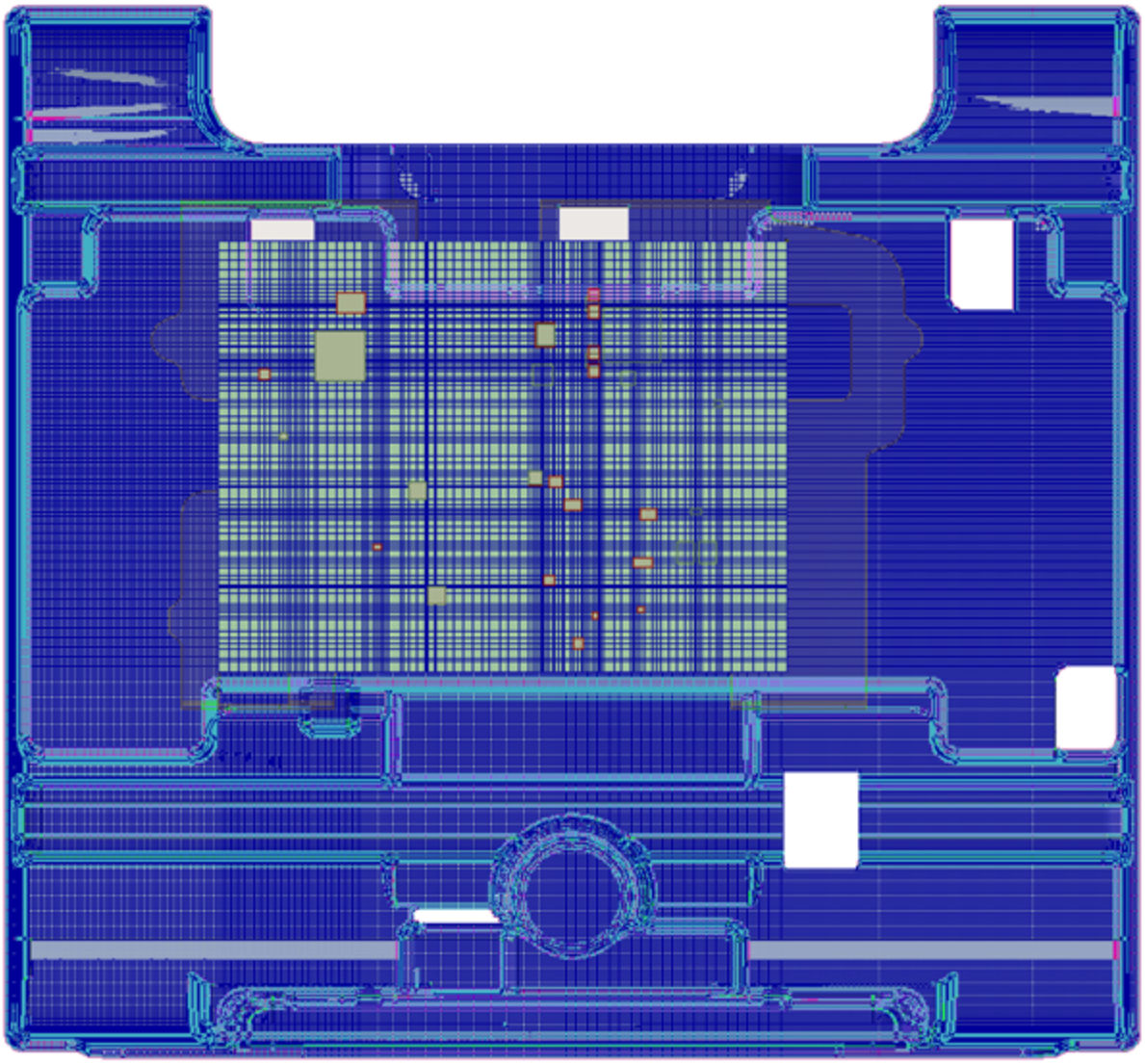
Mesh fusion automatically groups and meshes geometries in a mesh region that uses the mesh slider bar. When enabled, it divides the geometries into sub-domains, which are then meshed separately using automatically selected meshing methods to yield high quality meshes.

> **Note:** Mesh fusion can be enabled or disabled on a design level in the Global **Mesh Region** dialog box. It can be enabled or disabled for individual mesh regions on the **General** tab of the **Mesh Region** dialog box.

When you initially create a mesh region, its **Enable Mesh Fusion** setting is based on the Global mesh region **Enable Mesh Fusion** setting. Changing the **Enable Mesh Fusion** for the Global mesh region does not affect the settings of existing mesh regions.

In the model displayed below, a PCB is meshed with mesh fusion enabled.

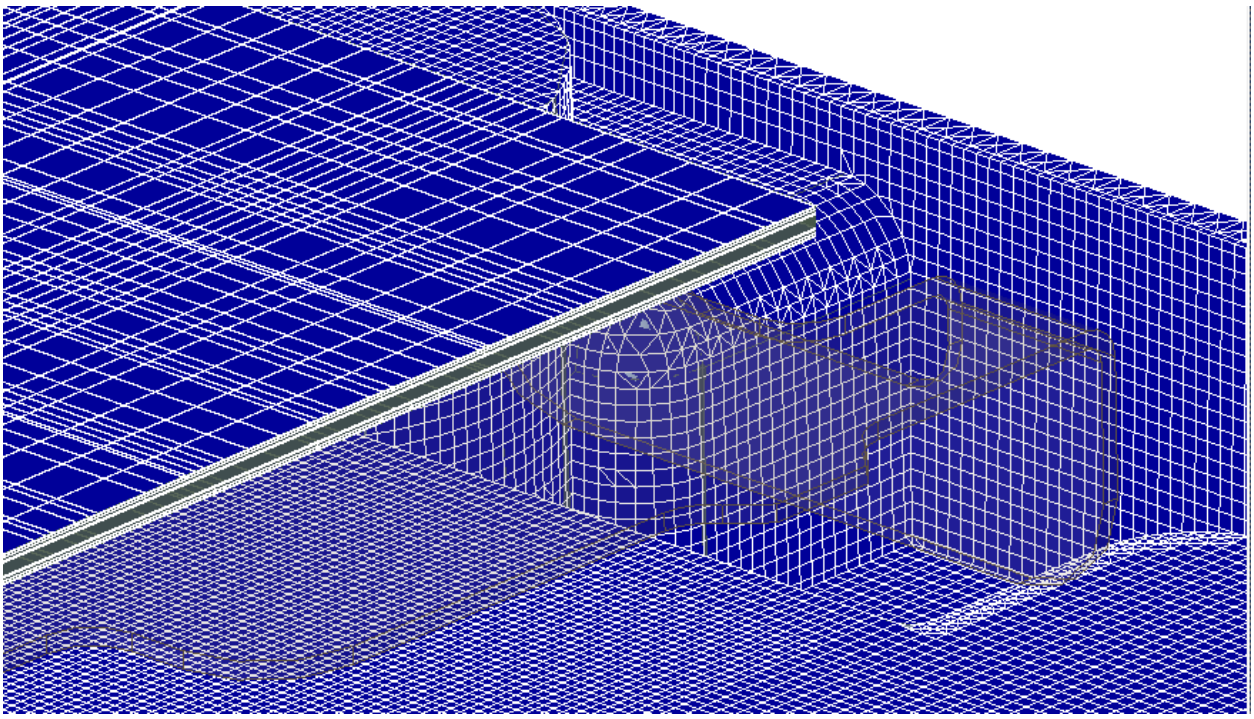
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



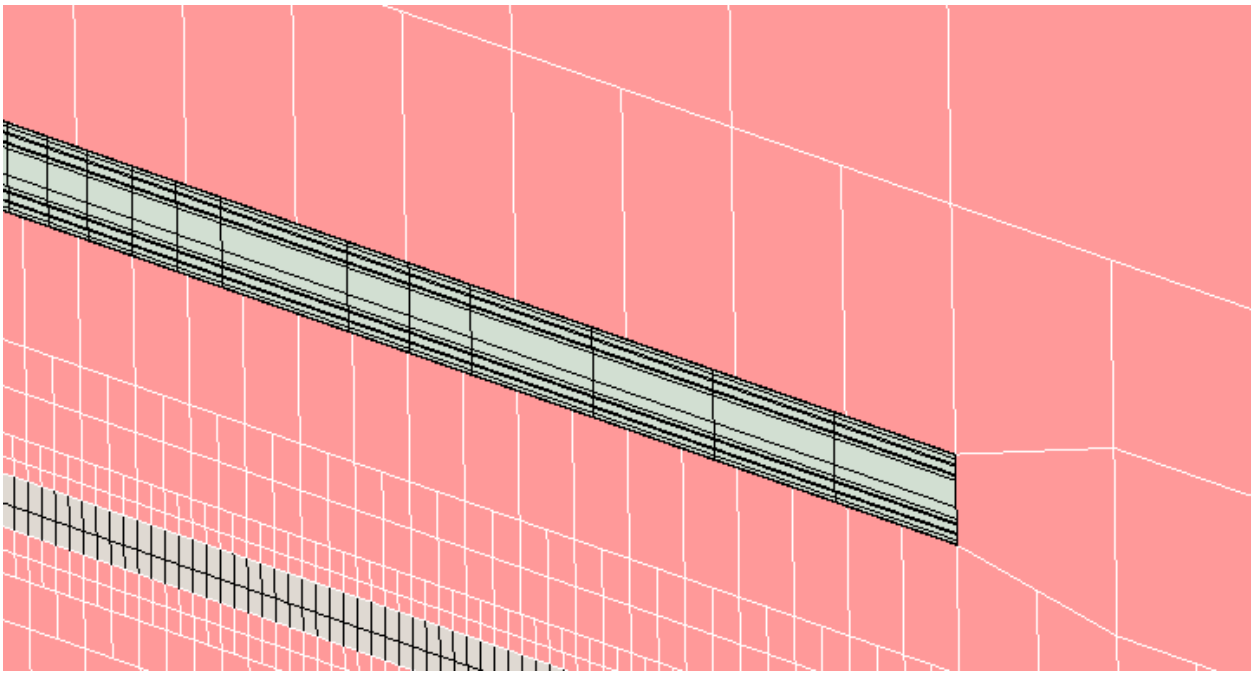
In the image below, note how simple Cartesian mesh is used to mesh the PCB, while 3D multi-level meshing is used to mesh the complex CAD enclosure.

> **Note:**
>
> The coarsest slider bat setting (level 1) is suitable for models consisted of only primitive geometries (no CAD bodies).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the image below, note the layered mesh of the PCB, which represents one sub-domain. The mesh surrounding the PCB exists outside the sub-domain and has a less refined mesh.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Mesh Operations

Multi-level meshing enables local refinement from a starting Cartesian mesh generated using the max sizes you specified. Multi-level meshing is useful if you have a large variation of length scales in your model. The refinement can be controlled such that the initial mesh is refined in an area where the mesh needs to be finer in order to resolve some geometrical features or gaps. This will significantly reduce the overall mesh count while providing adequate mesh resolution where it is needed. Multi-level meshing applies to 2D and 3D cut cell meshing.

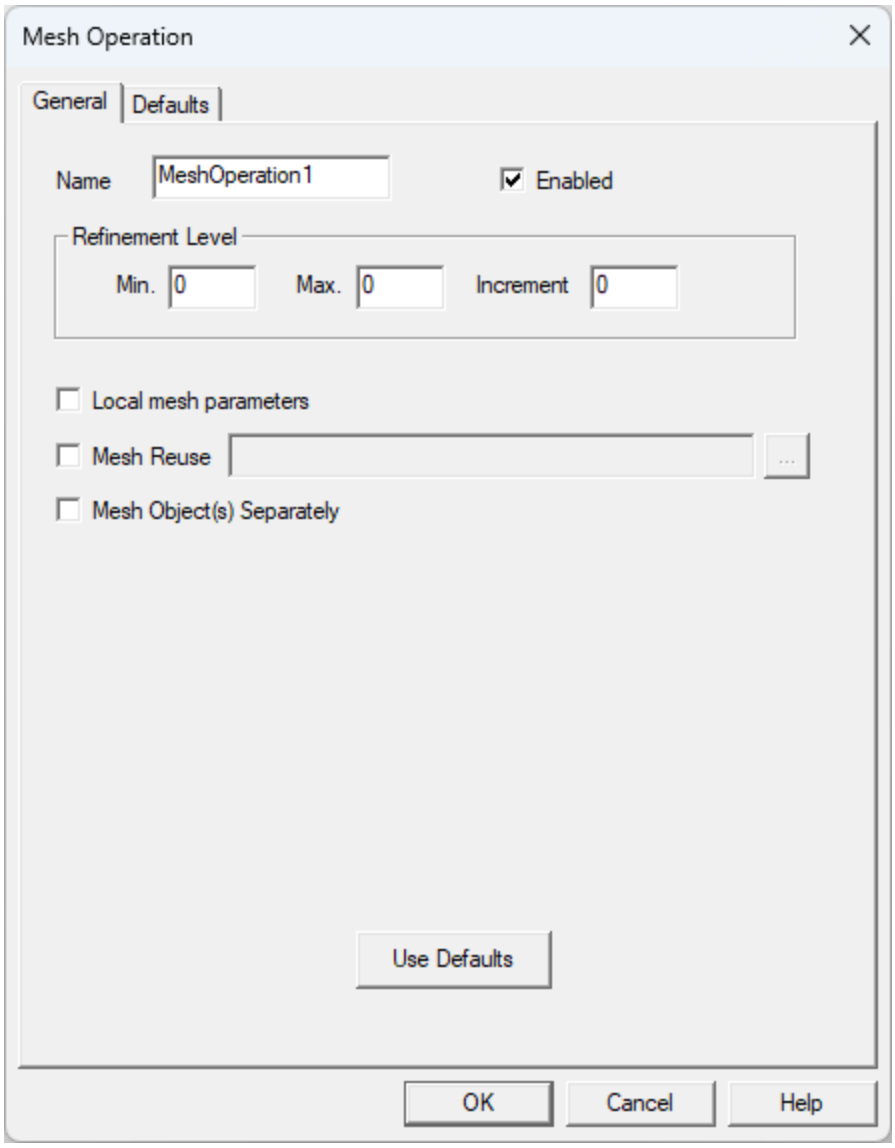
During 3D cut cell meshing, the mesh is refined based on min and max refinement level and size function requirements. If the per-object meshing level is specified for an object, the mesh around that object will be refined to that level, even if the per object meshing level is greater than the max level you specified. This enables you to localize control of the mesh refinement process whereas the use of size functions would have refined the mesh everywhere in your model where the size requirement is not met.

To specify per-object mesh levels, assign local mesh parameters, or reuse existing mesh, you must assign a mesh operation to geometry in the **3D Modeler** window.

> **Note:**
>
> To assign a mesh operation in the following procedure, the selected geometry must be model geometry. That is, in the geometry's properties, **Model** must be enabled. To assign a mesh operation that specifies element sizes, **Model** must be disabled. See Creating a Mesh Size Region for more information.

1. Enter the desired selection mode (object, face, or edge).
2. In the **3D Modeler** window or the history tree, right-click the geometry, and select **Assign Mesh Operation**.

Ansys Electromagnetics Suite 2026 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.
4. Under **Refinement Level**, enter **Min.** and **Max.** mesh levels. When using Auto Mesh Settings in any mesh regions, enter an **Increment** value. A positive number specifies a finer mesh resolution, and a negative number specifies a coarser resolution. Greater positive and negative values specify finer and coarser mesh resolutions, respectively.
5. If needed, select **Local mesh parameters**. On the **Local Mesh Parameter** tab, define the local mesh settings for the selected object, face, or edge. See Local Mesh Parameters for more information.
6. If needed, select **Mesh Reuse** and click the **[...]** button to open the **Select Mesh File** dialog box. Navigate to and select the mesh file to reuse in your Icepak design. See Mesh Reuse for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If the selected object is created in a relative coordinate system, select **Mesh Object(s) Separately** to generate mesh that is aligned with the relative coordinate system.
8. Click **OK**. The mesh operation is listed under **Mesh** > **MeshOperation** in the **Project Manager**.

Creating a Mesh Size Region

Mesh size regions allow you to refine mesh locally in a region defined by a non-model box geometry. They allow you to specify refinement size along the X, Y, and Z coordinate directions. The mesher uses the following rules to refine the mesh.

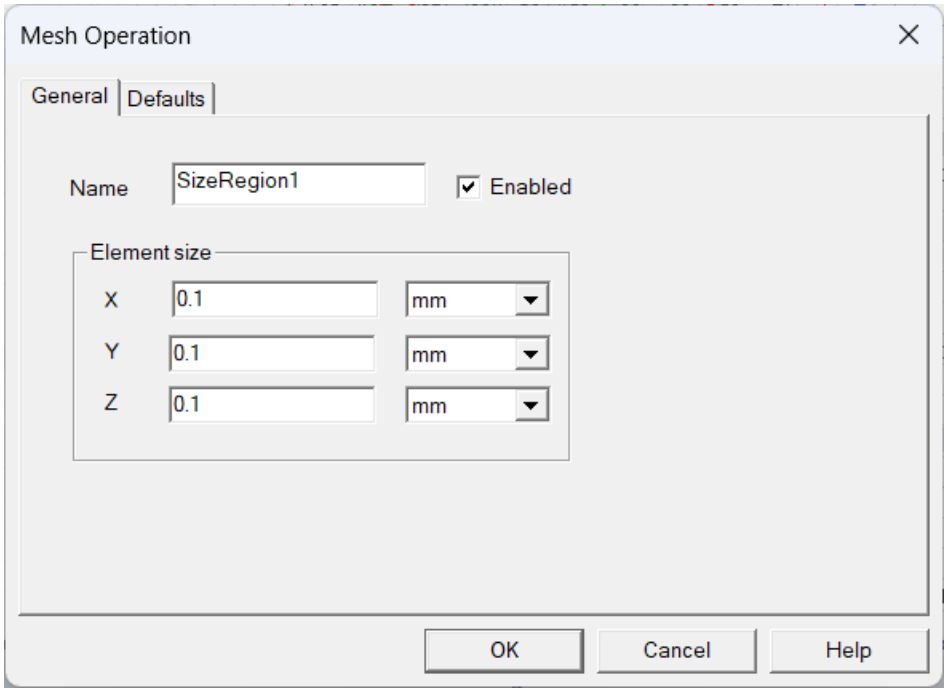
- If the surrounding mesh region has multi-level mesh enabled, the solver applies the element sizes specified in the enclosed mesh size region. If multi-level mesh is disabled in the surrounding mesh region, the mesh size region settings are not applied.
- If the surrounding mesh region has multi-level mesh and uniform mesh parameters enabled, the solver applies the smallest element size specified in the enclosed mesh size region in all coordinate directions.
- If the surrounding mesh region has multi-level mesh enabled and uniform mesh parameters disabled, the solver applies the element sizes specified in the enclosed mesh size region.
- If a mesh size region is only partially enclosed within a mesh region, only the portion within the surrounding mesh region is meshed using the element sizes specified in the enclosed size region.
- The mesh size region must be axis-aligned with the surrounding mesh region. If the mesh size region is assigned to a box that is rotated with respect to the surrounding mesh region, the mesh size region settings are not applied.

> **Note:**
>
> To assign a mesh operation in the following procedure, the selected geometry must be a non-model box. That is, in the box geometry's properties, **Model** must be disabled. To assign a mesh operation that specifies per-object mesh levels, assign local mesh parameters, or reuse existing mesh, **Model** must be enabled. See Assigning Mesh Operations for more information.

1. Press **O** to enter the object selection mode.
2. In the **3D Modeler** window or the history tree, right-click the geometry, and select **Assign Mesh Operation**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can assign the mesh size region to multiple non-model boxes. To do so, press and hold **Ctrl** and select multiple non-model boxes.



3. Enter a **Name** for the mesh size region.
4. Under **Element Size**, enter sizes for the X, Y, and Z coordinate directions.
5. Select a unit for each coordinate direction.
6. Click **OK**. The mesh operation is listed under **Mesh** > **MeshOperation** in the **Project Manager**.

Local Mesh Parameters

After reviewing an initial mesh (see Checking the Mesh and Mesh Quality and Type), if the mesh does not satisfy the requirements, assign mesh operations and define local mesh parameters to refine the mesh locally and improve the mesh quality.

- If the element count on the face of an object is low, use the X, Y, or Z count to change it.
- Use height and ratio to refine the mesh around objects.
- Refine the mesh around objects where the temperature and velocity gradients are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

expected to be high (for example, heated blocks and plates, objects blocking or diverting the flow, and fans).

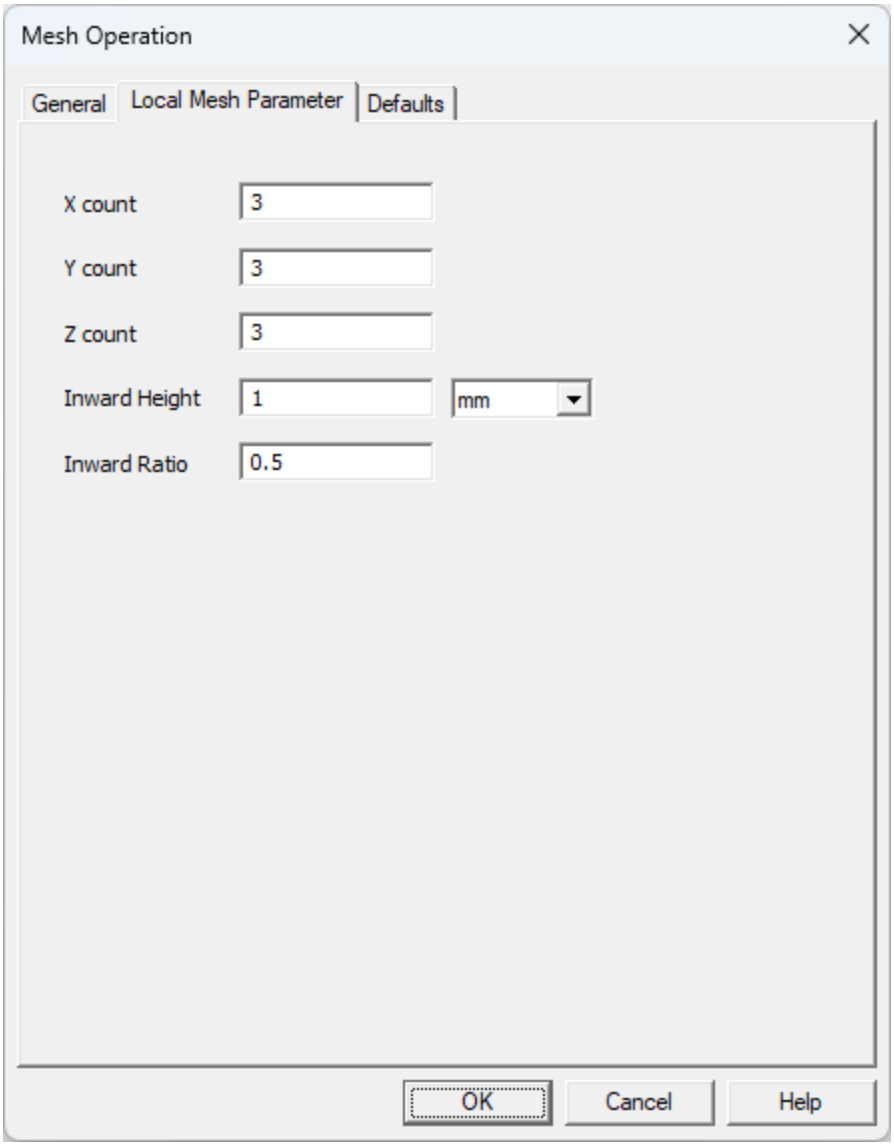
Local mesh parameters can be assigned to the following types of objects:

- Geometry
- Heat sinks

Mesh Parameters for Geometry

The settings on the **Local Mesh Parameter** tab depend on the selection mode and the geometry selected.

Box Object - Object Selection Mode

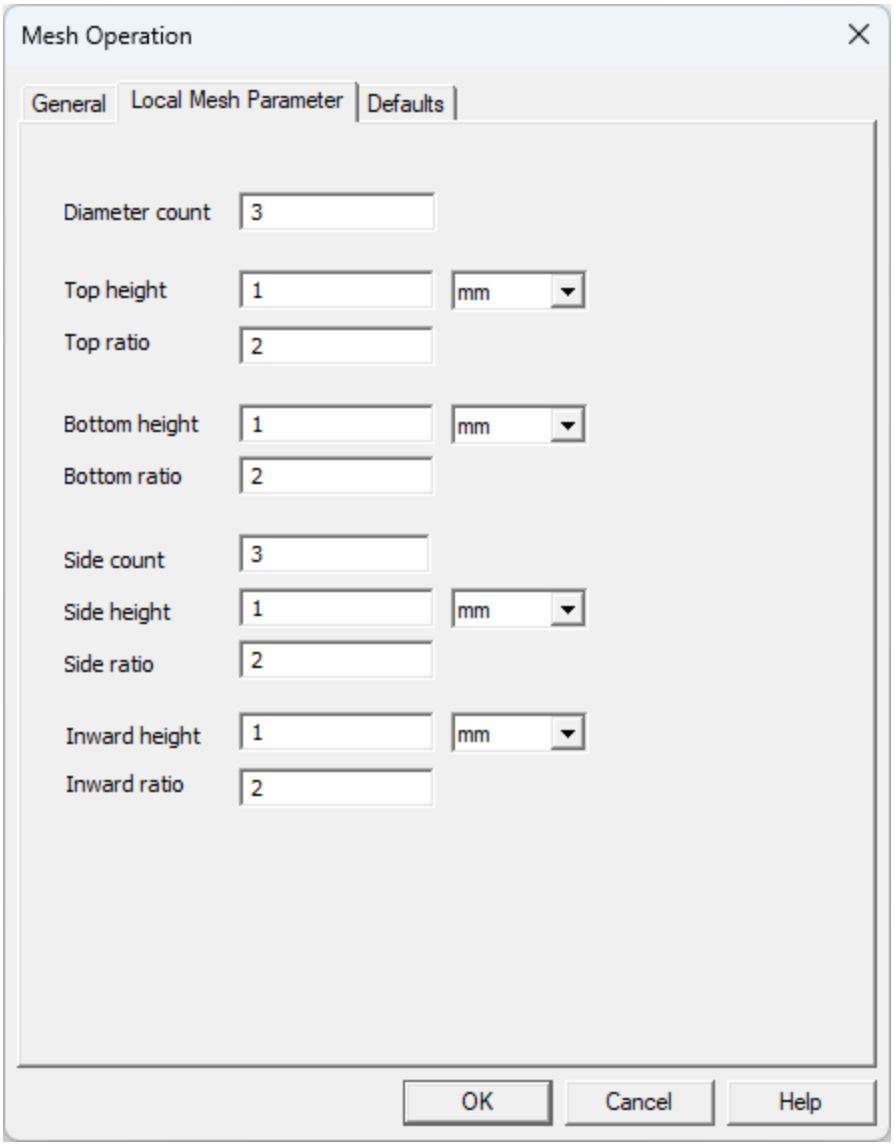


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Count	**X Count** specifies the number of elements along the X axis of the box.
Y Count	**Y Count** specifies the number of elements along the Y axis of the box.
Z Count	**Z Count** specifies the number of elements along the Z axis of the box.
Inward Height	**Inward Height** specifies the height of the first element on the surface of an object in the direction normal to that surface, pointing inside of the object.
Inward Ratio	**Inward Ratio** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the object, on the inside of the object. Together with the initial element height, this parameter controls mesh grading and density in the region near the surface of a modeling object, on the inside of the object.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder Object - Object Selection Mode



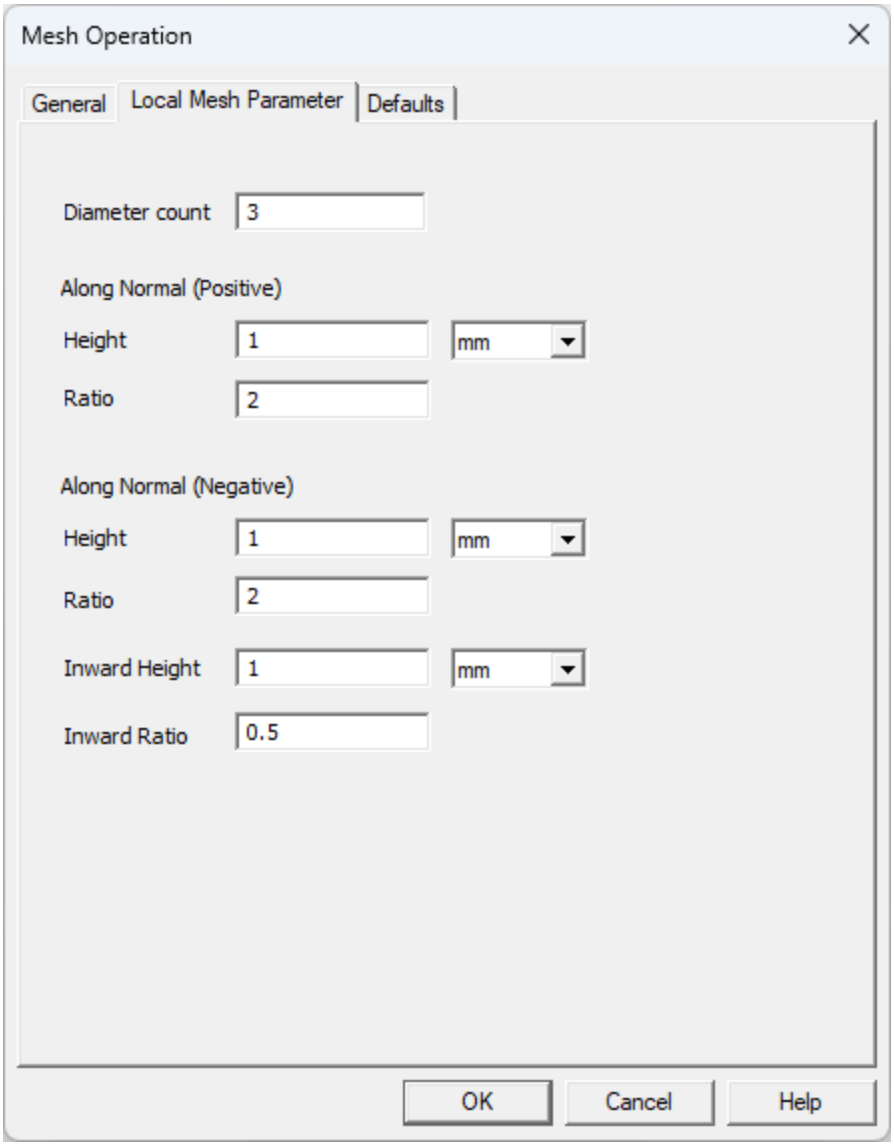
Diameter count	**Diameter count** specifies the number of elements along the diameter of the cylinder.
Top height	**Top height** specifies the height of the first mesh cell along the top face of the cylinder.
Top ratio	**Top ratio** specifies the ratio of the mesh growth in consecutive cells along the top of the cylinder.
Bottom height	**Bottom height** specifies the height of the first mesh cell along the bottom face of the cylinder.
Bottom ratio	**Bottom ratio** specifies the ratio of the mesh growth in consecutive

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	cells along the bottom of the cylinder.
Side count	**Side count** specifies the number of elements along the sides of the cylinder.
Side height	**Side height** specifies the height of the first mesh cell along the sides of the cylinder.
Side ratio	**Side ratio** specifies the ratio of the mesh growth in consecutive cells along the sides of the cylinder.
Inward height	**Inward height** specifies the initial element height from the edge of the cylinder in the radial direction towards the center of the cylinder. This setting is only applied if **No O-Grids** is disabled in the cylinder's corresponding mesh region.
Inward ratio	**Inward ratio** specifies the element height ratio from the edge of the cylinder in the radial direction towards the center of the cylinder. This setting is only applied if **No O-Grids** is disabled in the cylinder's corresponding mesh region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circle Object - Object Selection Mode



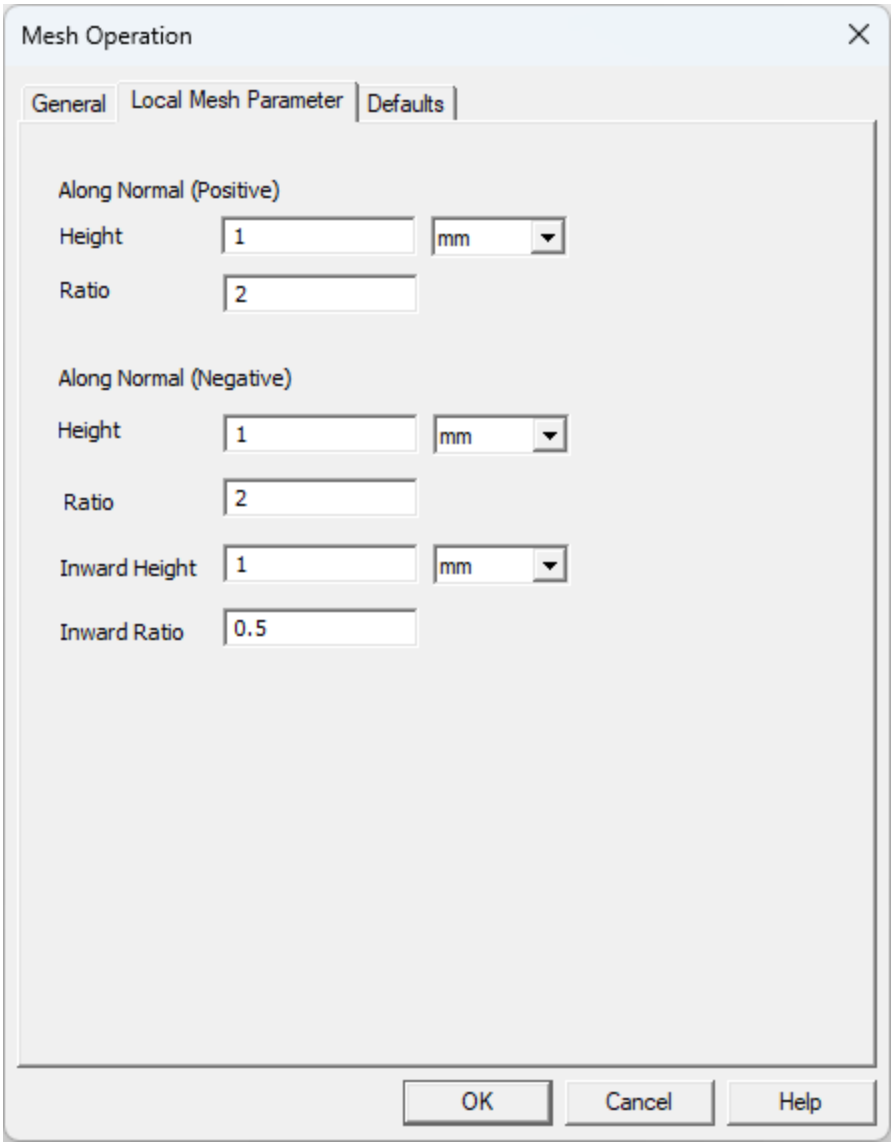
Diameter count	**Diameter count** specifies the number of elements along the diameter of the circle.
Along Normal (Positive)	
Height	**Height** specifies the height of the first mesh cell along the positive direction normal to the surface of the circle face.
Ratio	**Ratio** specifies the ratio of the mesh growth in consecutive cells along the positive direction normal to the surface of the circle face.
Along Normal (Negative)	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Height	**Height** specifies the height of the first mesh cell along the negative direction normal to the surface of the circle face.
Ratio	**Ratio** specifies the ratio of the mesh growth in consecutive cells along the negative direction normal to the surface of the circle face.
Inward Height	**Inward Height** specifies the initial element height from the edge of the circle in the radial direction towards the center of the circle. This setting is only applied if **No O-Grids** is disabled in the circle's corresponding mesh region.
Inward Ratio	**Inward Ratio** specifies the element height ratio from the edge of the circle in the radial direction towards the center of the circle. This setting is only applied if **No O-Grids** is disabled in the circle's corresponding mesh region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Rectangular Sheet Object - Object Selection Mode

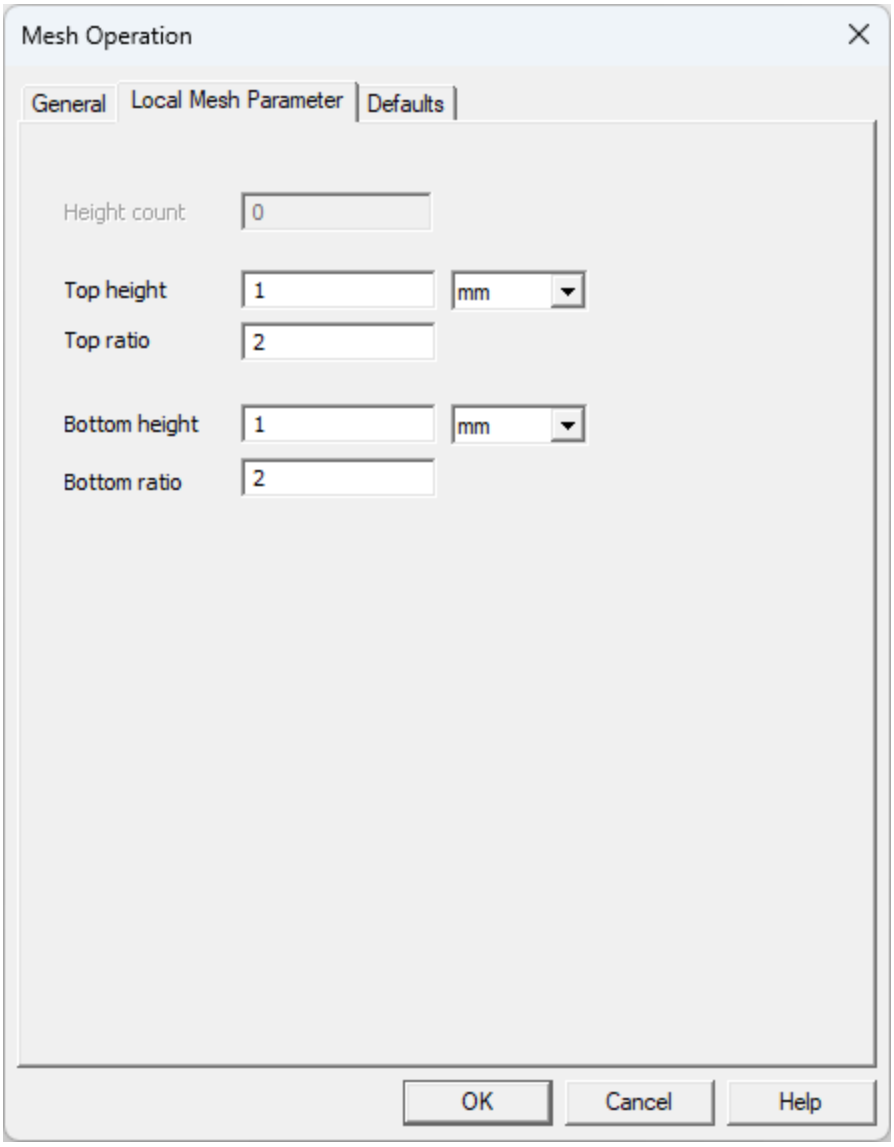


Along Normal (Positive)	
Height	**Height** specifies the height of the first mesh cell along the positive direction normal to the surface of the sheet face.
Ratio	**Ratio** specifies the ratio of the mesh growth in consecutive cells along the positive direction normal to the surface of the sheet face.
Along Normal (Negative)	
Height	**Height** specifies the height of the first mesh cell along the negative direction normal to the surface of the sheet face.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ratio	**Ratio** specifies the ratio of the mesh growth in consecutive cells along the negative direction normal to the surface of the sheet face.
Inward Height	**Inward Height** specifies the initial element height in the inward direction from the edge of the rectangle in both coordinate directions. This setting is only applied if **No O-Grids** is enabled in the rectangle's corresponding mesh region.
Inward Ratio	**Inward Ratio** specifies the element height ratio in the inward direction from the edge of the rectangle in both coordinate directions. This setting is only applied if **No O-Grids** is enabled in the rectangle's corresponding mesh region.

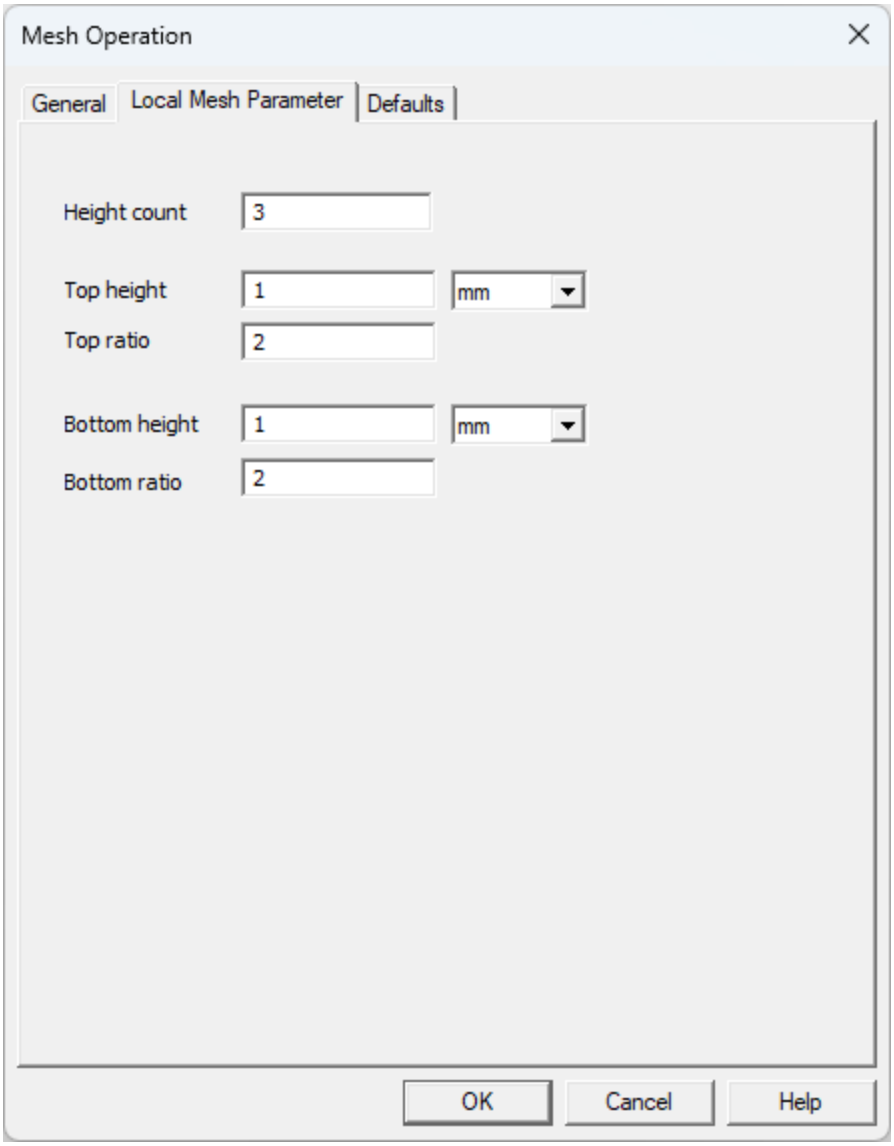
2D Polygon Object - Object Selection Mode



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Top height	**Top height** specifies the height of the first mesh cell along the top face of the polygon.
Top ratio	**Top ratio** specifies the ratio of the mesh growth in consecutive cells along the top of the polygon.
Bottom height	**Bottom height** specifies the height of the first mesh cell along the bottom face of the polygon.
Bottom ratio	**Bottom ratio** specifies the ratio of the mesh growth in consecutive cells along the bottom of the polygon.

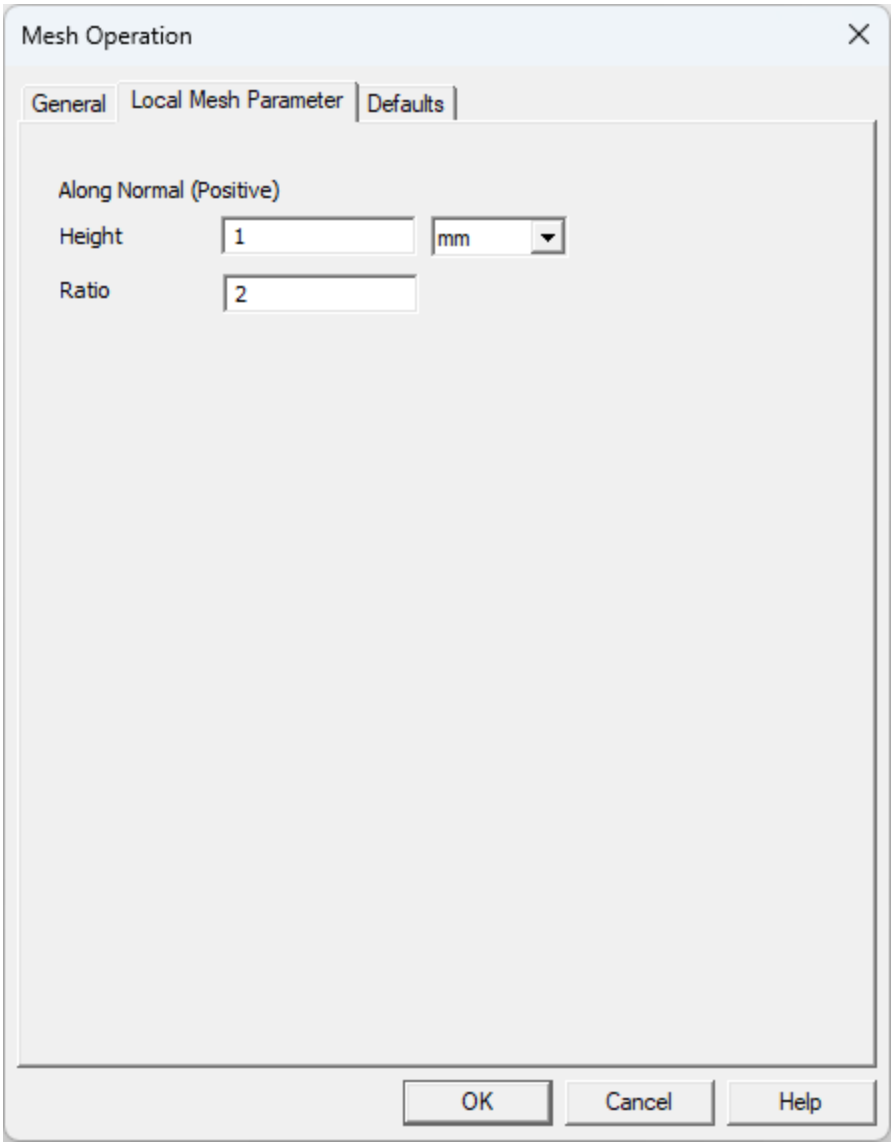
3D Polygon Object - Object Selection Mode



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Height count	**Height count** specifies the number of elements along the height of the polygon.
Top height	**Top height** specifies the height of the first mesh cell along the top face of the polygon.
Top ratio	**Top ratio** specifies the ratio of the mesh growth in consecutive cells along the top of the polygon.
Bottom height	**Bottom height** specifies the height of the first mesh cell along the bottom face of the polygon.
Bottom ratio	**Bottom ratio** specifies the ratio of the mesh growth in consecutive cells along the bottom of the polygon.

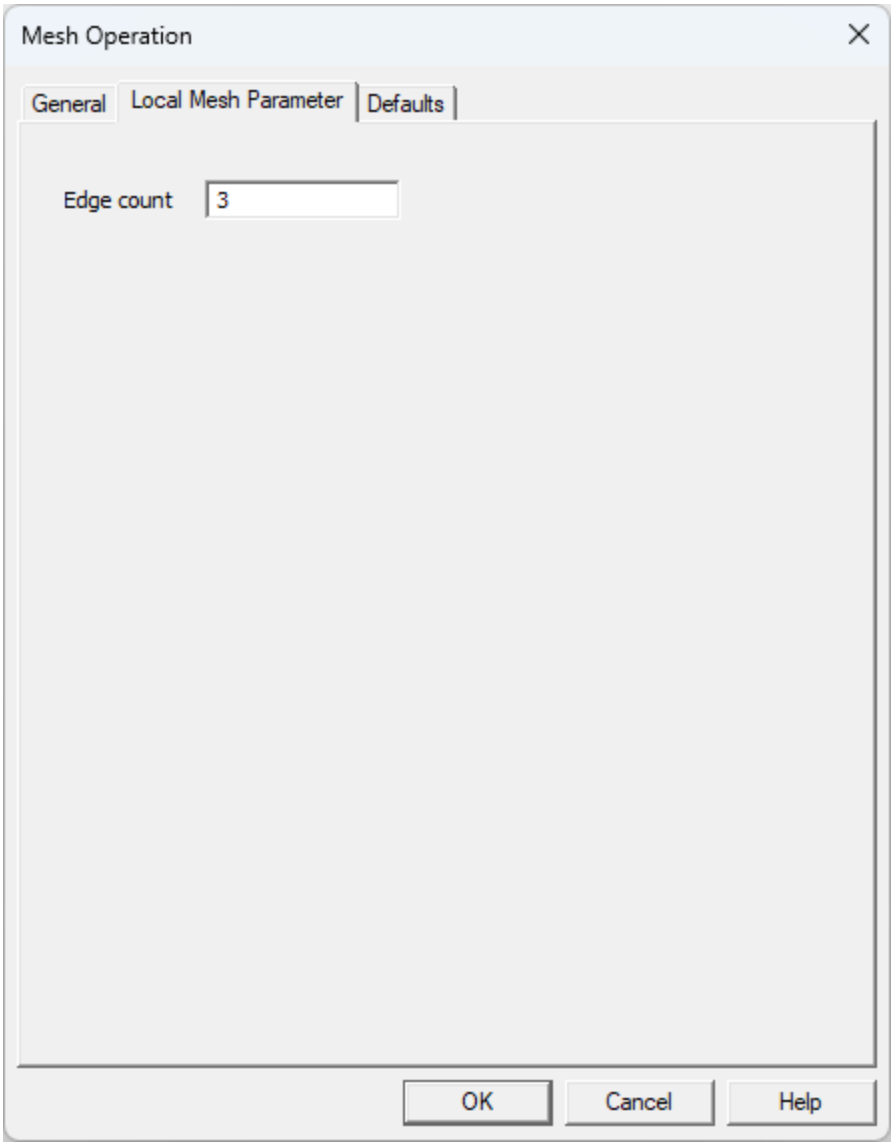
Object Face - Face Selection Mode



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Along Normal (Positive)	
Height	**Height** specifies the height of the first mesh cell along the positive direction normal to the surface of the face.
Ratio	**Ratio** specifies the ratio of the mesh growth in consecutive cells along the positive direction normal to the surface of the face.

Object Edge - Edge Selection Mode

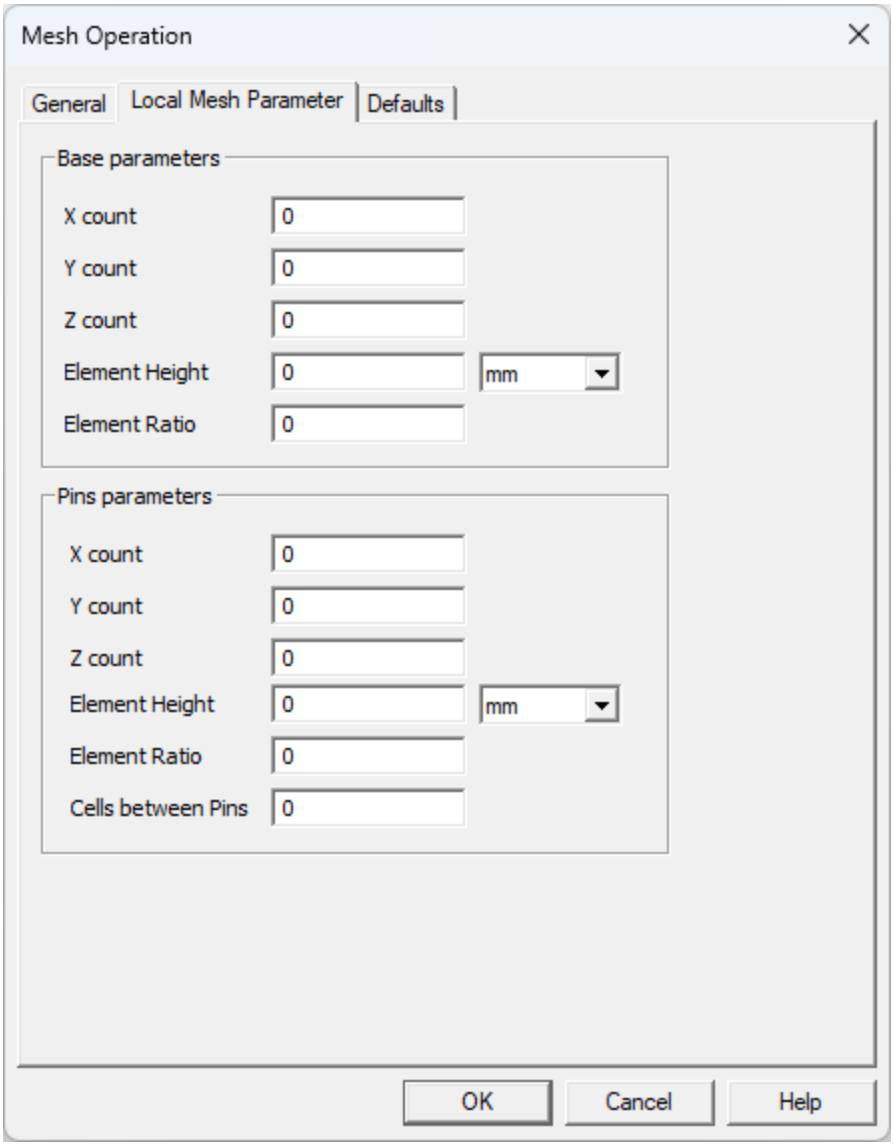


Edge Count	**Edge Count** specifies the number of elements along the edge.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Parameters for Heat Sinks

To assign a mesh operation with local mesh parameters on a heat sink component, select the heat sink instance in the History tree and assign a mesh operation. On the **General** tab, enable **Local mesh parameters** and click the **Local Mesh Parameter** tab.



Base Parameters	
X Count	**X Count** specifies the number of elements along the X axis of the base.
Y Count	**Y Count** specifies the number of elements along the Y axis of the base.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z Count	**Z Count** specifies the number of elements along the Z axis of the base.
Element Height	**Element Height** specifies the first element next to the base of the heat sink, on all sides of the base.
Element Ratio	**Element Ratio** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the base of the heat sink. Together with the height, this parameter controls mesh grading and density near the base of the heat sink.
Pins Parameters	
X Count	**X Count** specifies the number of elements along the X axis of the pins.
Y Count	**Y Count** specifies the number of elements along the Y axis of the pins.
Z Count	**Z Count** specifies the number of elements along the Z axis of the pins.
Element Height	**Element Height** specifies the first element next to the pins of the heat sink, on all sides of the pins.
Element Ratio	**Element Ratio** specifies the ratio of the height of a given element to the height of an adjacent element in the row one level nearer to the pins of the heat sink. Together with the height, this parameter controls mesh grading and density near the pins of the heat sink.
Cells between Pins	**Cells between Pins** specifies the minimum number of cells between each heat sink pin.

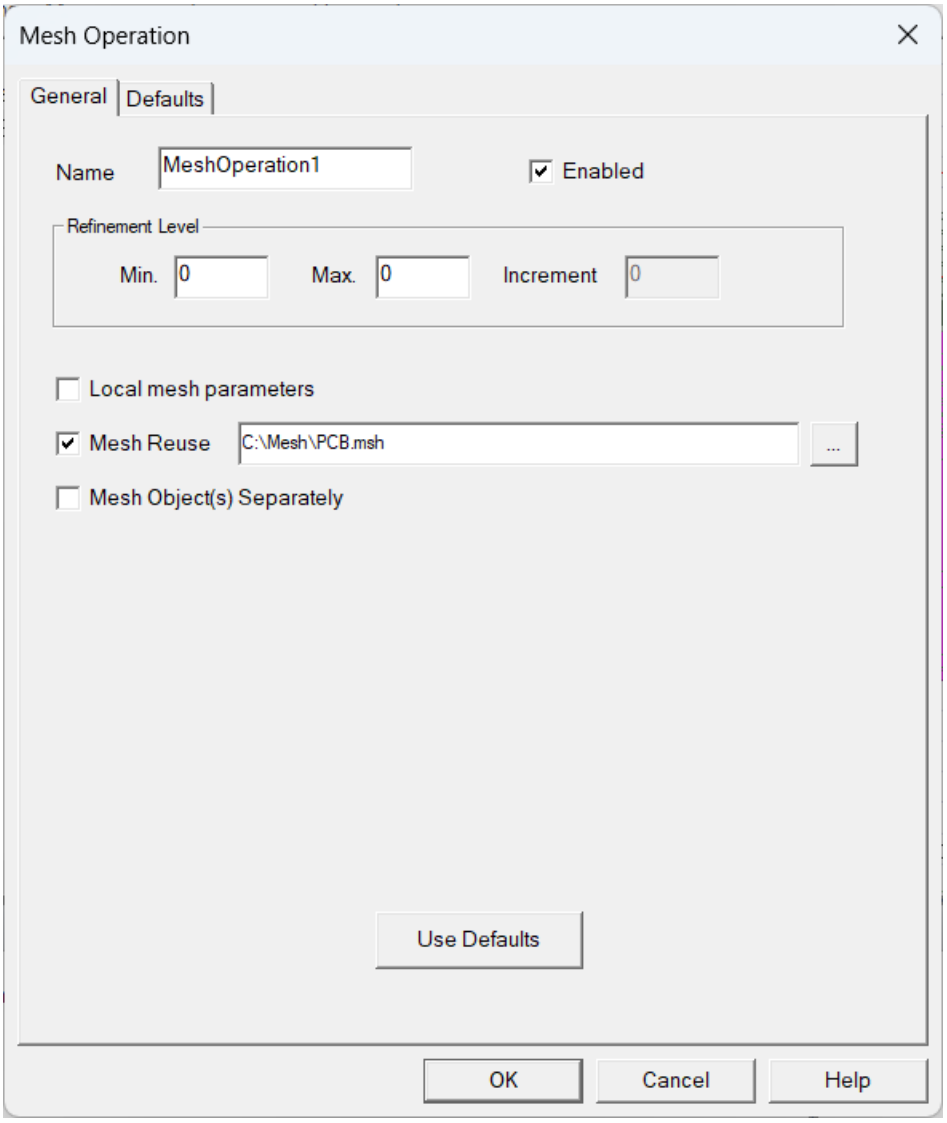
Mesh Reuse

By default, mesh is created for all objects in the model each time it is generated. Using a Mesh Operation, you can reuse mesh for specific objects or shapes if a mesh has been generated for them previously in a different project. The supported mesh file types are the Icepak grid_output file or a Fluent .msh file.

> **Note:** When an object has mesh reuse assigned, Icepak may not display its surface mesh in the **3D Modeler** window.

> **Note:** Mesh reuse is not supported in the following scenarios:
>
> - for lightweight CAD objects
> - when running optimetric parametric trials

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Reassigning a Mesh Region

To reassign a previously assigned mesh setup, do the following in the **Project Manager**:

1. Select the object or objects to which you want to reassign the mesh region.
2. Under **Mesh**, right-click the mesh region you had previously defined.
   A shortcut menu appears.
3. Select **Reassign** from the shortcut menu.

The mesh region is reassigned to the currently selected objects.

To view the objects to which a mesh region is assigned:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Project Manager**, select the mesh region.
2. Right-click the mesh region you had previously completed.
   A shortcut menu appears.
3. Select **Select Assignment** from the shortcut menu.

The objects to which the region is assigned are selected.

Enable or Disable a Mesh Region

To enable or disable a mesh region, do the following from the **Project Manager**:

1. Expand **Mesh** and right-click a mesh region.
   A shortcut menu appears.
2. Select **Enable** or **Disable** from the shortcut menu.

Viewing a List of Mesh Regions

To list all mesh regions for the project, do the following from the project tree:

1. Right-click **Mesh**.
   A shortcut menu appears.
2. Select **List** from the shortcut menu.
   The **Design List** dialog box appears, with the **Mesh** tab displayed.
3. Click **Done** to close this dialog box.

Viewing Mesh Region Properties

To view mesh properties, do the following in the **Project Manager**:

1. Under **Mesh**, right-click the mesh region you had previously created.
   A shortcut menu appears.
2. Select **Properties** from the shortcut menu.

Deleting All Previously Assigned Mesh Regions

To delete previously assigned mesh regions:

- Click **Icepak**>**Mesh**>**Delete All**.

Mesh Region Intersections

When creating mesh regions in Icepak, you must determine which areas within the computational domain require unique mesh settings. When creating mesh regions, you are able to create them in relation to model geometry and other mesh regions in the following ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Intersecting no objects
- Intersecting one or more objects
- Touching the outer region
- Touching one or more mesh regions
- Touching one or more instances of 3D geometry from the inside or outside
- Touching 2D objects

Mesh Region Intersection Limitations

The following are limitations for mesh region intersections:

- Two or more mesh regions can not intersect.
- A conducting plate with **Shell conduction** enabled can not intersect a mesh region boundary.
- Network faces can not intersect a mesh region boundary.

Mesh Region Intersecting No Objects

Mesh regions in Icepak can completely surround object geometry without intersecting or touching it. Figure 1 shows an example of a mesh region surrounding box geometry without intersecting or touching it.

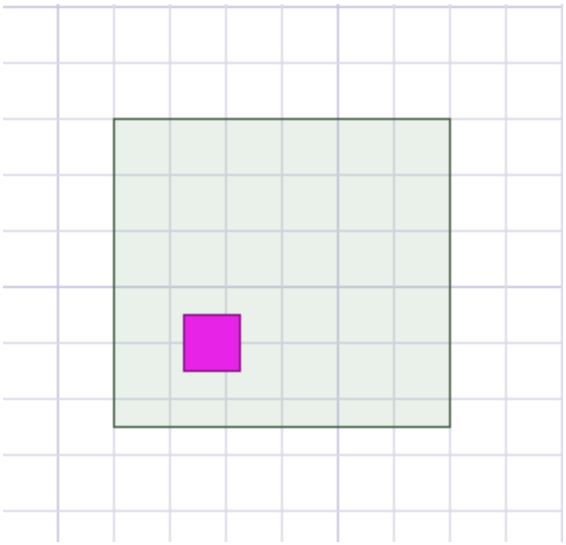


Fig. 1 Mesh Region Intersecting No Objects

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Region Intersecting Objects

Mesh regions in Icepak can intersect object geometry. Figure 1 shows an example of a mesh region intersecting box geometry.

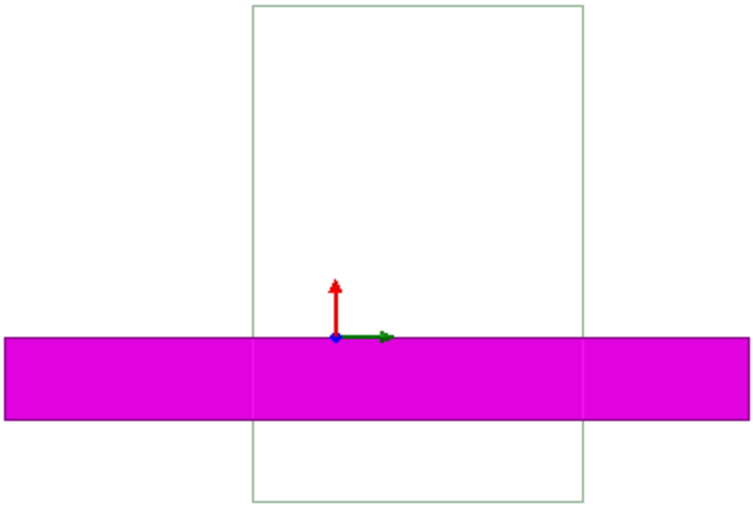


Fig. 1 Mesh Region Intersecting Object Geometry

Mesh Region Intersecting the Outer Region

Mesh regions in Icepak can touch the outer region, which defines the computational domain. Figure 1 shows an example of a mesh region touching the outer region.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

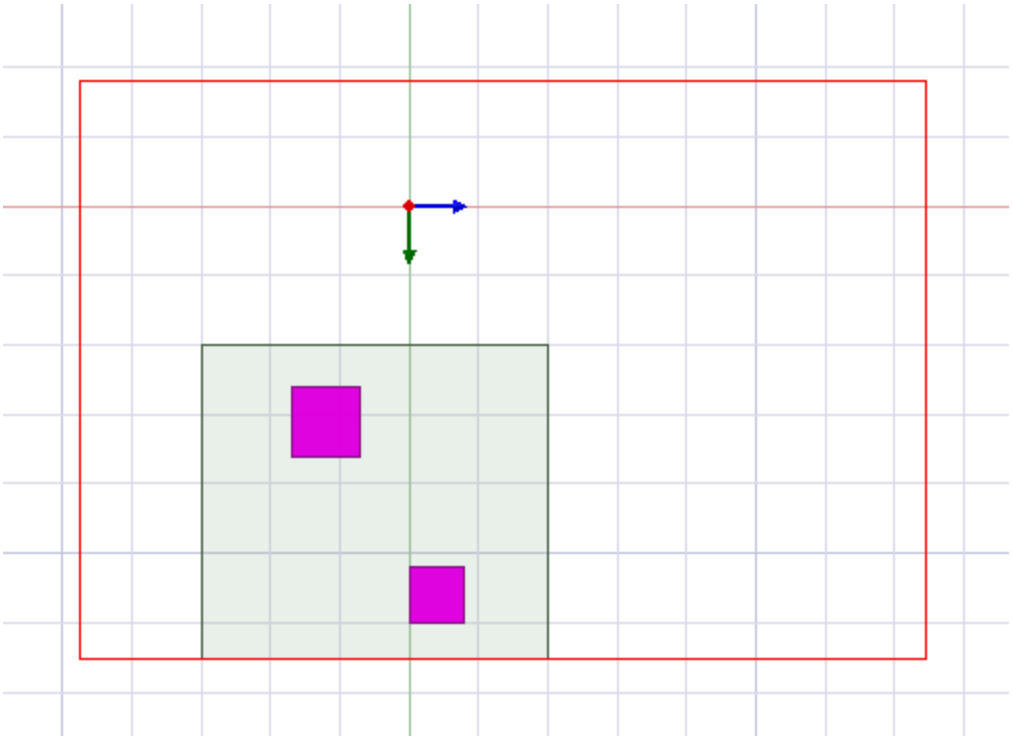


Fig. 1 Mesh Region Touching Outer Region

Mesh Region Touching Other Mesh Regions

Mesh regions in Icepak can touch other mesh regions. Figure 1 shows an example of four mesh regions with each mesh region touching one or more mesh regions. Mesh regions in Icepak can touch other mesh regions including those with no padding.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

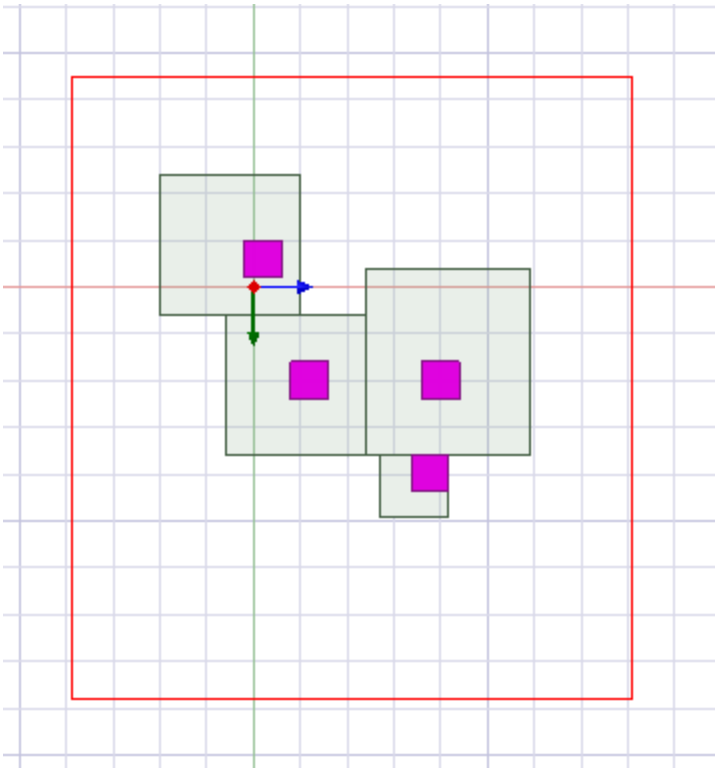


Fig. 1 Mesh Regions Touching One or More Mesh Regions

Mesh Region with No Padding Touching 2D Geometry

Mesh regions with no padding can touch 2D object geometry. Figure 1 shows an example of a mesh region with no padding touching rectangle geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

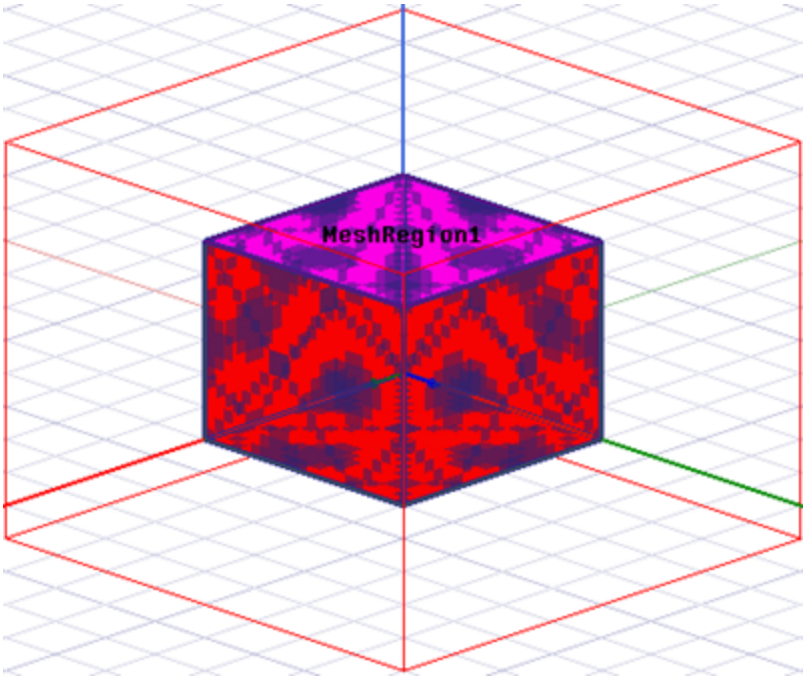


Fig. 1 Mesh Region with No Padding Touching 2D Geometry

Mesh Region with No Padding Touching 3D Geometry

Mesh regions with no padding can touch 3D object geometry. Figure 1 shows an example of a mesh region with no padding touching box geometry.

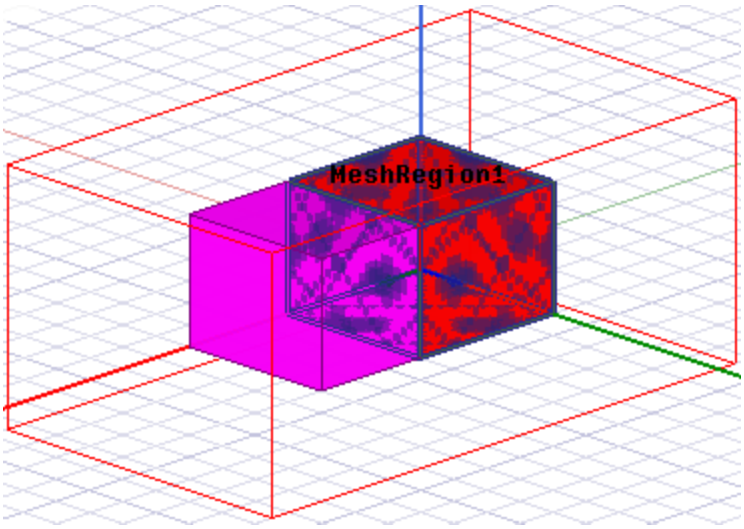


Fig. 1 Mesh Region with No Padding Touching 3D Geometry

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Priority

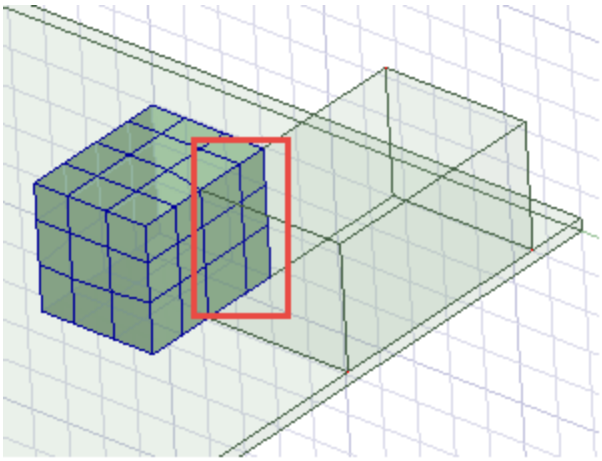
Nested Mesh Regions

In Icepak, you can create mesh regions within larger mesh regions. Mesh regions can contain multiple nested, smaller mesh regions, each with differing mesh settings. The global mesh region is the outermost region, which is assigned the lowest mesh priority. The innermost mesh region is assigned the highest mesh priority. Mesh priority escalates from the innermost mesh region to the outermost region.

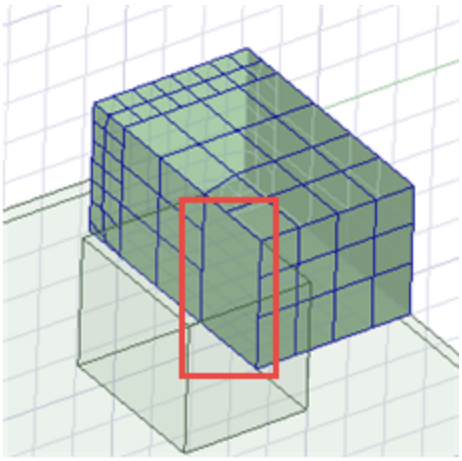
Mesh settings assigned to an outer mesh region are applied to the mesh in the region with the exception the space occupied by any inner mesh regions. Mesh priority applies to three-dimensional and two-dimensional regions.

Overlapping Mesh

Two-dimensional objects and faces of three-dimensional objects can have overlapping surfaces, and three-dimensional objects can have overlapping volumes. When displaying mesh on overlapping objects, the overlapping mesh area is displayed on the object with the higher priority as in the figures below. Note the overlapping areas in the red boxes. Because Box 1 is a higher priority object the overlapping area's mesh is displayed with it.

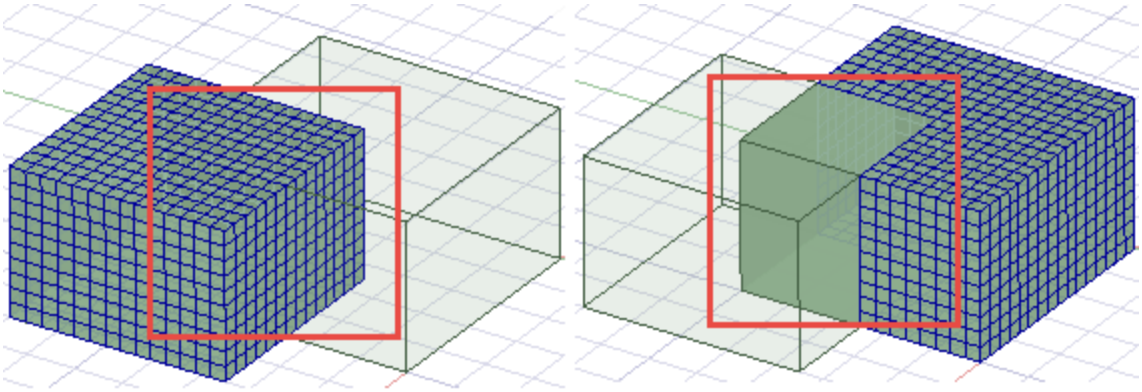


Box 1 (Higher Priority)



Box 2 (Lower Priority)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

When displaying post-processing data on objects with overlapping surfaces, the overlapping surface area is included in the calculation for both objects regardless of priority.

Generating the Mesh

After you have defined global mesh settings and assigned locally refined mesh regions, you can generate the mesh.

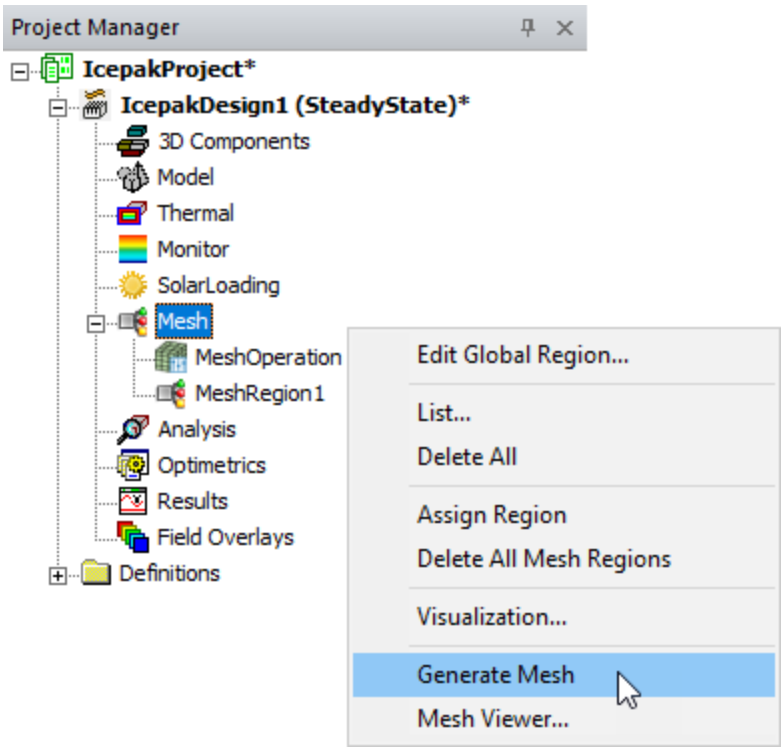
To generate the mesh:

Note:

If you have not yet saved the project, Icepak will prompt you to do so before generating the mesh.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Project Manager**, right-click **Mesh** and select **Generate Mesh**.



2. You can monitor the progress of the mesh generation in the **Progress** window. When mesh generation is complete, a message is displayed in the **Message Manager** window.

Note:

Before meshing, Icepak checks the design for the following issues:

- Mesh regions intersecting one another
- Sheets assigned a thin conducting plate boundary condition intersecting a mesh region
- Network faces blocked by a hollow object (an object with **Solve Inside** disabled)

When **Skip Intersection Checks** in the Icepak Design Settings is disabled, the solver checks the design for the following issue:

- Polygonal or inclined geometry assigned a boundary condition that intersect a mesh region

If a mesh region intersects another, meshing is aborted with an error message. If any other issues are found, a warning message is displayed in the **Message Manager**, and the analysis proceeds.

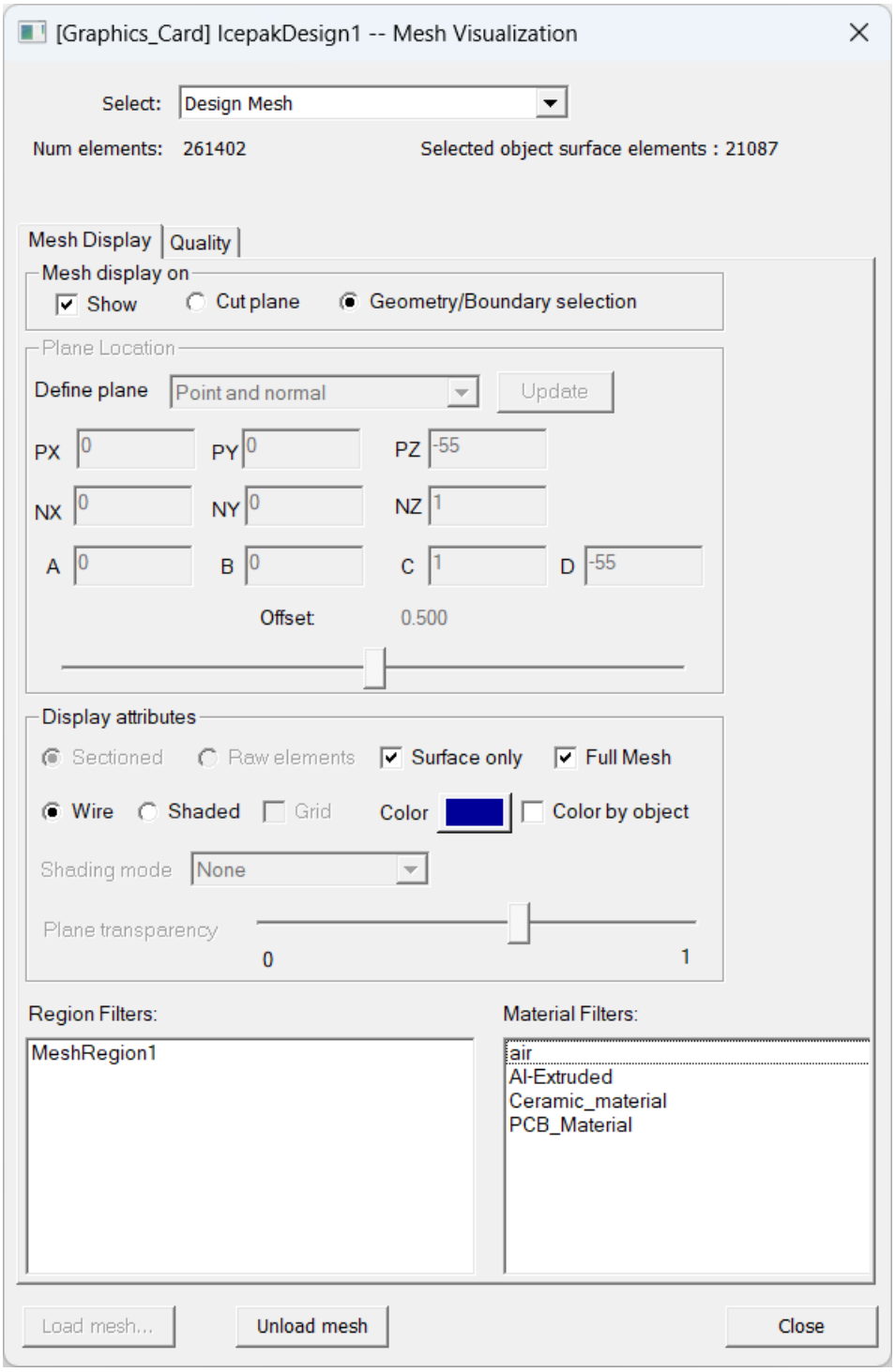
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displaying the Mesh

Once you have generated a mesh, you can examine it to determine if it meets your needs. Icepak provides tools for viewing the mesh on individual objects, as well as on cross-sectional planes that extend throughout the cabinet.

> **Note:**
>
> Scripting does not record steps performed in the **Mesh Visualization** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select	From the **Select** drop-down list, select the mesh to display in the **3D Modeler** window. You can select the **Design Mesh** or another solution setup from the design. The design mesh is the mesh

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

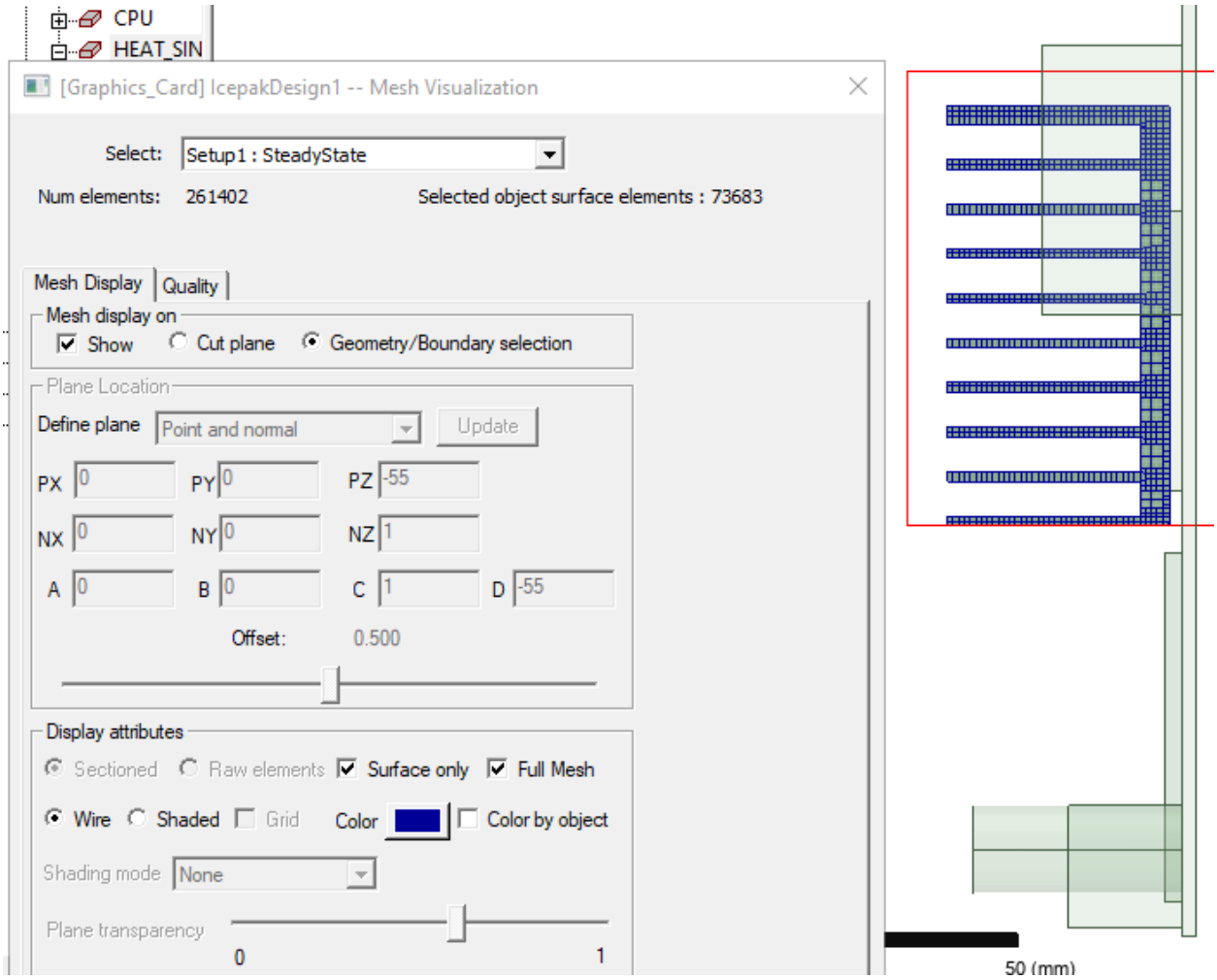
	generated based on the current design's geometry and mesh settings. Other solution setups can contain imported mesh from linked designs. See Adding a Mesh Linked Solution Setup and Importing Mesh for more information.
Num elements	**Num elements** displays the number of mesh elements that exist in the global mesh region. Fine mesh settings result in higher element counts.
Selected object surface elements	When **Geometry/Boundary selection** is enabled, **Selected object surface elements** displays the number of surface mesh elements of the currently selected objects.
Mesh display on	
Show	**Show** displays the mesh.
Cut plane	**Cut plane** displays the mesh on a cross-sectional plane.
Geometry/Boundary selection	**Geometry/Boundary selection** displays the mesh on object geometry selected in the **3D Modeler** window or history tree or a thermal boundary condition selected in the **Project Manager**. When selecting geometry in the **3D Modeler** window, you can select and display mesh on objects or faces.
Plane Location	
X, Y, Z plane through center	**X plane through center**, **Y plane through center**, or **Z plane through center** specifies a plane cut through the center of the model that is aligned with the Cartesian axis.
Select a face/plane	**Select a face/plane** displays the mesh on a plane based on your selection of a geometry face or plane in the **3D Modeler** window or history tree.
Point and normal	**Point and normal** specifies a point on the plane and the normal direction to the plane. Enter coordinates (**PX**, **PY**, and **PZ**) to specify the point. Enter a vector defining the direction normal to the plane (**NX**, **NY**, and **NZ**). For example, entering NX = 1, NY = 0, and NZ = 0 for the vector defines normal pointing in the *x* direction.
Coeffs (Ax + By + Cz = D)	**Coeffs (Ax + By + Cz = D)** specifies an equation that defines the plane. Enter the coefficients **A**, **B**, **C**, and **D** for the equation.
Horizontal - screen select	**Horizontal - screen select** displays a horizontal plane cut of the mesh at a selected point in the **3D Modeler** window.
Vertical - screen select	**Vertical - screen select** displays a vertical plane cut of the mesh at a selected point in the **3D Modeler** window.
Display Attributes	
Sectioned	**Sectioned** displays a two-dimensional cross-section of the mesh

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	when viewing a plane cut.
Raw elements	**Raw elements** displays a three-dimensional plane cut of the mesh. When **Surface only** is selected, only the outer surface of the plane is displayed.
Surface only	**Surface only** displays only the surface mesh.
Full Mesh	**Full Mesh** displays the full mesh of selected geometry or boundary conditions, including areas overlapping with other geometry. See "Reduced Mesh and Full Mesh" on the facing page.
Wire	**Wire** displays the generated surface mesh lines.
Shaded	**Shaded** displays the generated surface mesh filled with the color for the selection in the drop-down menu. If you select Shaded, the shading mode is enabled. This can be None, Flat, or Smooth.
Grid	**Grid** displays wire mesh lines over the **Shaded** mesh display.
Color	**Color** defines the color in which the mesh is displayed.
Color by object	**Color by object** displays object mesh faces and/or cells using the object color defined in its **Properties** window. Otherwise, the mesh faces will be shown using the **Color** setting.
Plane transparency	**Plane transparency** controls the mesh plane transparency when Shaded is selected. Move the slider bar to display different levels of transparency.
Geometry Filters (Cut plane)	**Geometry Filters**displays mesh only on the selected geometry or mesh regions. Press Ctrl and click to select multiple pieces of geometry and mesh regions.
Region Filters (Geometry/Boundary selection)	**Region Filters**displays mesh only on the cut plane through selected geometry. Press Ctrl and click to select multiple pieces of geometry.
Material Filters (Cut plane and Geometry/Boundary selection)	**Material Filters** displays mesh only on the geometry assigned the selected materials. Press Ctrl and click to select multiple materials.
Load Mesh	**Load Mesh** displays the mesh in the **3D Modeler** window.
Unload Mesh	**Unload Mesh** hides the mesh in the **3D Modeler** window.

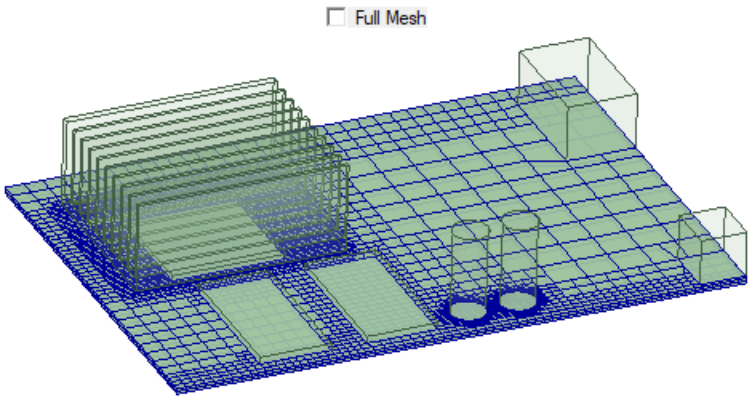
The following image displays the mesh for the selected object, which is the HEAT_SINK selected in the history tree.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

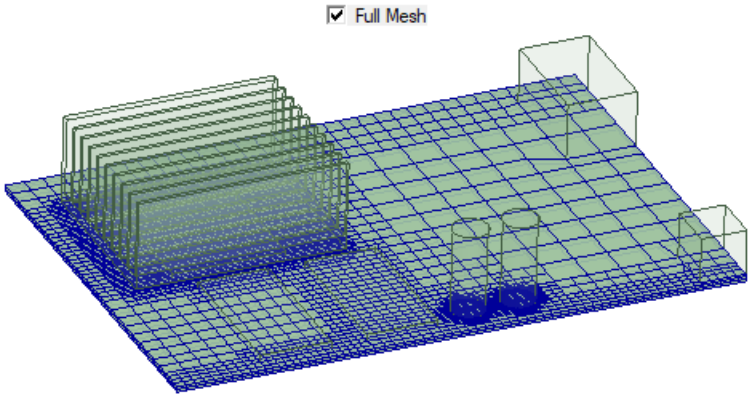


Reduced Mesh and Full Mesh

The following images illustrate how displaying reduced mesh compares to full mesh.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To display the mesh:

> **Note:**
>
> To inspect the mesh fully, use various combinations of the options available in the procedure below.

1. Do one of the following:
   - In the **Project Manager**, right-click **Field Overlays** and select **Mesh Viewer**.
   - Under **Analysis**, right-click a solution setup and select **Mesh Viewer** to view the mesh specific to that setup.

     > **Note:**
     >
     > If you modify the geometry of an object, the solution is invalidated, and the mesh is unloaded.

2. From the **Select** drop-down list, select the mesh to display in the **3D Modeler** window.

   > **Note:**
   >
   > If you select **Design Mesh** or a solution setup that is not linked to another design's mesh, the design mesh is displayed. If you select a solution setup that is linked to another design's mesh, the imported mesh is displayed.

3. In the **Mesh Visualization** dialog box under **Mesh display on**, select **Show** and then **Cut plane** or **Geometry/Boundary selection**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Define settings under **Display attributes** to customize the mesh display.

4. Under **Plane Location**, select an option from the **Set Position** drop-down menu:
   - **X**, **Y**, or **Z plane through center**: Use the slider bar to move the plane cut.
   - **Point and normal**: To move the plane cut, use the slider bar or enter values in the coordinate text entry fields and click **Update**.
   - **Select a face/plane**: Select a face or plane.
   - **Coeffs (Ax + By + Cz = D)**: Enter values in the **A**, **B**, **C**, and **D** text entry fields and click **Update**. Then, use the slider bar to move the plane cut.
   - **Horizontal - screen select**: Select a point in the **3D Modeler** window.
   - **Vertical - screen select**: Select a point in the **3D Modeler** window.

> **Note:**
>
> If needed, select one or more pieces of geometry, mesh regions, or materials from the filters lists to display mesh only on selected objects.

5. Use the **3D Modeler** window viewing tools to inspect the mesh from various angles.

Checking the Mesh

Icepak provides three ways to check the quality of your mesh: face alignment, element volume, and skewness.

> **Note:**
>
> Face alignment, element volume, and skewness results are shown for 3D elements or cell elements.

> **Note:**
>
> When a mesh is loaded in the **Mesh Viewer** and you click the **Quality** tab, Icepak calculates the mesh quality results (face alignment, element volume, and skewness). The calculation will not occur again unless changes to the model require the mesh to be regenerated.

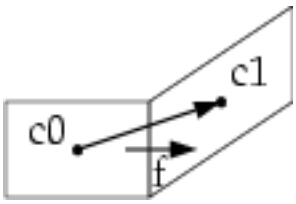
Checking the Face Alignment

Face alignment is a measure of mesh quality defined by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{face alignment index} = \overrightarrow{c_0 c_1} \cdot \vec{f}$$

where c_0 and c_1 are the centroids of two adjacent elements, and $\vec{f}$ is the normal vector to the face between the two elements, as shown in the 2D example below.



Adjacent mesh faces that are not aligned can result in long, narrow elements. A value of 1 indicates perfect alignment. Values less than 0.15 indicate a severely distorted mesh.

To check face alignment for the currently loaded mesh:

1. From the **Icepak**>**Fields** menu, select **Mesh Viewer**.
2. Click on the **Quality** tab to show the mesh diagnostic tools.
3. Select the **Face alignment** option. Icepak displays a histogram of the face alignment, as shown in Figure 1.

> **Note:**
>
> By default, **All** is selected, which displays values for mesh within the entire global mesh region. Select **Geometry selection** to display values for geometry selected in the **3D Modeler** window or history tree, including non-model boxes representing mesh regions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

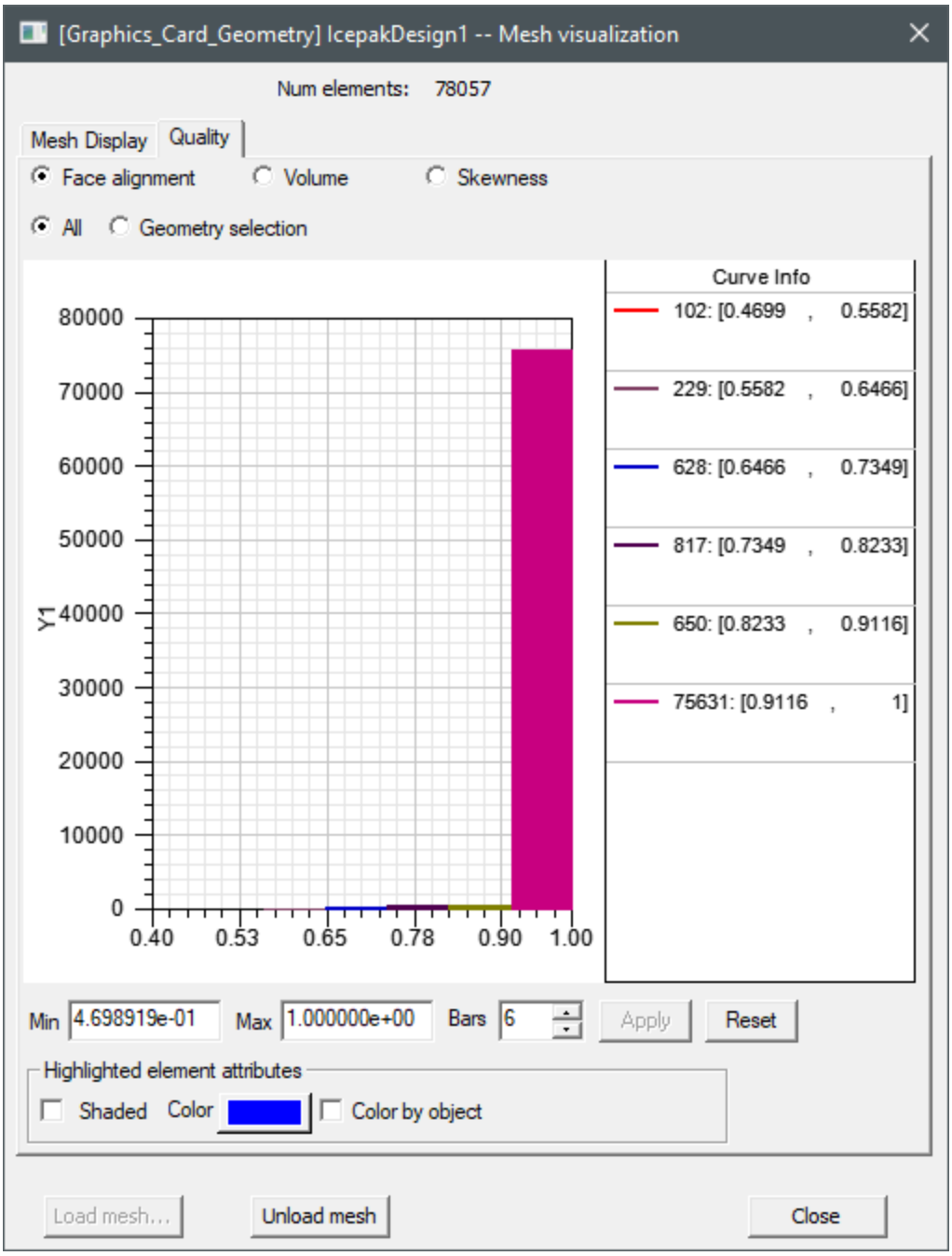


Fig. 1 Face alignment Histogram

4. To modify the range of face alignments viewed, enter a new value in the **Min** or **Max** field and then press the Enter key on your keyboard or click **Apply** to update the histogram. To modify the number of bars in the histogram, enter a new value in the **Bar** field and click **Apply**.Click **Reset** to rest to the range.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To view the elements of the mesh within a particular range of face alignments, click on a bar in the histogram or a range in the **Curve Info** legend. Icepak displays the elements in the selected range in the **3D Modeler** window. Select the **Shaded** option if you want to view these elements with solid shading. Click **Color** and select a desired color if needed.

Checking the Element Volume

For a mesh to be valid for the solver, there should be no negative or zero volume elements in the mesh. It is therefore recommended to check the minimum element volumes in your mesh.

To check element volume for the currently loaded mesh:

1. From the **Icepak**>**Fields** menu, select **Mesh Viewer**.
2. Click on the **Quality** tab to show the mesh diagnostic tools.
3. Select the **Volume** option. Icepak displays a histogram of the element volume, as shown in Figure 1.

> **Note:**
>
> By default, **All** is selected, which displays values for mesh within the entire global mesh region. Select **Geometry selection** to display values for geometry selected in the **3D Modeler** window or history tree, including non-model boxes representing mesh regions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

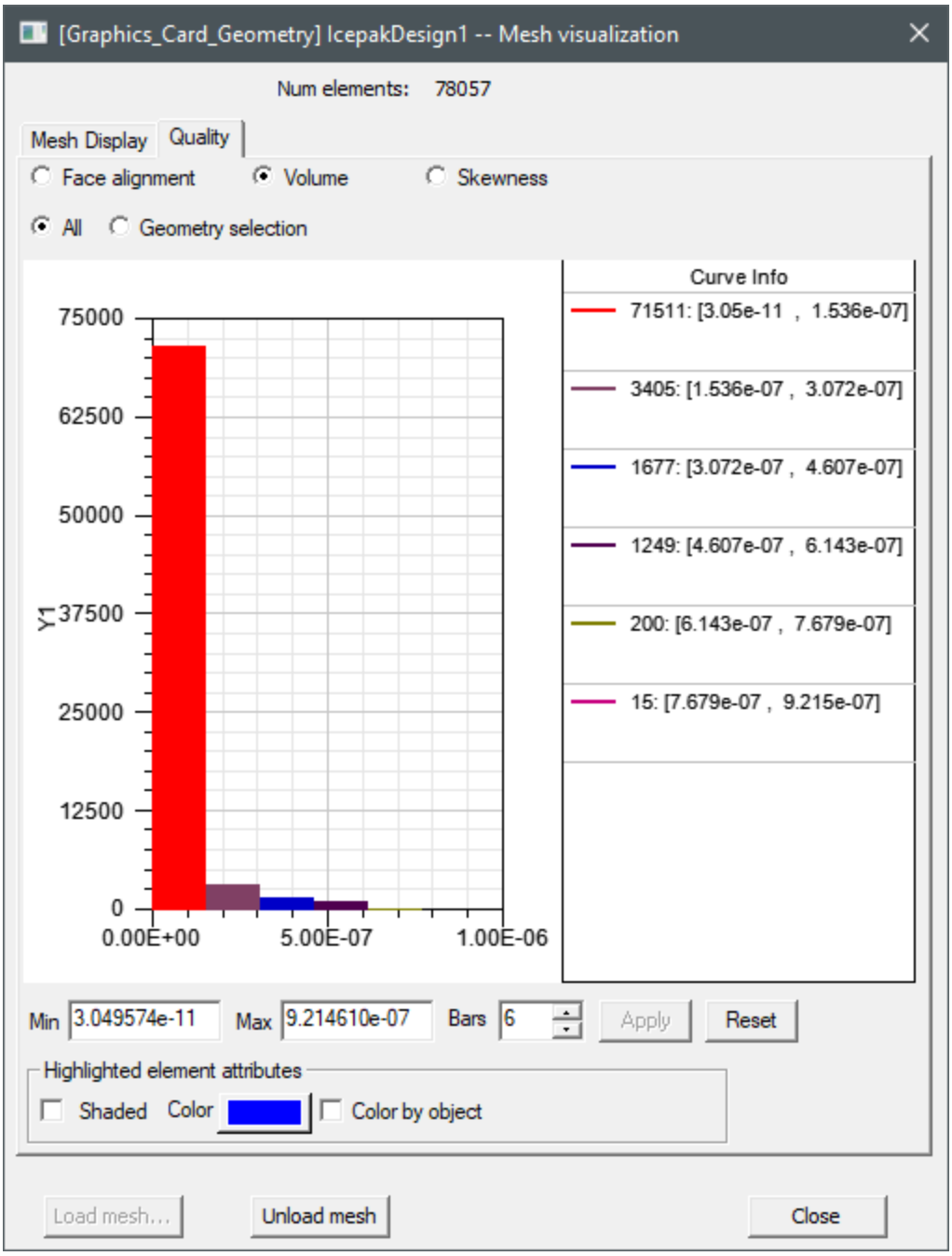


Fig. 1 Element Volume Histogram

4. To modify the range of element volumes viewed, enter a new value in the **Min** or **Max** field and then press the Enter key on your keyboard or click **Apply** to update the histogram. To modify the number of bars in the histogram, enter a new value in the **Bar** field and click **Apply**.Click **Reset** to rest to the range.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To view the elements of the mesh within a particular range of element volumes, click on a bar in the histogram or a range in the **Curve Info** legend. Icepak displays the elements in the selected range in the **3D Modeler** window. Select the **Shaded** option if you want to view these elements with solid shading. Click **Color** and select a desired color if needed.

Checking the Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (that is, equilateral or equiangular) a face or cell is (see Figure 1).

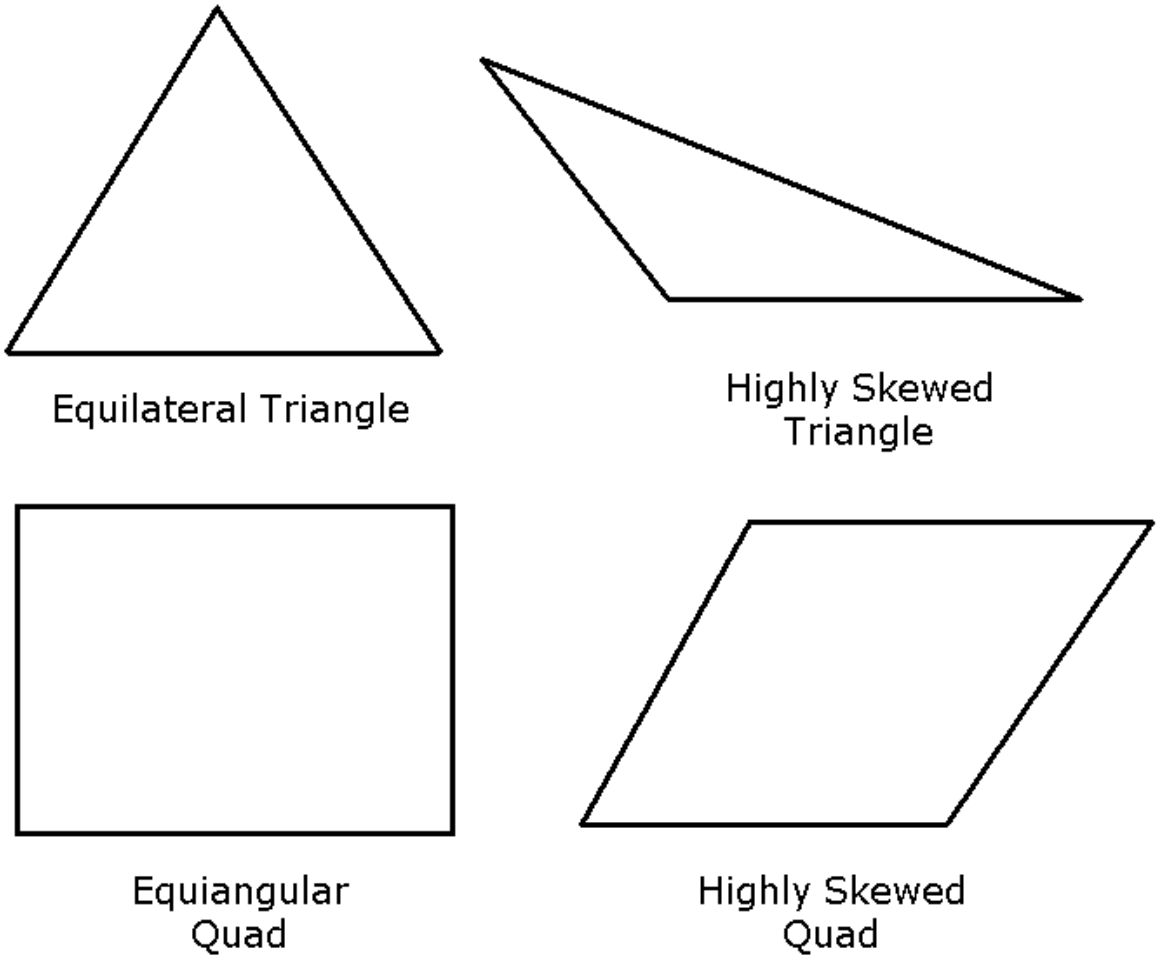


Fig. 1 Ideal and Skewed Triangles and Quadrilaterals

The following table lists the range of skewness values and the corresponding cell quality for equilateral-volume-based skewness and skewness-normalized equiangular skewness.

Value of Skewness	Cell Quality
0	degenerate

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

< 0.02	bad (sliver)
0.02 - 0.25	poor
0.25 - 0.5	fair
0.5 - 0.75	good
0.75 - 1	excellent
1	equilateral

According to the definition of skewness, a value of 1 indicates an equilateral cell (best) and a value of 0 indicates a completely degenerate cell (worst). Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D).

Highly skewed faces and cells are unacceptable because the equations being solved assume that the cells are relatively equilateral/equiangular.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to triangles).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, for example, pyramids and prisms.

The skewness method is equiangular for every type.

Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$Skewness = \frac{CellSize}{OptimalCellSize}$$

where the optimal cell size is the size of an equilateral cell with the same circumradius.

The presence of cells that are fair or worse indicates poor boundary node placement. You should try to improve your boundary mesh as much as possible, because the quality of the overall mesh can be no better than that of the boundary mesh.

In 3D, most cells should be good or better, but a small percentage will generally be in the fair range and there are usually even a few poor cells.

Skewness-Normalized Equiangular

In the normalized angle deviation method, skewness is defined (in general) as

$$1.0 - \max\left[\frac{\Theta_{\max} - \Theta_e}{180 - \theta_e}, \frac{\Theta_e - \Theta_{\min}}{\Theta_e}\right]$$

where

$\Theta_{\max}$ = largest angle in the face or cell

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

θ_{min} = smallest angle in the face or cell

θ_e = angle for an equiangular face or cell (for example, 60 for a triangle, 90 for a square)

Check the Skewness

To check skewness for the currently loaded mesh:

1. From the **Icepak**>**Fields** menu, select **Mesh Viewer**.
2. Click on the **Quality** tab to show the mesh diagnostic tools.
3. Select the **Skewness** option. Icepak displays a histogram of the skewness, as shown in Figure 2.

> **Note:**
>
> By default, **All** is selected, which displays values for mesh within the entire global mesh region. Select **Geometry selection** to display values for geometry selected in the **3D Modeler** window or history tree, including non-model boxes representing mesh regions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

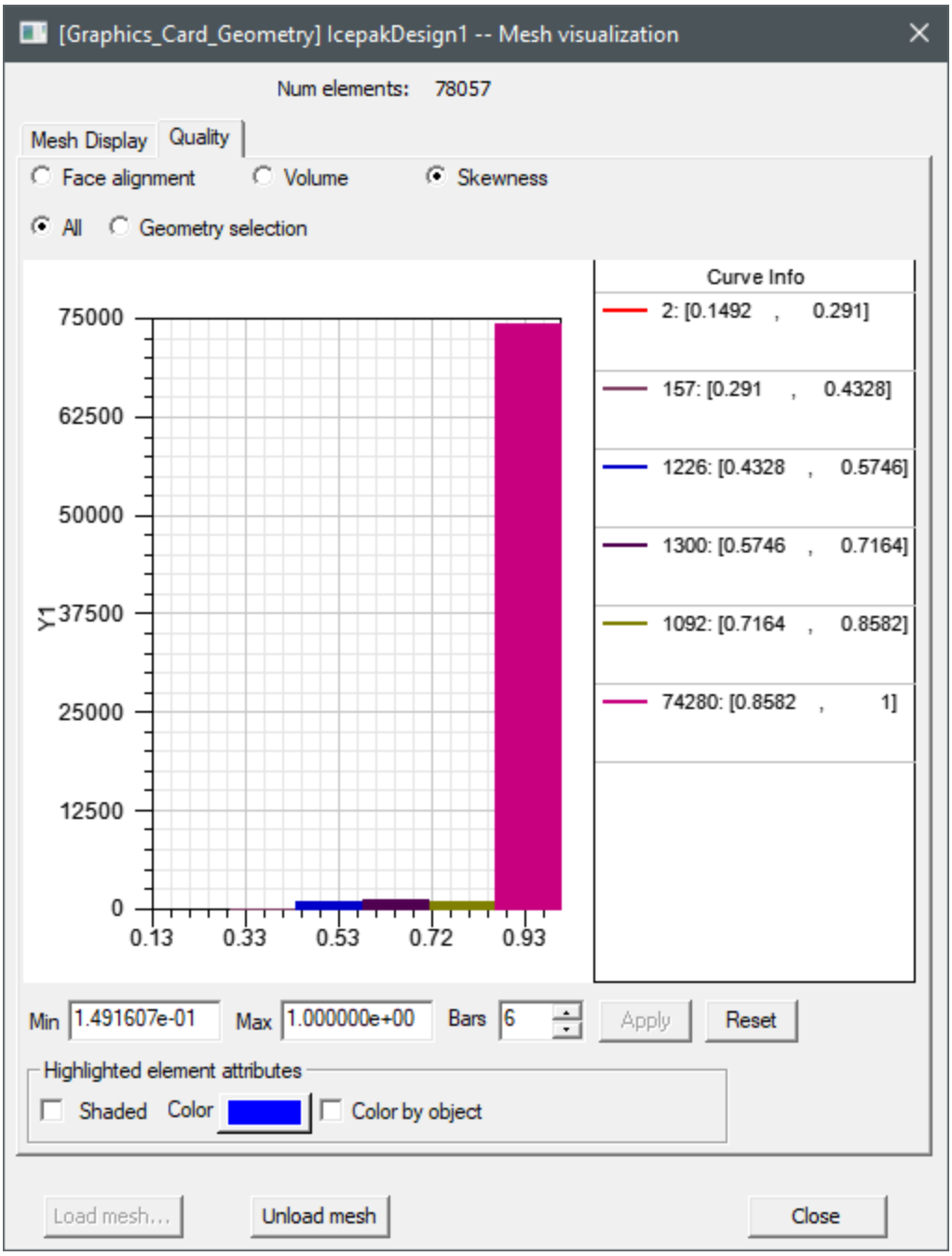


Fig. 2 Skewness Histogram

4. To modify the range of skewnesses viewed, enter a new value in the **Min** or **Max** field and then press the Enter key on your keyboard or click **Apply** to update the histogram. To modify the number of bars in the histogram, enter a new value in the **Bar** field and click **Apply**.Click **Reset** to rest to the range.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To view the elements of the mesh within a particular range of skewnesses, click on a bar in the histogram or a range in the **Curve Info** legend. Icepak displays the elements in the selected range in the **3D Modeler** window. Select the **Shaded** option if you want to view these elements with solid shading. Click **Color** and select a desired color if needed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13 - Assigning Monitors

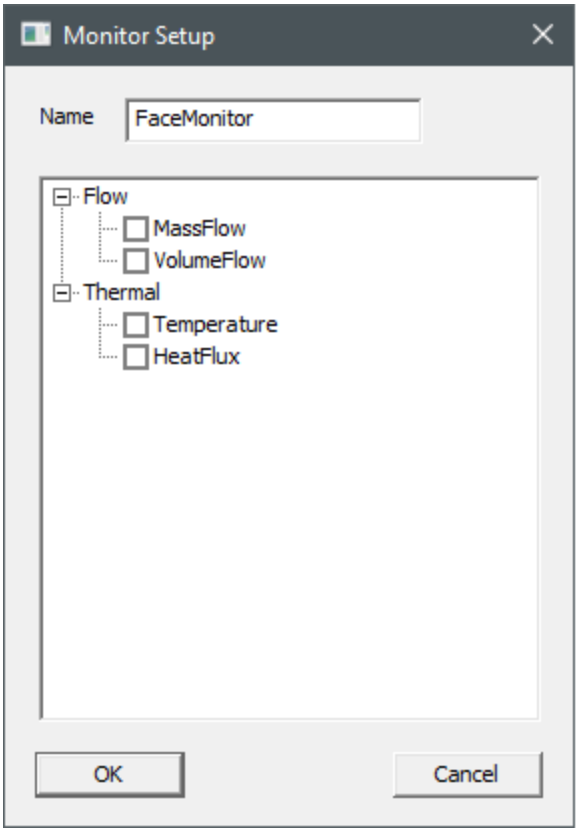
Solution monitors are variables that display selected data during the calculation while running a solution. Solution monitors can be assigned to geometry in the **3D Modeler** window or from the **Icepak**>**Monitor** menu. The following table describes the behavior of monitor assignment for various geometry selections.

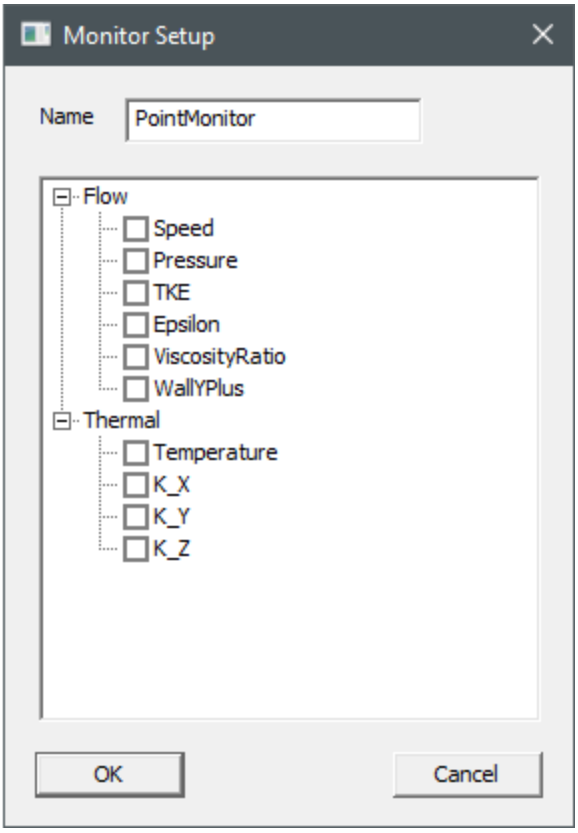
Geometry	Face Monitor Assignment	Point Monitor Assignment
Objects	Cannot be assigned to an object.	Assigned to the centroid of the selected object.
Faces	Assigned to the selected face.	Assigned to the center of the selected face. Cannot be assigned to non-planar faces.
Edges	Cannot be assigned to an edge.	Assigned to the center of the selected linear edge. Cannot be assigned to curved edges.
Vertices	Cannot be assigned to a vertex.	Assigned to the selected vertex.
Points	Cannot be assigned to a point.	Assigned to the selected point.

> **Note:**
>
> When using the **Selection Mode Multi**, multiple face monitors can be assigned simultaneously with a multi-selection that includes only faces. Multiple point monitors can be assigned simultaneously with a multi-selection that includes faces, points, edges, and vertices.

See the figures below for available data for face or point monitors. Solution monitors appear in the **Design List**>**Monitors** tab, which can be accessed from the **Icepak**>**Monitor** menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



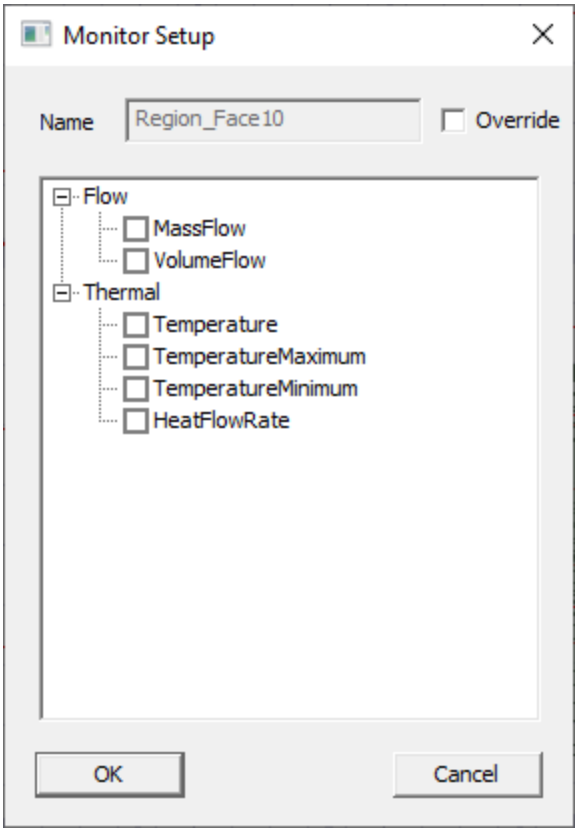


Assign a Face Monitor

Face monitors can be assigned to two-dimensional geometry or individual faces of three-dimensional geometry. To assign a face monitor and specify its data:

1. Press F to enable face-selection mode.
2. In the **3D Modeler** window, select and right-click a two-dimensional geometry component or face of a three-dimensional geometry component.
3. From the right-click menu, select **Assign Monitor** > **Face** to open the **Monitor Setup** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Alternatively, access the **Monitor Setup** dialog box from the **Icepak**>**Monitor** menu or by right-clicking **Monitor** in the **Project Manager**.

4. If needed, select **Override** and enter a **Name** for the face monitor.

> **Note:**
>
> By default, the monitor name is based on the selected object. To create a custom base name, see Setting Monitor Base Names for more information.

5. Select the data to display during the calculation:
   - Expand **Flow** and select the flow data. The following flow data types are available:
     - **Mass Flow**
     - **Volume Flow**
   - Expand **Thermal**, select the thermal data. The following thermal data types are available:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

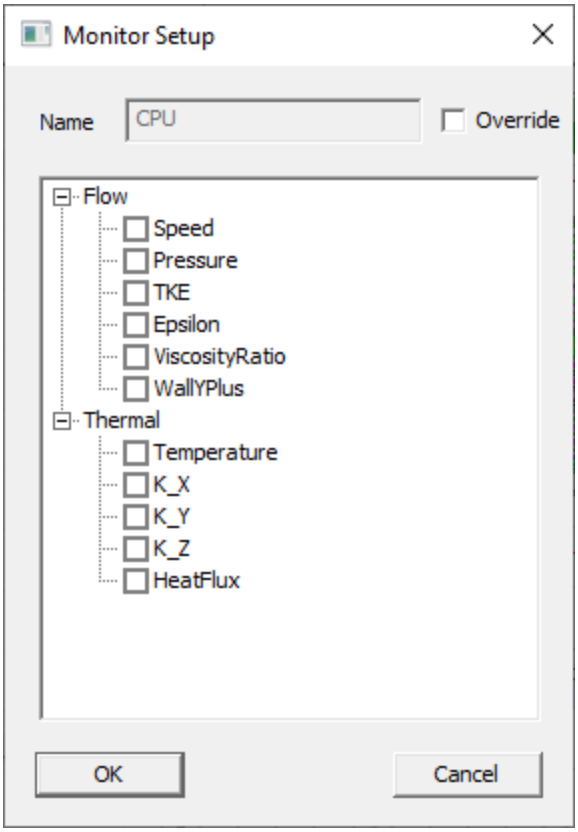
- **Temperature**
- **TemperatureMaximum**
- **TemperatureMinimum**
- **Heat Flux**

6. Click **OK** to save the face monitor. It is displayed under **Monitor** in the **Project Manager**.

Assign a Point Monitor

Point monitors can be assigned to objects, faces, edges, vertices, and points. To assign a point monitor and specify its data:

1. Select the geometry from the history tree or in the **3D Modeler**.
2. Right-click in the **3D Modeler**.
3. From the right-click menu, select **Assign Monitor**>**Point** to open the **Monitor Setup** dialog box.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Alternatively, access the **Monitor Setup** dialog box from the **Icepak**>**Monitor** menu or by right-clicking **Monitor** in the **Project Manager**.

4. If needed, select **Override** and enter a **Name** for the point monitor.

> **Note:**
>
> By default, the monitor name is based on the selected object. To create a custom base name, see Setting Monitor Base Names for more information.

5. Select the data to display during the calculation:
   - Expand **Flow** and select the flow data. The following flow data types are available:
     - **Speed**
     - **Pressure**
     - **TKE**
     - **Epsilon**
     - **ViscosityRatio**
     - **WallYPlus**
   - Expand **Thermal**, select the thermal data. The following thermal data types are available:
     - **Temperature**
     - **K_X**
     - **K_Y**
     - **K_Z**
     - **HeatFlux**
     - **TemperatureMaximum**
     - **TemperatureMinimum**
6. Click **OK** to save the point monitor. It is displayed under **Monitor** in the **Project Manager**

Edit Monitor Definitions

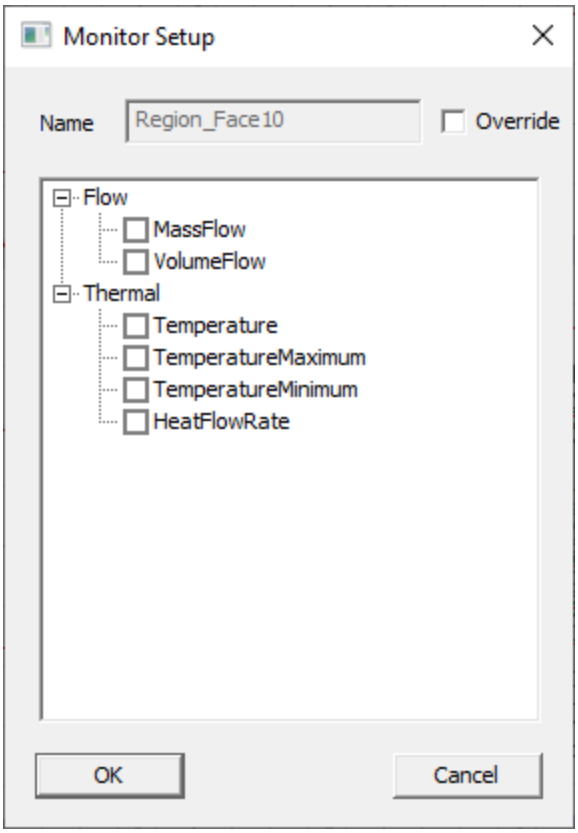
You can change the data quantity that monitors display individually or with multiple monitors selected.

To edit a monitor individually:

1. In the **Project Manager** under **Monitor**, double-click the monitor to open the **Monitor Setup** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the desired **Flow** and/or **Thermal** data quantities.



3. Click **OK**.

> **Note:**
>
> Alternatively, you can select a monitor in the **Project Manager** and edit it directly in the **Properties** window.

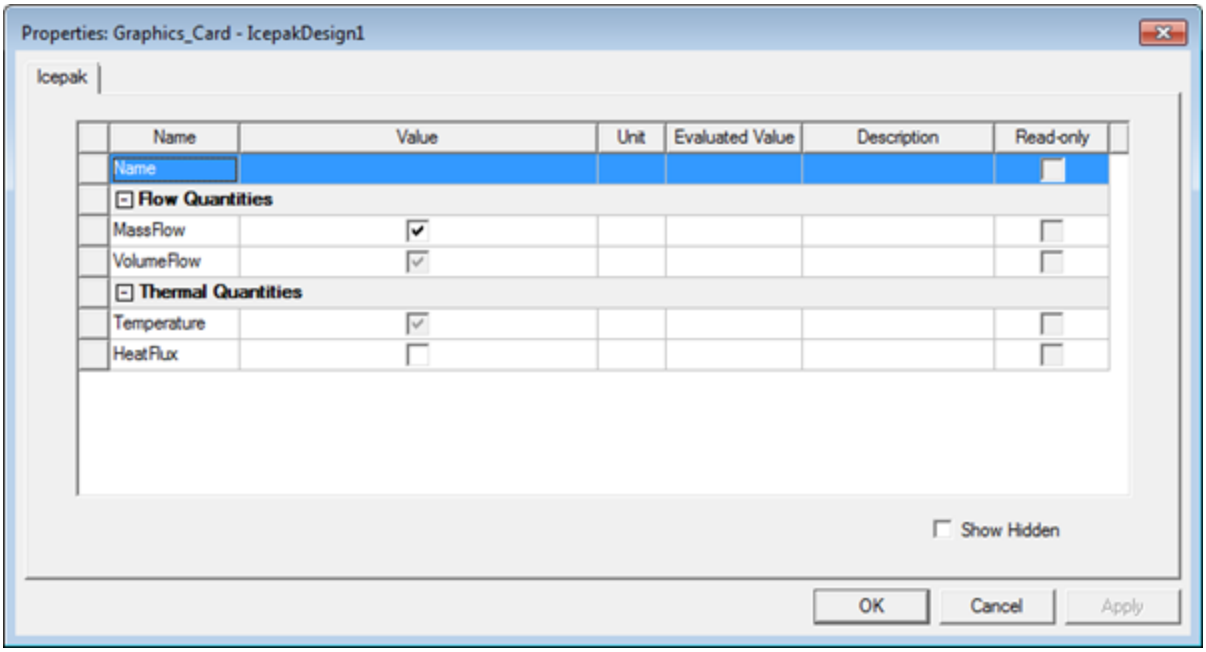
To edit multiple monitors simultaneously:

1. In the **Project Manager**, double-click **Monitor** to open the **Design List** dialog box.
2. Press **Ctrl** and select multiple monitors you want to edit.
3. Click **Properties** to open the **Properties** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Data quantities whose selection are the same between all monitor points included in the selection are active and editable. Data quantities whose selection are different among the monitor points included in the selection are inactive.



4. Select the desired **Flow** and/or **Thermal** data quantities.
5. Click **OK**.

Reassign a Face Monitor

Face monitors can be reassigned to different geometry after their initial creation. To reassign a face monitor and specify its data:

1. Press F to enable face-selection mode.
2. In the **3D Modeler** window, select a different two-dimensional geometry component or face of a three-dimensional geometry component.
3. Under **Monitor** in the **Project Manager**, right-click the face monitor you want to reassign and select **Reassign**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The face monitor is reassigned to new geometry and retains the selected data to display during the calculation. To edit the data selection, double-click the reassigned face monitor in the **Project Manager**.

Reassign a Point Monitor

Point monitors can be reassigned to different geometry after their initial creation. To reassign a point monitor and specify its data:

1. Select a different piece of geometry from the history tree or in the **3D Modeler**.
2. Under **Monitor** in the **Project Manager**, right-click the point monitor you want to reassign and select **Reassign**.

> **Note:**
>
> The point monitor is reassigned to new geometry and retains the selected data to display during the calculation. To edit the data selection, double-click the reassigned point monitor in the **Project Manager**.

Deleting Monitors

Monitors can be deleted individually or all simultaneously. To delete a monitor:

Do one of the following:

- To delete an individual monitor, right-click the monitor in the **Project Manager** and select **Delete**.
- To delete an individual monitor, select the monitor in the **Project Manager** and press **Delete**.
- To delete all monitors, right-click **Monitor** in the **Project Manager** and select **Delete All**.

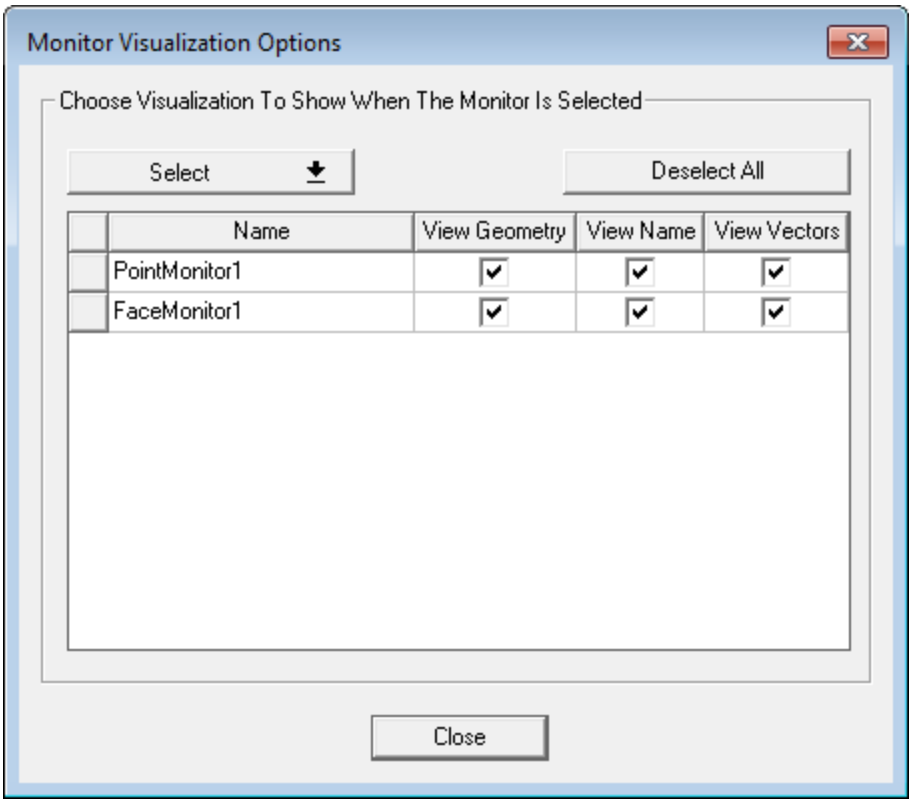
> **Note:**
>
> Alternatively, access the **Delete All** function from the **Icepak**>**Monitor** menu.

Monitor Visualization Options

Monitor information displayed in the **3D Modeler** window can be enabled or disabled using the **Monitor Visualization Options**, which can be accessed from the **Icepak**>**Monitor** menu or by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

right-clicking **Monitor** in the **Project Manager**.



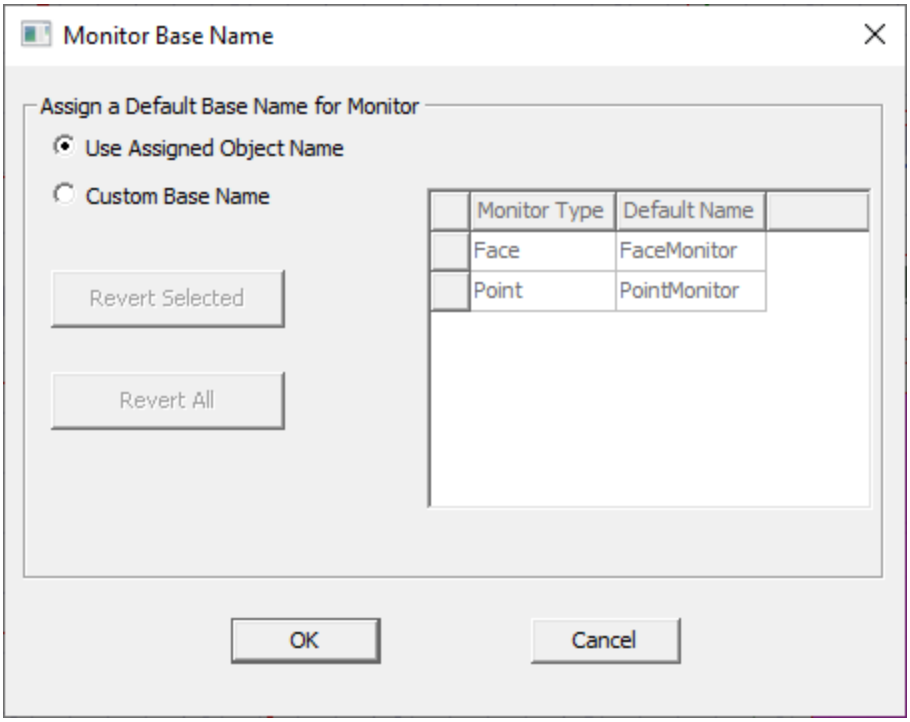
Select	The **Select** drop-down contains options to select **All** monitors, **All Face Monitors**, or **All Point Monitors**. Select **By Name** to open the **Select By Name** dialog box in which you can enter a monitor name and click **OK** to select it.
Deselect All	Click **Deselect All** to deselect all face and point monitors.
Name	The **Name** of the monitor appears in the **3D Modeler** window.
View Geometry	Enable **View Geometry** for the monitors for which you want to display geometry in the **3D Modeler**.
View Name	Enable **View Name** for the monitors for which you want to display names in the **3D Modeler**.
View Vectors	Enable **View Vectors** for the monitors for which you want to display vectors in the **3D Modeler**.
Close	Click **Close** to close the **Monitor Visualization Options** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Monitor Base Names

When creating a monitor, the name is based on the selected object by default. The naming convention can be customized using the **Monitor Base Name** dialog box, which can be accessed from the **Icepak**>**Monitor** menu or by right-clicking **Monitor** in the **Project Manager**.

> **Note:**
>
> Monitor names can be changed by selecting them in the **Project Manager** and pressing F2 or right-clicking them in the **Project Manager** and selecting **Rename**.



Use Assigned Object Base Name	Select **Use Assigned Object Name** for new monitor points to be named based on the selected object.
Custom Base Name	Select **Custom Base Name** to enable the base name options below.
Monitor Type	**Monitor Type** specifies the type of monitor.
Default Name	Enter a **Default Name** to use as the base name for new monitors.
Revert Selected	Click **Revert Selected** to restore the default value for the **Default Name** of the selected monitor type.
Revert All	Click **Revert All** to restore the default value for the **Default Name** of all

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	monitor types.
OK	Enable **View Name** for the monitors for which you want to display names in the **3D Modeler**.
Cancel	Click **Cancel** to close the **Monitor Base Name** dialog box without saving any changes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14 - Assigning Solar Loading

Icepak includes the ability to model solar loads, which includes the effects of direct solar illumination as well as diffuse solar radiation. Given the model geometry and pertinent solar information such as terrestrial location, date, and time, the model performs a ray tracing shading test for all boundary surfaces. This approach includes a two-band (visible and infrared) spectral model for solar illumination. The transparent material model computes the absorptivity and transmissivity as functions of incident angle.

Ansys Icepak provides a solar calculator that can be used to compute solar beam direction and irradiation. Alternatively you can specify a value for the Direct solar irradiation, Diffuse solar irradiation flux and the sun direction vector. To use the solar load model, follow the procedure outlined below. On the **Solar Radiation** tab of the Icepak Solve Setup dialox box, define the solar radiation parameters.

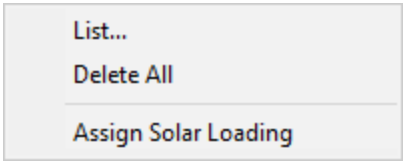
Assign Solar Loading

To assign solar loading for a piece of geometry:

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Solar Loading**.
4. In the **Solar Loading** dialog box, enter a **Name** for the solar loading assignment.
5. Click **OK**.

Solar Loading Context Menus

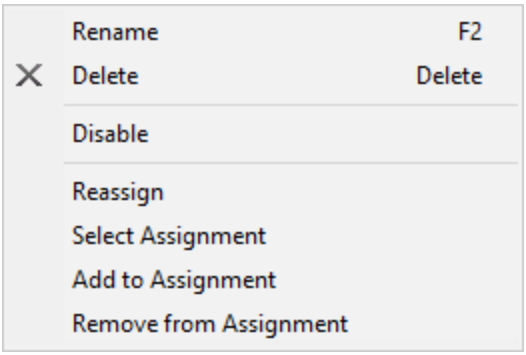
In the **Project Manager**, right-click the **Solar Loading** module to display the context menu.



List	**List** opens the **Design List** dialog box on the **Solar Loading** tab. See Viewing the Design List for Icepak for more information.
Delete All	**Delete All** removes all solar loading assignments from the design.
Assign Solar Loading	**Assign Solar Loading** opens the **Solar Loading** dialog box to assign solar loading to the currently selected geometry.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project Manager**, expand **Solar Loading** and right-click a solar loading assignment to display the context menu.

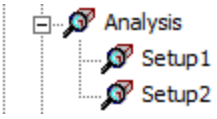


Rename	**Rename** allows you to edit the name of the solar loading assignment.
Delete	**Delete** removes the solar loading assignment from the design.
Disable	**Disable** excludes the solar loading assignment from the solver's calculations when running the analysis.
Reassign	**Reassign** assigns the solar loading to the currently selected geometry.
Select Assignment	**Select Assignment** selects the geometry assigned to the solar loading in the **3D Modeler** window and the history tree.
Add to Assignment	**Add to Assignment** adds the currently selected geometry to the solar loading assignment.
Remove from Assignment	**Remove from Assignment** removes the selected geometry from the solar loading assignment.

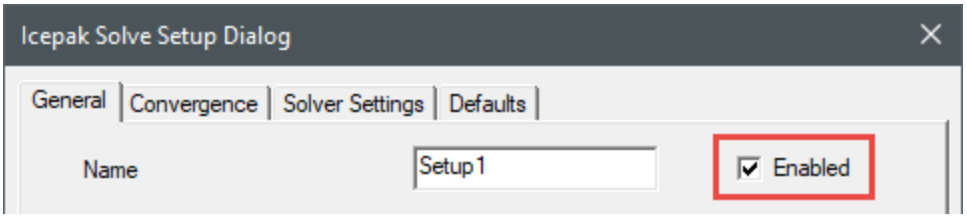
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - Specifying Solution Settings

Specify how Ansys Electronics Desktop will compute a solution by adding a solution setup to the design. In the **Project Manager**, right-click **Analysis** and click **Add Solution Setup**.



You can define more than one solution setup per design. To apply a solution setup's settings, ensure that the **Enabled** check box is selected on the **General** tab of the **Icepak Solve Setup Dialog**.



The Icepak Solve Setup Dialog

> **Note:** For information on transient simulations, see The Icepak Solve Setup Dialog (Transient).

A solution setup includes the settings you need to define for the Icepak simulation. The **Icepak Solve Setup** dialog box contains the following tabs:

- **General**
- **Convergence**
- **Solver Settings**
- **Radiation**
- **Solar Radiation**
- **Defaults**

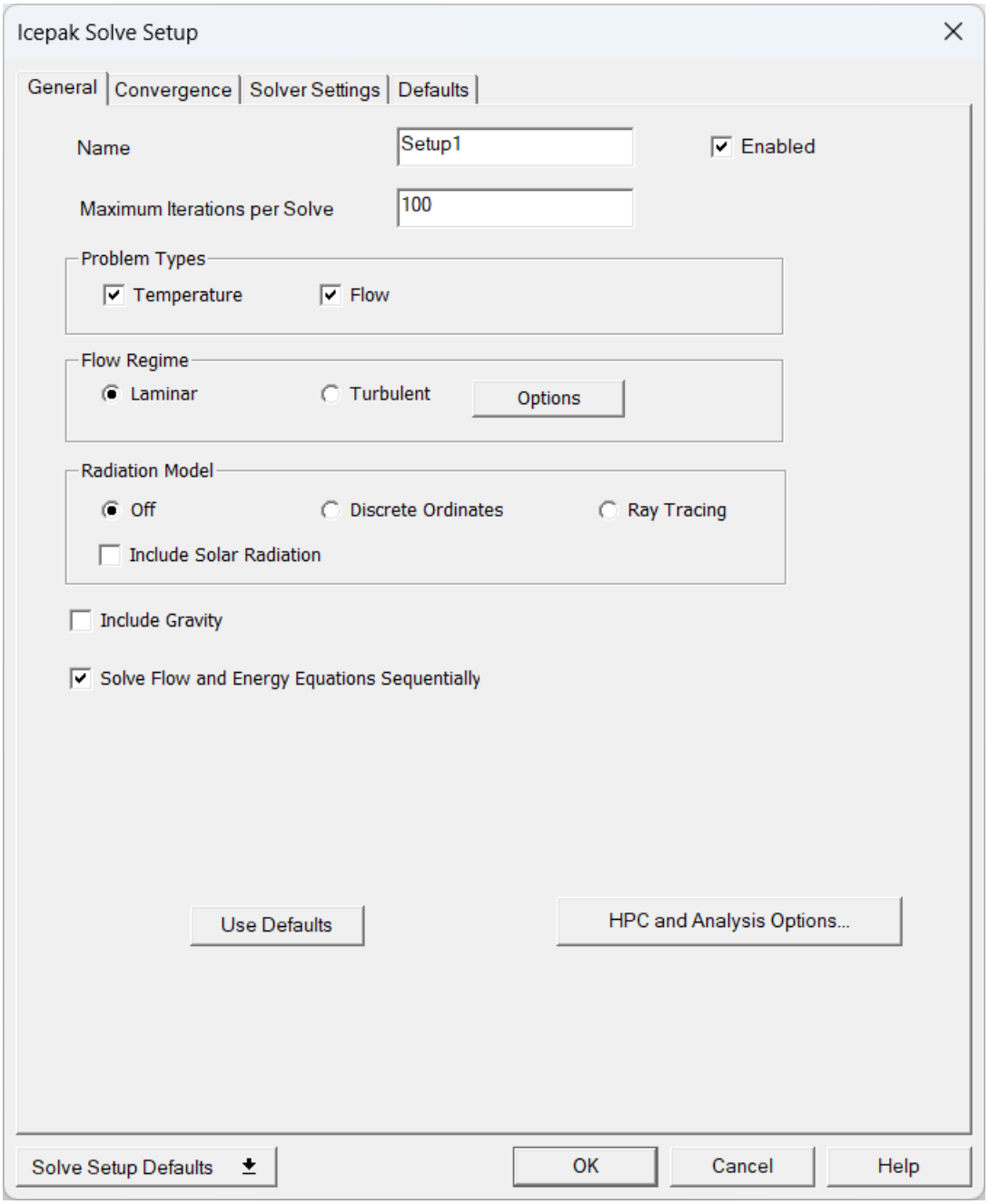
The **Solve Setup Defaults** drop-down list contains the following options that automatically configure settings based on your selection. See Icepak Solve Setup Defaults for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Solve Setup Defaults**
- **Forced Convection Defaults**
- **Natural Convection Defaults**
- **Mixed Convection Defaults**
- **Conduction Only Defaults**

> **Note:**
>
> Some settings are enabled or disabled based on other setting selections. For example, the **Radiation** tab is only displayed if radiation is enabled by setting the **Radiation Model** to **Discrete Ordinates** or **Ray Tracing**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** of the setup appears in the **Project Manager** window.
Enabled	The **Enabled** check box disables/enables a setup. Only enabled setups are run when you select **Analyze All**.
Maximum Iterations per Solve	**Maximum Iterations per Solve** is the number of solution iterations to be performed in a simulation. The simulation will

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	stop when these iterations have been performed or the **Convergence Criteria** are satisfied, whichever happens first. For relatively simple models, the default of 100 should be sufficient for the solution to converge, but for more complex models you may need to increase this value.
Note: If **Start/Continue from a previously solved setup** is enabled, the simulation runs the specified number of iterations for the new simulation.	
Problem Types	
Temperature	When enabled, the **Temperature** check box instructs the solver to include thermal calculations in the analysis.
Flow	When enabled, the **Flow** check box instructs the solver to include fluid calculations in the analysis.
Flow Regime	
Note: For detailed information regarding turbulence theory, see the Icepak Technical Notes Turbulence section.	
Laminar	**Laminar** flow is smooth, regular, deterministic, and steady. In laminar flow, fluid mixing and heat transfer take place on a molecular level. The molecular (or dynamic) viscosity and the thermal conductivity are the quantities that measure the amount of mixing and heat transfer. **Laminar** is the default flow regime.
Turbulent	**Turbulent** flows are characterized by fluctuating velocity fields. In turbulent flow, the degree of fluid mixing and heat transfer is much greater than in laminar flow, and takes place on a global, or macroscopic, level rather than on a molecular level. The amount of fluid mixing is measured by an effective viscosity, which is the sum of the dynamic viscosity and the turbulent eddy viscosity. If you select **Turbulent**, click **Options** to open the **Turbulent Flow Model** dialog and select a model. The default turbulent flow model is **Zero Equation**.
Radiation Model	
Note: In addition to enabling radiation in a solution setup, it might be necessary to define radiation settings for boundary conditions that require it (for example, network face nodes). See Assigning Thermal Boundary Conditions for more information.	
Off	**Off** disables radiation for the solution. Off is the default setting.
Discrete Ordinates	The **Discrete Ordinates** (DO) radiation model solves the radiative transfer equation (RTE) for a finite number of discrete solid angles, each associated with a vector direction $\vec{s}$ fixed in the global Cartesian system (x, y, z). The DO model

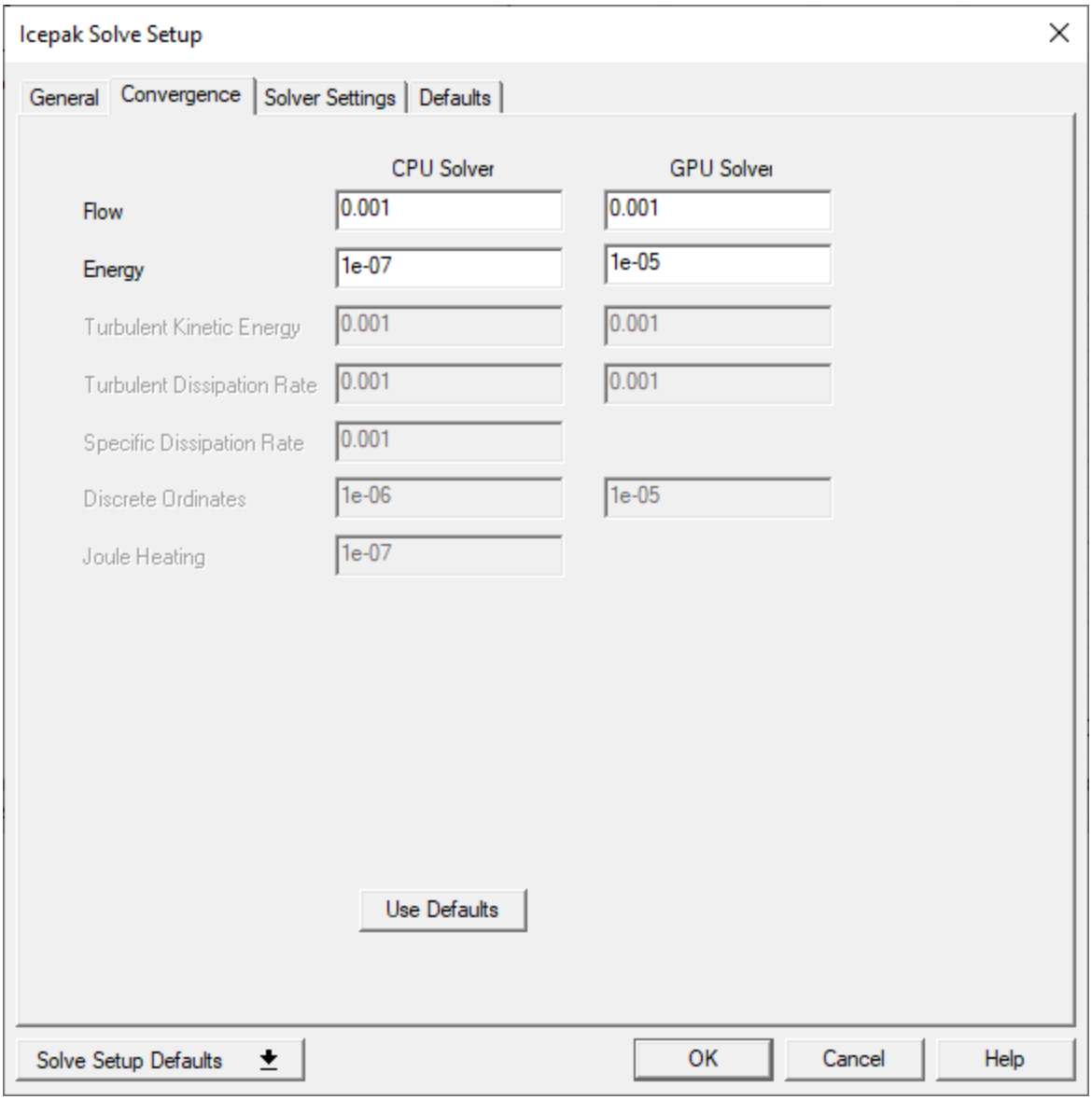
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	solves for as many transport equations as there are directions $\vec{s}$. The solution method is identical to that used for the fluid flow and energy equations.
Ray Tracing	With the **Ray Tracing** radiation model, the ray paths are calculated and stored prior to the fluid flow calculation. At each radiating face, rays are fired at discrete values of the polar and azimuthal angles. To cover the radiating hemisphere, θ is varied from 0 to $\frac{\pi}{2}$ and ϕ from 0 to 2π. Each ray is then traced to determine the control volumes it intercepts as well as its length within each control volume. This information is then stored in the radiation file (.s2s.gz), which is then read in before the fluid flow calculations begin.
Include Solar Radiation	To include the effect of solar radiation in the simulation, select **Include Solar Radiation**. Define the solar radiation settings on the **Solar Radiation** tab.
Include Gravity	To include the effect of gravity in the simulation for natural convection models, select **Include Gravity**. Define the gravity vector settings in the Icepak Design Settings. Natural convection models require **Include Gravity** to be enabled.
Solve Energy and Flow Equations Sequentially	**Solve Energy and Flow Equations Sequentially** instructs the solver to solve the flow equations first and then solve the energy equation. This is suitable for cases where there is no coupling between the flow and energy equations (for example, forced convection problems with no gravity). This option may reduce the time required to obtain a solution.
HPC and Analysis Options	High performance computing (HPC) enables a range of different technologies in Icepak that allows efficient simulation of extremely large and complex problems. See High Performance Computing for more information.

Note:

Ray tracing radiation is not calculated on the faces of mesh regions that touch other objects.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

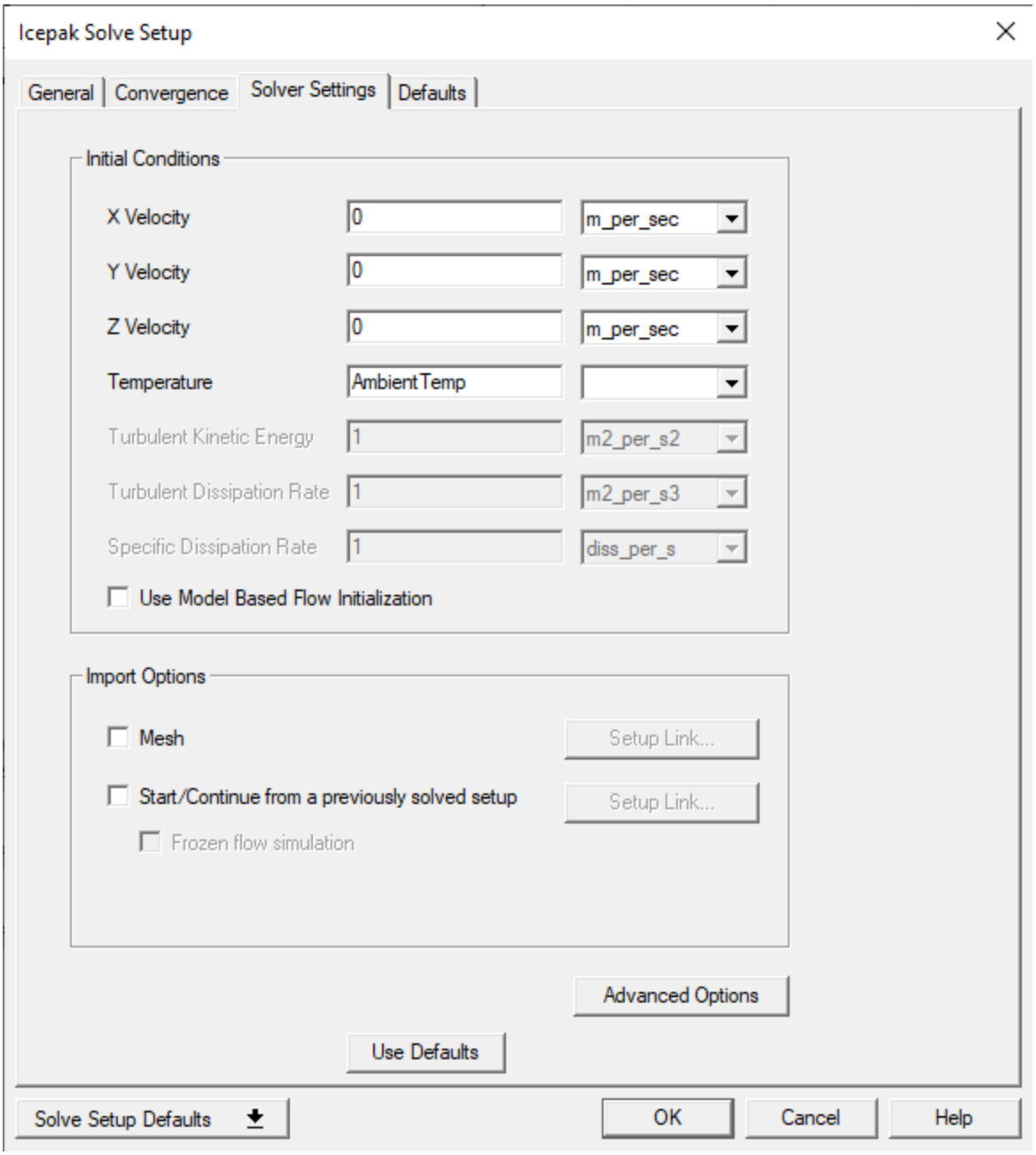


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow	These settings are the solution-residual values used to determine convergence for simulations run on the CPU or GPU (beta). Solution residuals measure the error or imbalance in the conservation equations that Icepak solves. When all solution residuals are less than or equal to their specified convergence criteria, the solution will be considered converged. **Turbulent Kinetic Energy**, **Turbulent Dissipation**, and **Specific Dissipation Rate** are enabled by the selection of certain turbulent flow models. **Discrete Ordinates** is enabled if the **Radiation Model** is set to **Discrete Ordinates**. **Joule Heating** is enabled if the power for a block boundary condition is defined as joule heating.
Energy	
Turbulent Kinetic Energy	
Turbulent Dissipation	
Specific Dissipation Rate	
Discrete Ordinates	
Joule Heating (CPU Only)	

> **Note:** GPU convergence settings is a beta feature. See Enabling GPU for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

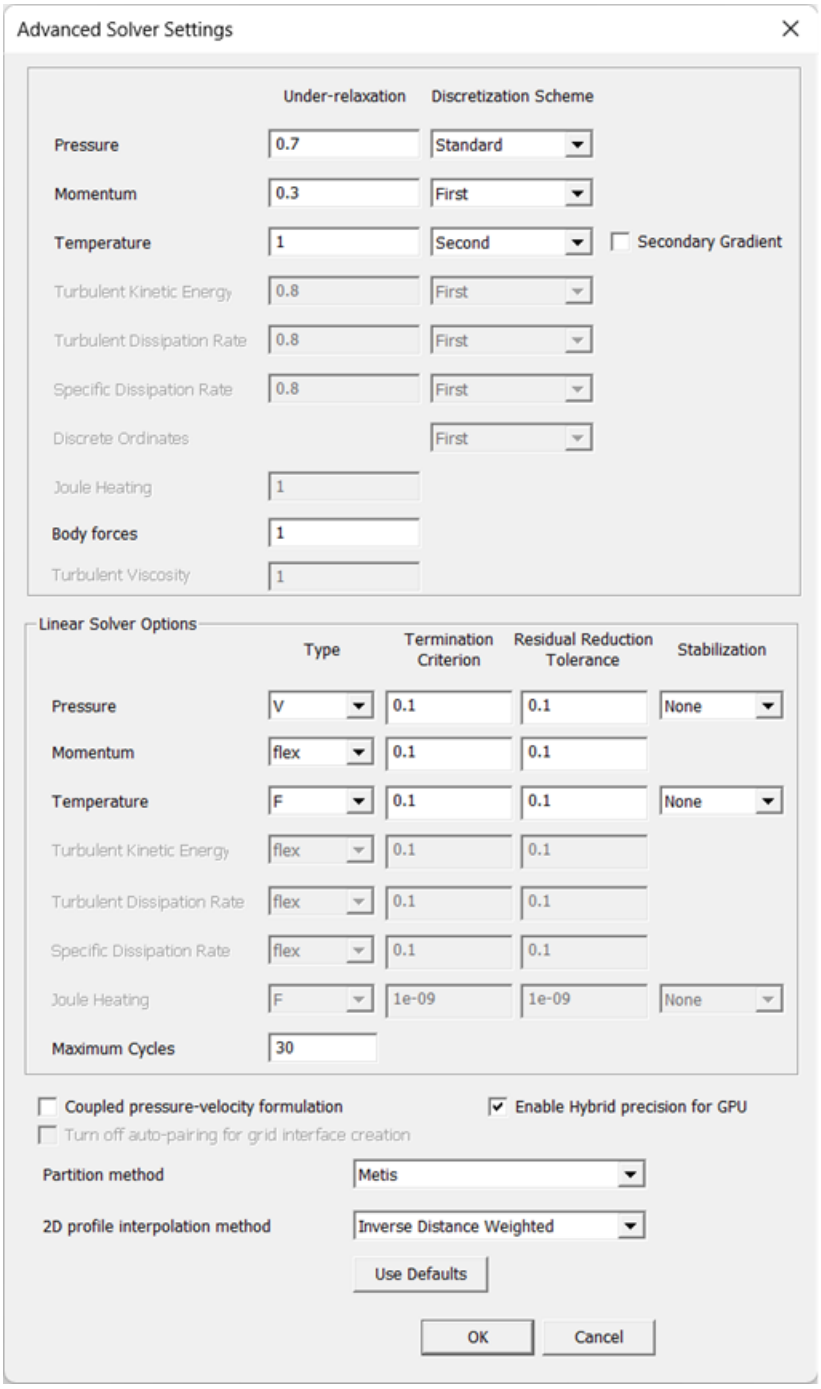


Initial Conditions

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z Velocity	These settings specify the initial conditions for the fluid in the flow region. The initial conditions are the initial guess for the various solution fields used by the solution procedure. **Turbulent Kinetic Energy**, **Turbulent Dissipation**, and **Specific Dissipation Rate** are enabled by the selection of certain turbulent flow models.
Temperature	
Turbulent Kinetic Energy	
Turbulent Dissipation	
Specific Dissipation Rate	
Use Model Based Flow Initialization	**Use Model Based Flow Initialization** instructs the Electronics Desktop to automatically evaluate inlet and outlet conditions to use optimal values for solution variables.
Import Options	
Mesh	**Mesh** allows to import mesh from an existing Icepak design and is automatically enabled after using the Add a Mesh Linked Solution Setup option. Mesh from the selected design is imported and used in the simulation. The mesh from the source design must conform to the target design's model geometry. Seeing Importing Mesh for more information.
Start/Continue...	**Start/Continue from a previously solved setup** allows you to begin or resume a simulation from a design you have previously solved.
Note: If the design has a mesh link and solve setup link, if you remove the solve setup link, the mesh link is not automatically removed.	
Frozen flow simulation	**Frozen flow simulation** allows you to disable the flow of the linked steady-state or transient design. If enabled, only the energy equation is solved during the analysis. If the linked solution did not solve for flow, this option is not active.
Advanced Options	Click **Advanced Options** to display the **Advanced Solver Settings** dialog.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under-relaxation	**Under-relaxation** settings control the update of computed variables at each iteration. This means that all equations solved using Icepak will have under-relaxation factors associated with them.
Discretization Scheme	**Discretization Scheme** specifies the discretization scheme for

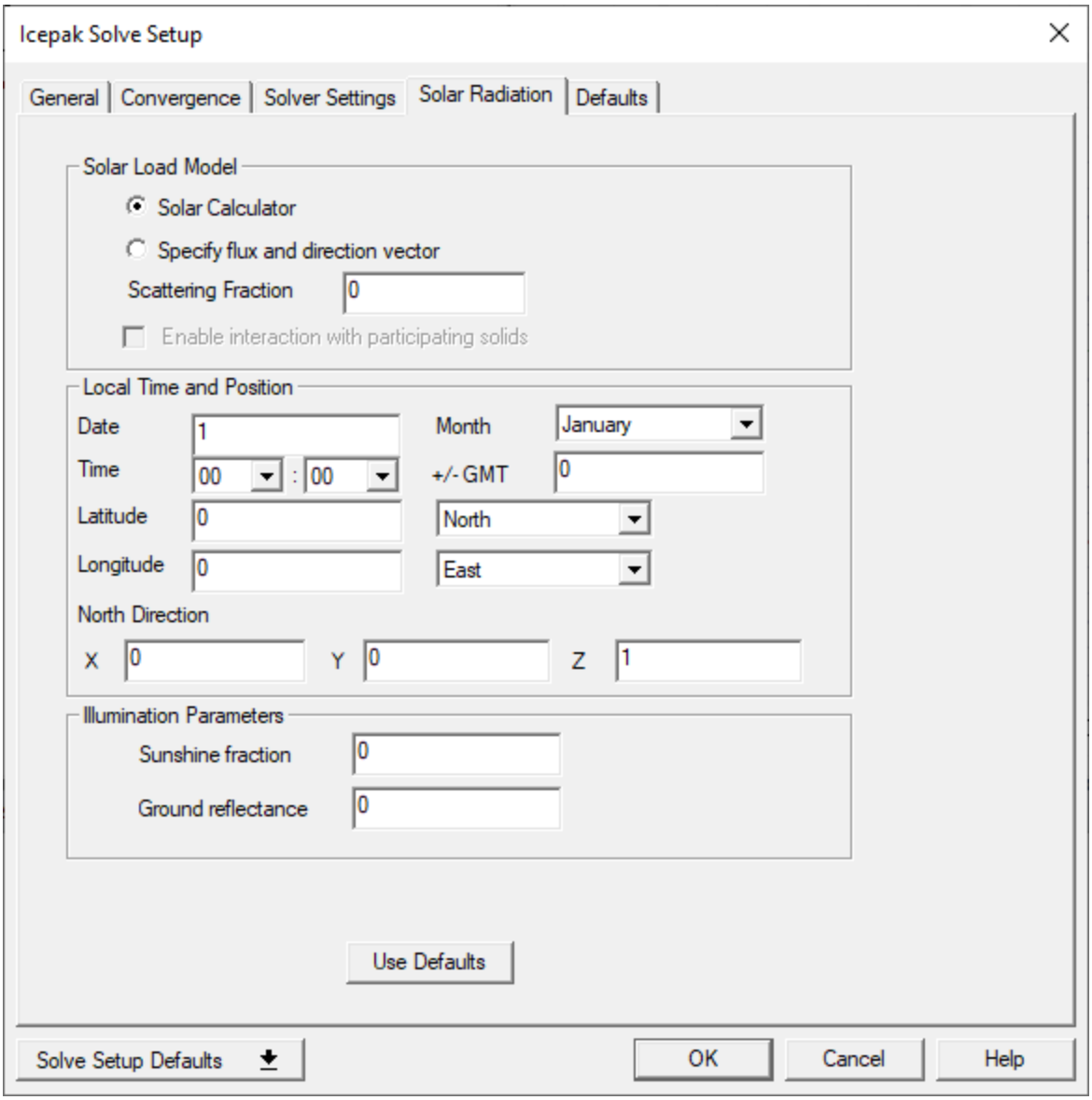
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the convection terms of each governing equation. For the **Temperature** equation, the **Secondary gradient** option is used to include or exclude temperature gradients in the discretization scheme.
Linear Solver Options	
Icepak uses a multigrid scheme to accelerate solution convergence. You can set the parameters related to the multigrid solver under **Linear Solver Options**.	
Type	**Type** specifies the multigrid cycle type for each equation that Ansys Icepak solves. By default, the V cycle is used for the pressure equation, the F cycle for temperature and joule heating potential, and the flex cycle is used for all other equations. You should generally not need to modify these settings.
Termination Criterion	**Termination Criterion** controls the multigrid solver in different ways for different cycles. For the flex cycle, the Termination criterion governs when the solver should return to a finer grid level (i.e., when the residuals have improved sufficiently on the current level). For the V, W and F cycles, the Termination criterion determines whether or not another cycle should be performed on the finest (original) grid level. If the current residual on the finest level does not satisfy the Termination criterion, Icepak will perform another multigrid cycle. For most cases, you should not need to modify the settings for the **Termination Criterion**.
Residual Reduction Tolerance	**Residual Reduction Tolerance** dictates when a coarser grid level must be visited (due to insufficient improvement in the solution on the current level). This parameter is used only by the flex cycle. With a larger value of the Residual reduction tolerance, coarse levels will be visited less often (and vice versa). For most cases, you should not need to modify the settings for the **Residual Reduction Tolerance**.
Stablilization	**Stabilization** If desired, you can choose the bi-conjugate gradient stabilized method (BCGSTAB) for the pressure and temperature equations. This option can be used in situations involving high temperature and/or pressure gradients to improve the convergence of the linear solver.
Maximum cycles	**Maximum cycles** sets the maximum number of levels cells are recursively grouped to coarsen the grid. When there are significant variations in the solution variables (such are temperature, velocity, etc.) across the entire domain, a larger maximum cycles value will help speed up the iterative linear solver. For models where solution variations are localized, a smaller maximum cycles value can speed up the solve time by reducing unnecessary coarsening. The default value is 30. For the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	latter scenario, a value of 10 is recommended.
Coupled velocity-pressure formulation	**Coupled velocity-pressure formulation** enables the pressure-based coupled algorithm which offers some advantages over the default segregated pressure-velocity formulation using the SIMPLE algorithm. The pressure-based coupled algorithm obtains a more robust and efficient single phase implementation for steady-state flows.
Turn off auto-pairing for grid interface creation	By default, the automatic pairing of mesh region boundaries (or interfaces) is done based on proximity of interfaces with each other. **Turn off auto-pairing for grid interface creation** disables auto-pairing (i.e., the mesh region interfaces are paired explicitly per region). Note that auto-pairing requires additional setup time during solve. In a few rare cases, the auto-pairing method might result in the divergence of the solver. In such instances, auto-pairing can be disabled to aid in the convergence of the solver.
Partition method	**Partition method** enables you to specify the algorithm for partitioning the global mesh region and subregions. See Partition Methods for more information.
2D profile interpolation method	**2D profile interpolation method** enables you to specify the interpolation method for point profiles. Select one of the three choices from the drop-down list. **Constant** is a zeroth-order interpolation. For each cell face at the boundary, the solver uses the value from the profile file closest to the cell. Therefore, the accuracy of the interpolated profile will be affected by the density of the data points in your profile file. **Inverse distance weighted** assigns a value to each cell face at the boundary, based on weighted contributions from the values in the profile file. The weighted factor is inversely proportional to the distance between the profile point and the cell face center. This is the default interpolation method for point profiles. **Least Squares** assigns values to the cell faces at the boundary through a first-order interpolation method that tries to minimize the sum of the squares of the offsets.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



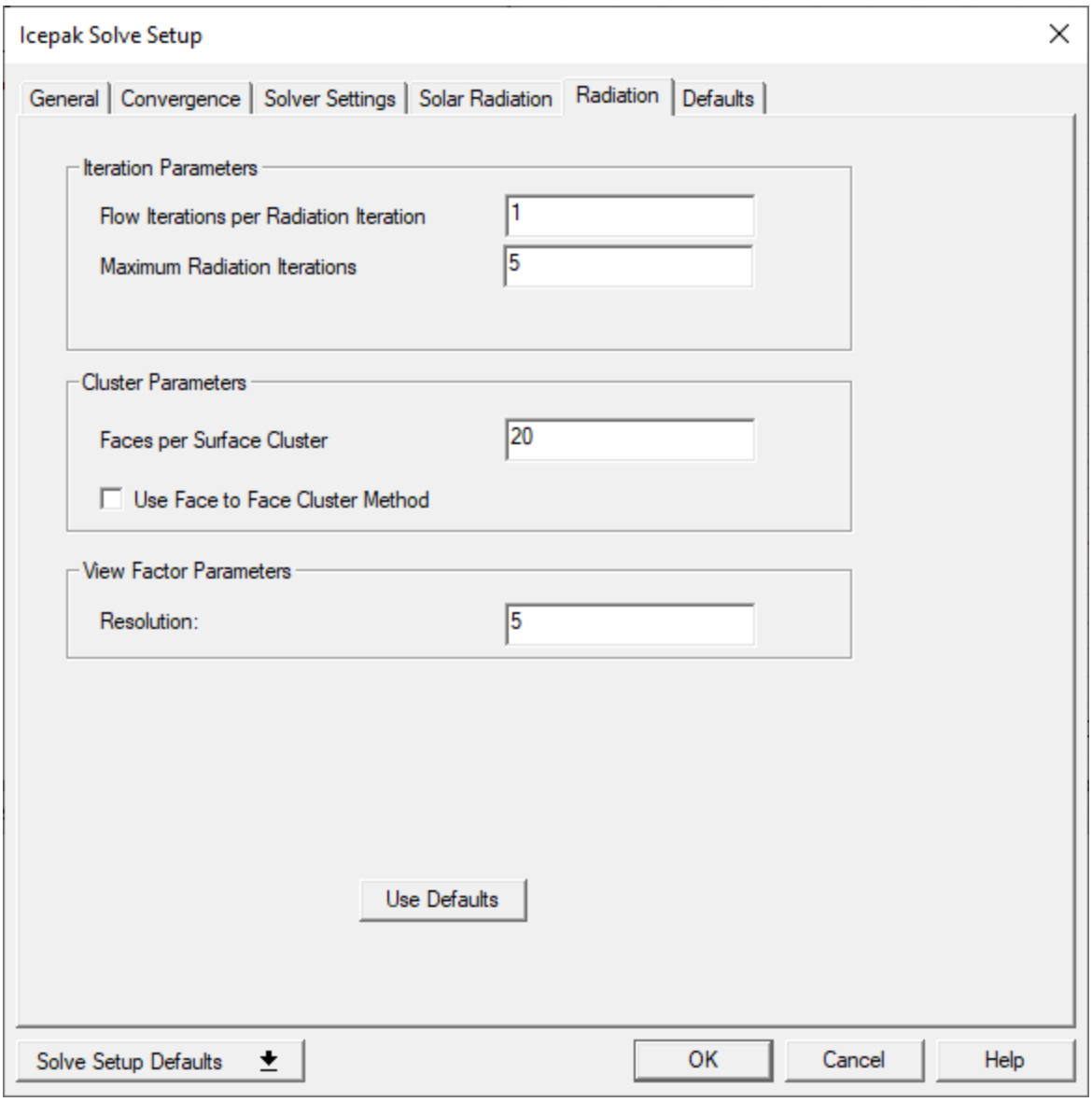
Solar Load Model	
Solar Load Model	Select **Solar Calculator** to compute the solar radiation based on time and geographic parameters or **Specify flux and direction vector** to specify solar radiation values and a solar radiation direction vector.
Scattering Fraction	**Scattering Fraction** specifies the amount of direct solar radiation that is reflected from opaque objects in your model. The reflected radiation is evenly distributed among all objects that participate in solar loading.
Enable interaction with	When **Discrete Ordinates** is defined as the radiation model,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

participating solids	select **Enable interaction with participating solids** to model the solar irradiation as discrete ordinate fluxes at the global region boundaries. This allows semi-transparent walls and participating solids to absorb, refract and scatter the incident solar irradiation.
Local Time and Position (Solar Calculator)	
Date and **Month**	Specify a **Date** and **Month**.
Time	**Time** specifies the local time at the desired location. Select the hour and minute from the drop-down lists. The time is based on a 24-hour clock, therefore acceptable values range from 0 h 0 min (12:00 a.m.) to 23 h 59 min (11:59 p.m.).
+/- GMT	**+/- GMT** specifies the local time zone of the desired location. If the time you enter is a local time, specify the current time zone by providing the offset in the +/- GMT entry. If the time you enter is already in GMT, then +/- GMT should be set to 0 (zero). For example, enter -5 to specify Eastern Standard Time (EST).
Latitude	Specify the **Latitude** of the desired location. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator. Select the hemisphere (N or S) from the menu to the right of the Local Longitude entry field.
Longitude	Specify the **Longitude** of the desired location. The longitude is approximated if you specify the local time zone, but you can enter a more precise value if you know it. Any value you enter here will take precedence over the time zone. Values may range from 0° to 180°. Select the hemisphere (W or E) from the menu to the right of the Local Longitude entry field.
North Direction Vector	The **North Direction Vector** specifies the northward direction relative to the model. Enter the appropriate values in the **X**, **Y**, and **Z** fields. The default northward direction is in the Z direction.
Illumination Parameters (Solar Calculator)	
Sunshine fraction	**Sunshine fraction** is a factor between 0 and 1 used to account for the effects of clouds that may reduce the direct solar irradiation. Clear sky is modeled by setting the value equal to 1 and complete cloud cover is modeled by setting the value equal to 0. Partial cloud cover is modeled by setting the value to be between 0 and 1. The default value is 1.0.
Ground reflectance	**Ground reflectance** is a parameter that is used in determining the contributions of reflected solar radiation from ground surfaces. Reflected solar radiation from ground surfaces is a function of the direct normal irradiation, the time of the year, the tilt angle of the surface, and the ground reflectance. If is treated as part of the total diffuse solar irradiation. Ground reflectance values can vary

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	depending on the ground surface (that is, concrete, grass, rock, gravel, asphalt). The default value is 0.2.
Solar Flux and Direction Vector	
Direct solar irradiation	**Direct solar irradiation** specifies the amount of energy per unit area due to direct solar irradiation. This value may depend on the time of the year and the clearness of the sky.
Diffuse solar irradiation	**Diffuse solar irradiation** specifies the amount of energy per unit area due to diffuse solar irradiation. This value may depend on the time of year, the clearness of the sky, and also on ground reflectivity.
Solar Direction	**Solar Direction** specifies the direction of solar irradiation relative to the model. Enter the appropriate values in the **X**, **Y**, and **Z** fields.



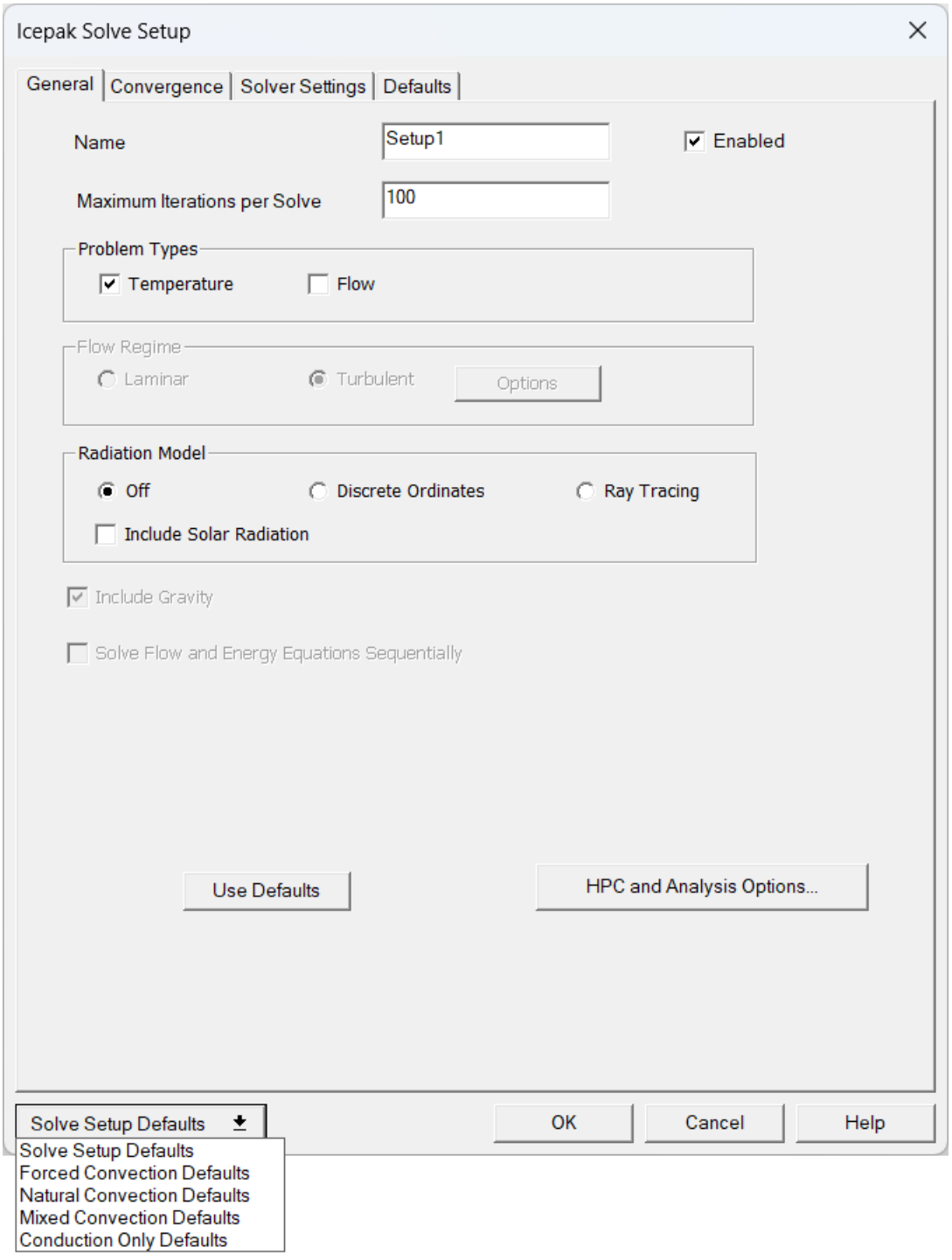
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iteration Parameters	
Flow Iterations per Radiation Iteration	The **Flow Iterations per Radiation Iteration** parameter is set to 1 by default. This implies that the radiation calculation is performed once every iteration of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.
Maximum Radiation Iterations (Ray Tracing only)	The **Maximum Radiation Iterations** controls the maximum number of iterations of the radiation calculation during each global iteration.
Angular Discretization (Discrete Ordinates only)	
Theta and **Phi Divisions**	**Theta Divisions** ($N\theta$) and **Phi Divisions** ($N\phi$) will define the number of control angles used to discretize each octant of the angular space. A finer angular discretization can be specified to better resolve the influence of small geometric features or strong spatial variations in temperature, but larger numbers of **Theta Divisions** and **Phi Divisions** will add to the cost of the computation.
Theta and **Phi Pixels**	**Theta Pixels** and **Phi Pixels** are used to control the pixelation that accounts for any control volume overhang. For problems involving gray-diffuse radiation, the default pixelation of 1 × 1 is usually sufficient. The computational effort, as a result of increasing the pixelation, is less than the computational effort caused by increasing the divisions. However, increasing the pixelation does add to the cost of computation.
Cluster Parameters (Ray Tracing only)	
Faces per Surface Cluster	**Faces per Surface Cluster** controls the number of radiating surfaces. By default, each is set to 20, so the number of surface clusters (radiating surfaces) will be equal to the total number of surface mesh elements divided by 20. For larger problems, you may want to reduce the number of surface clusters by increasing the faces per surface cluster to reduce both the size of the view factor file and the memory requirement. Such a reduction in the number of clusters, however, comes at the cost of some accuracy. The surfaces that are not adjacent to fluid regions will not participate in radiation.
View Factor Parameters (Ray Tracing only)	
Resolution	Increasing **Resolution** helps reduce the numerical errors caused by the finite-resolution approach used to compute the projected areas of the surface clusters and the resulting view factors. By default, it is set to 5. In most cases, however, the default settings will be sufficient.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Solve Setup Defaults

The **Icepak Solve Setup Defaults** drop-down list contains options that automatically define solution settings based on the selected simulation type.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following sections detail which settings are defined for each simulation type.

> **Note:**
>
> The **Solve Setup Defaults** options restores all settings to the application defaults.

Forced Convection

Select **Forced Convection Defaults** if your simulation includes forced convection, in which fluid moves within the computational domain by a source (for example, a fan). The following solution settings are defined as forced convection defaults.

> **Note:**
>
> **Forced Convection Defaults** does not appear as an option if the **Solution Type** is set to **Temperature Only**. The settings automatically defined depend on the **Solution Type**.

Solution Type = Flow Only	
Maximum Iterations per Solve	100
Problem Types	**Temperature** and **Flow** enabled
Radiation Model	**Off**
Under-relaxation	**Pressure** = 0.3 and **Momentum** = 0.7
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Solution Type = Temperature and Flow	
Maximum Number of Iterations	100
Problem Types	**Temperature** and **Flow** enabled
Radiation Model	**Off**
Gravity	Disabled
Solve Flow and Energy Equations Separately	Enabled
Under-relaxation	**Pressure** = 0.3 and **Momentum** = 0.7
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Convergence	**Energy** = 1e-12
Linear Solver Options: Temperature	**Termination Criterion** = 1e-06 and **Residual Reduction Tolerance** = 1e-06

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Natural Convection

Select **Natural Convection Defaults** if your simulation includes natural convection, in which fluid moves within the computational domain by changes in the density of the fluid due to variations in temperature. The following solution settings are defined as natural convection defaults.

> **Note:**
>
> **Natural Convection Defaults** does not appear as an option if the **Solution Type** is set to **Temperature Only** or **Flow Only**.

Maximum Iterations per Solve	100
Problem Types	**Temperature** and **Flow** enabled
Include Gravity	Enabled
Under-relaxation	**Pressure** = 0.7 and **Momentum** = 0.3
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Initial Conditions: X, Y, or Z Velocity	-/+ 1e-4 depending on the direction of the **Gravity Vector** defined in the **Design Settings**
Radiation Model	**Discrete Ordinates** enabled
Flow Iterations per Radiation Iteration	10
Angular Discretization Settings	2

Mixed Convection

Select **Mixed Convection Defaults** if your simulation includes forced convention and natural convection. The following solution settings are defined as mixed convection defaults.

> **Note:**
>
> **Mixed Convection Defaults** does not appear as an option if the **Solution Type** is set to **Temperature Only** or **Flow Only**.

Maximum Iterations per Solve	100
Problem Types	**Temperature** and **Flow** enabled
Include Gravity	Enabled
Solve Flow and Energy Equations Separately	Disabled
Under-relaxation	**Pressure** = 0.7 and **Momentum** = 0.3

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Initial Conditions: X, Y, or Z Velocity	-/+ 1e-4 depending on the direction of the **Gravity Vector** defined in the **Design Settings**
Radiation Model	**Discrete Ordinates** enabled
Flow Iterations per Radiation Iteration	10
Angular Discretization Settings	2

Conduction Only

Select **Conduction Only Defaults** if your simulation includes only temperature analysis. The following solution settings are defined as conduction only defaults.

> **Note:**
>
> **Conduction Only Defaults** does not appear as an option if the **Solution Type** is set to **Flow Only**.

Problem Types	**Temperature** enabled and **Flow** disabled
Maximum Iterations per Solve	40
Convergence	**Energy** = 1e-12
Linear Solver Options: Temperature	**Termination Criterion** = 1e-06 and **Residual Reduction Tolerance** = 1e-06

Partition Methods

The mesh is partitioned using a bisection or METIS algorithm. The selected algorithm is applied to the parent region, and then recursively applied to the subregions. For example, to divide the mesh into four partitions with a bisection method, the solver will bisect the entire (parent) region into two child regions, and then repeat the bisection for each of the child regions, yielding four partitions in total. To divide the mesh into three partitions with a bisection method, the solver will "bisect" the parent region to create two partitions—one approximately twice as large as the other—and then bisect the larger child region again to create three partitions in total. METIS uses graph partitioning techniques that generally provide more optimal partitions than the geometric methods.

The mesh can be partitioned using one of the two algorithms listed below. The most efficient choice is problem-dependent, so you can try either method to find the one that is best for your problem. See Ansys Fluent user documentation for more information.

- **Metis** uses the METIS software package for partitioning irregular graphs, developed by Karypis and Kumar at the University of Minnesota and the Army HPC Research Center. It uses a multilevel approach in which the vertices and edges on the fine graph are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coalesced to form a coarse graph. The coarse graph is partitioned, and then uncoarsened back to the original graph. During coarsening and uncoarsening, algorithms are applied to permit high-quality partitions. METIS routines can handle partitioning with model-weighted multiple constraints: when automatically partitioning, solid cell zones are weighted with a default value of 0.1 relative to the fluid cell weighting; when manually partitioning, you can control weighting for cells, solid cell zones, VOF, DPM, and ISAT table lookup.

- **Principal Axes** bisects the region based on a coordinate frame aligned with the principal axes of the region. This reduces to Cartesian bisection when the principal axes are aligned with the Cartesian axes. The algorithm is also referred to as moment, inertial, or moment-of-inertia partitioning.

Adding a Solution Setup to an Icepak Design

To add a new solution setup to a design:

1. Select a design in the project tree.
2. Click **Icepak**>**Analysis Setup**>**Add Solution Setup** .
   - Alternatively, right-click **Analysis** in the project tree, and then click **Add Solution Setup** on the shortcut menu.
   - If you have an existing setup, you can **Copy** and **Paste** it, and then edit parameters.
3. The **Icepak Solve Setup Dialog** dialog box appears.
4. On the **General** tab, enter a **Name** for the solution setup.

> **Note:**
>
> It is recommended that the following names not be used for a Solve Setup:
>
> - Feedback
> - Initial
> - Current
> - Field

> **Note:**
>
> The **Enabled** check box disables/enables a setup. Only enabled setups are run when you select **Analyze All**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the **Maximum Number of Iterations** for which the solution should run.

> **Note:**
>
> The lower right corner also contains a button for HPC and Analysis options. Here you can select or create an analysis configuration.

6. Under **Problem Types**, enable or disable **Temperature** or **Flow** to include or exclude thermal or fluid calcuations in the analysis.

> **Note:**
>
> **Problem Types** is active only when **Temperature and Flow** is selected in the **Solution Type** dialog box.

7. Under **Flow Regime**, select **Laminar** or **Turbulent**. If the flow regime is turbulent, click **Options** to open the **Turbulent Flow Model** dialog and select a model.

> **Note:**
>
> For detailed information regarding turbulence theory, see the Icepak Technical Notes Turbulence section.

8. If solving for radiation, select **Discrete Ordinates** or **Ray Tracing** under **Radiation Model**.

> **Note:**
>
> In addition to enabling radiation in a solution setup, it might be necessary to define radiation settings for boundary conditions that require it (for example, network face nodes). See Assigning Thermal Boundary Conditions for more information.

9. If solving a natural convection model, select **Include Gravity**.
10. Click the **Convergence** tab and enter values for the available parameters based on the selected turbulence model.
11. Click the **Solver Settings** tab.
    - If needed, under **Initial Conditions**, enter initial values for the available parameters based on the selected turbulence model.
    - If needed, under **Import Options**, enable Mesh to import mesh from another Icepak design and **Start/Continue from a previously solved setup** to begin or resume a simulation from a design you have previously solved.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click **Advanced Options** to open the **Advanced Solver Settings** dialog.
13. If needed, enter **Under-relaxation** values.
14. Select **Discretization** schemes.
15. Under **Linear Solver Options**, select or enter the following settings for the available parameters based on the turbulence and radiation models (if applicable).
    - **Type**
    - **Termination Criterion**
    - **Residual Reduction Tolerance**
    - **Stabilization**
16. Click **OK** close the **Advanced Solver Settings** dialog.
17. If solving for solar radiation, click the **Solar Radiation** tab and select a **Solar Load Model** and define the **North Direction Vector**.
    - For **Solar Calculator**, define the **Local Time and Position** and **Illumination Parameters**.
    - For **Specify flux and direction vector**, define the **Solar Flux and Direction Vector** parameters.
18. If solving for radiation, click the **Radiation** tab and enter **Flow Iterations per Radiation Iteration** under **Iteration Parameters**. If you selected the **Ray Tracing** model, enter the **Maximum Radiation Iterations**.
    - For **Discrete Ordinates**, enter **Angular Discretization** values.
    - For **Ray Tracing**, enter the **Faces per Surface Cluster** under **Cluster Parameters** and the **Resolution** under **View Factor Parameters**.
19. Click **OK** to accept the solution settings and close the **Icepak Solve Setup Dialog**.

Renaming a Solution Setup

Do the following to rename a solution setup:

1. In the project tree, under **Analysis**, right-click the setup you want to rename.
   A shortcut menu appears.
2. Select **Rename** from the shortcut menu.
   The setup name text is highlighted in the project tree.
3. Type the new name for the setup, and press **Enter**.

You can also rename the solution setup by changing the text in the **Name** text box of the appropriate dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying a Solution Setup

Solution setups may be copied and pasted within a design or across designs of the same type. This is beneficial for setups having a large number of parameters to specify, or where minor changes to a setup are being evaluated.

Do the following to copy a solution setup:

1. In the project tree, under **Analysis**, right-click the setup you want to copy.

   A shortcut menu appears.

2. Select **Copy** from the shortcut menu.

   The setup parameters are copied to the clipboard.

3. In the project tree, right-click the **Analysis folder to receive the copied setup**.

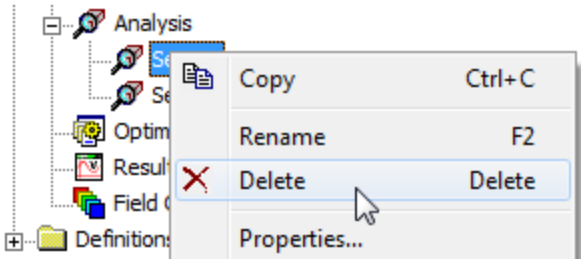
   A shortcut menu appears.

4. Select **Paste** from the shortcut menu.

   The setup parameters are copied to the **Analysis** folder as a new setup.

Deleting a Solution Setup

To delete an existing solution setup, select it in the Project tree, right-click to display the shortcut menu and select **Delete**.



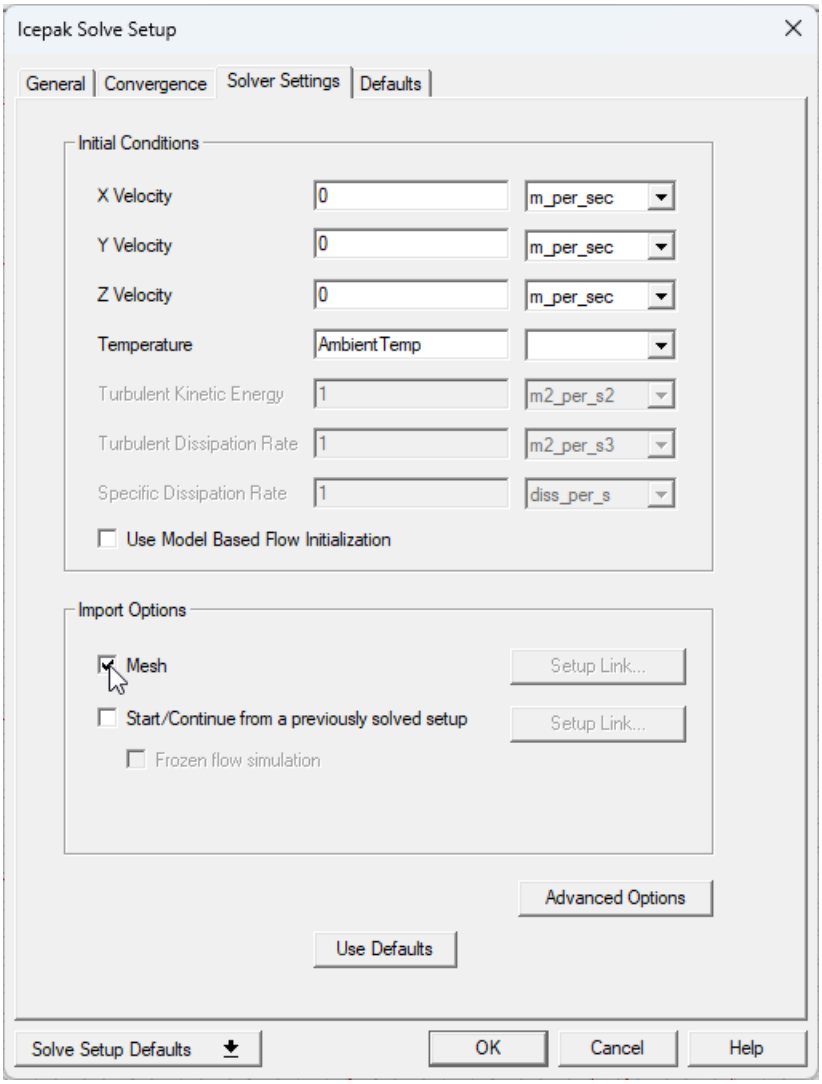
Importing Mesh

An Icepak solution setup can use mesh from an existing Icepak design. Mesh from the selected design is imported and used in the simulation. The mesh from the source design must conform to the target design's model geometry.

1. In the **Project Manager**, double-click a solution setup to open the **Icepak Solve Setup** dialog box.

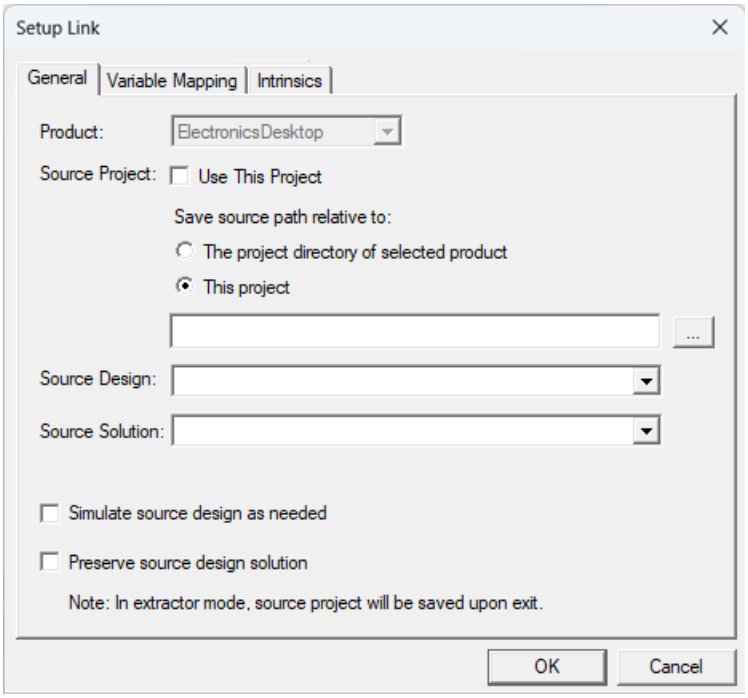
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. On the **Solver Settings** tab under **Import Options**, enable **Mesh** and click **Setup Link**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

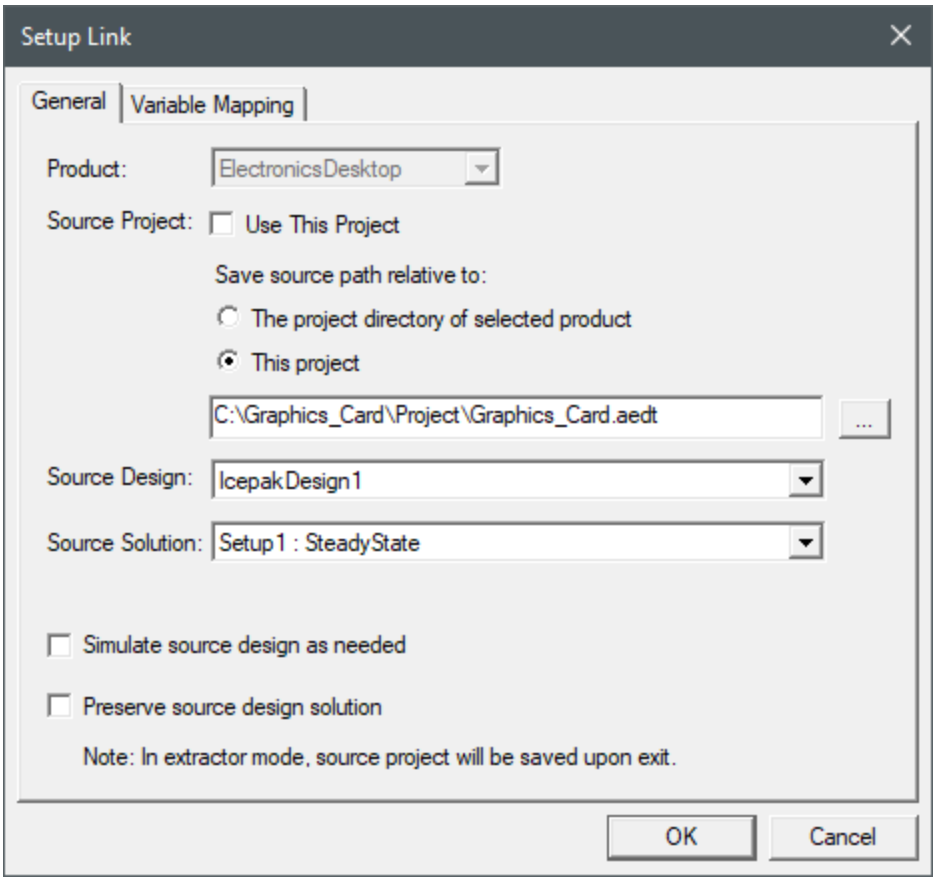
The **Setup Link** dialog box is displayed.



3. Specify the project that has the source design in one of the following ways:
    a. Check **Use This Project** to pick a design from a current working project.
    b. Click the ellipsis and browse your computer to pick the design.

> **Note:**
>
> The option **Use this project** points to the directory relative to where your current working project is located.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

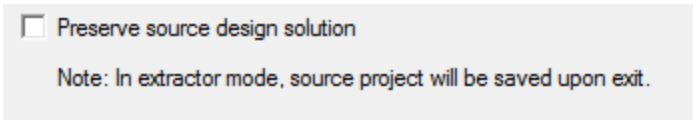


4. Pick the source design from the **Source Design** drop-down menu which lists all the designs in the specified project.
5. Pick the setup type from the **Source Solution** pull-down menu (SteadyState or Transient).
6. Select the last two check boxes as needed and click **OK**.

> **Note:**
>
> Go to Simulate source design as needed, for more information about that option.

Preserve source design solution

This is the last option on the **Setup Link** dialog box. By default, the source design results are not saved. Select this option to save the source design results.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simulate source design as needed

Select this option if you modify the source design and you want an updated source design solution to be linked to the target design. If you select this option, make sure that you also invalidate the cached link data in the target. To do this, right-click **Analysis** and select the option **Clear Linked Data** from the short-cut menu.



7. Click the **Variable Mapping** tab and ensure the **Variable Mapped to Source Design** values are as desired for each variable.

> **Note:**
>
> You can click **Map Variable by Name** to compare the source design variables to the Icepak design variables. Any variables with the same name are mapped so that the source design conforms to the Icepak design.

8. Click **OK** to close the **Setup Link** dialog.

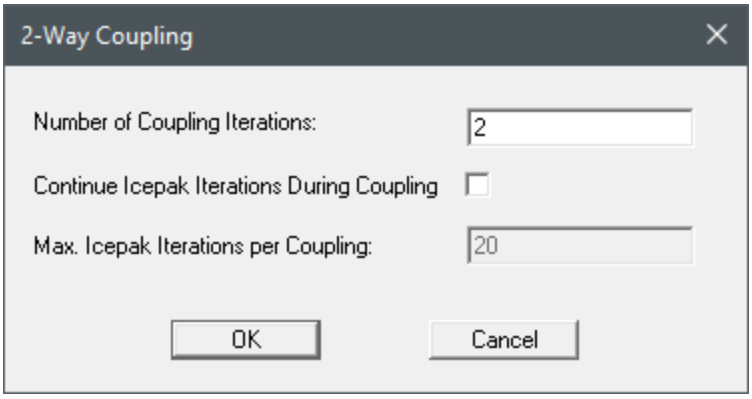
Adding Two-Way Coupling to an Icepak Design

If your Icepak design has an EM Loss thermal boundary condition, a PCB 3D native component, or a layout component (beta) linked to an HFSS, HFSS 3D Layout, Maxwell, or Q3D design, you can perform two-way coupling to produce more accurate results. Loss data calculated by the source HFSS, HFSS 3D Layout, Maxwell, or Q3D design is consumed by Icepak when calculating thermal data in the current coupling iteration. The thermal data is then consumed by the source design in the next coupling iteration's calculation.

The following design types' solution types and solution setups are supported:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	EM Loss Boundary Condition	PCB Native Component	Layout Component (Beta)
HFSS Driven Modal	Supported		
HFSS Driven Terminal	Supported		
HFSS Eigenmode	Supported		
HFSS 3D Layout Slwave DCIR Solution Setup		Supported	
HFSS 3D Layout HFSS Solution Setup (DC)		Supported	Supported
HFSS 3D Layout HFSS Solution Setup (AC)		Supported	Supported
Maxwell Magnetostatic	Supported		
Maxwell AC Magnetic	Supported		
Maxwell Transient	Supported		
Maxwell DC Conduction	Supported		
Q3D Harmonic	Supported		
Q3D DC	Supported		
Q3D AC	Supported		



Number of Coupling Iterations	The **Number of Coupling Iterations** specifies the numbers of times Icepak and the source design will both run an analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continue Icepak Iterations During Coupling	**Continue Icepak Iteration During Coupling** enables the Icepak solver to resume the analysis using the latest loss data reusing the solution results from the previous coupling iteration for solution initialization. The solution continues from the last iteration of the previous solution run.
Max. Icepak Iterations per Coupling	The **Max. Icepak Iterations per Coupling** specifies how many iterations the Icepak solver runs during each coupling iteration. This setting is applied when **Continue Icepak Iteration During Coupling** is enabled.

To add two-way coupling to a solution setup:

> **Note:**
>
> In the HFSS, Maxwell, or Q3D design, you must enable **Include Temperature Dependence** and **Enable Feedback** in the **Temperature of Objects** dialog box, which you can open by right-clicking the design and selecting **Set Object Temperature**. In HFSS 3D Layout, you must enable **Include Temperature Dependence** and **Enable Feedback** in the **Set Temperature** dialog box, which you can open by right-clicking the design and selecting **Set Temperature**. For an HFSS 3D Layout HFSS solution setup, you must enable **Save Fields**.

1. Add a solution setup to the Icepak design.
2. In the **Project Manager** under **Analysis**, right-click the solution setup and select **Add 2-way Coupling**.
3. In the **2-Way Coupling** dialog box, enter the **Number of Coupling Iterations**.
4. If desired, select **Continue Icepak Iteration During Coupling** and enter the **Max. Icepak Iterations per Coupling**.

   > **Note:**
   >
   > For transient simulations, after each coupling iteration from the source design, loss data is imported into the target Icepak design for a full transient simulation before the next coupling iteration begins.

5. Click **OK**. The two-way coupling appears under the solution setup in the **Project Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Right-click on a two-way coupling and select **Delete** to remove it. Removing two-way coupling from a solution setup results in one-way coupled designs.

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 tree.
2. Right-click the icon for the solution setup. In the shortcut menu that appears, click **Disable Setup**. To enable the analysis setup, open the shortcut menu again and select **Enable Setup**.

Alternatively, you can enable or disable a solution setup in the following ways:

- In the **Properties** window, check or uncheck **Enabled** to enable or disable the solution setup.
- You can double-click a solution setup in the project tree to open the **Icepak Solve Setup Dialog**. On the **General** tab, check or uncheck **Enabled** to enable or disable the solution setup.

Specifying the Number of Processors

If you want to use more than one processor, see HPC and Analysis Options.

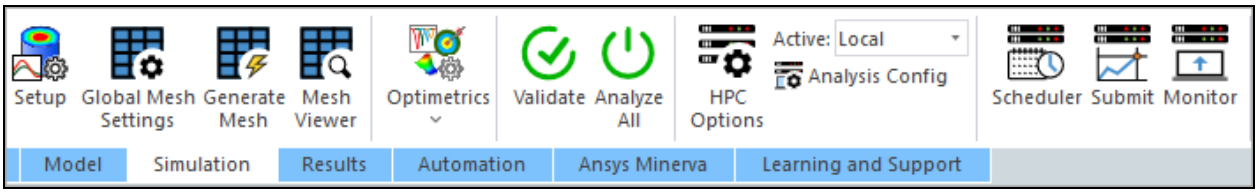
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

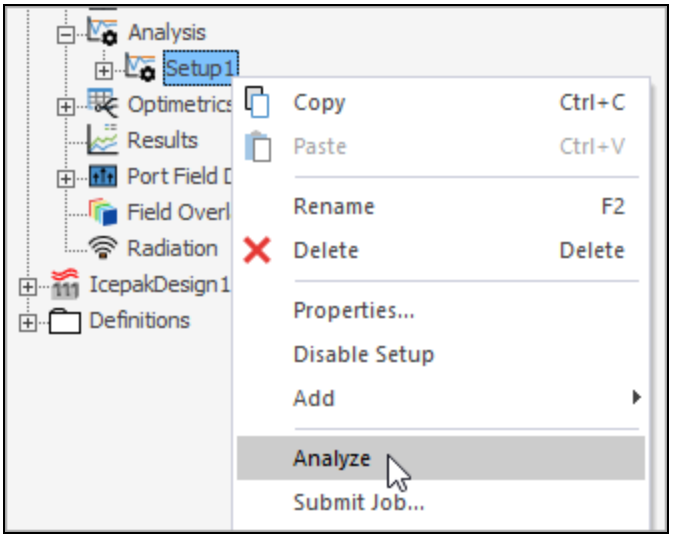
16 - Running Simulations

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.

> **Note:**
>
> For detailed information regarding Icepak simulation theory, see the Icepak Technical Notes Governing Equations section.

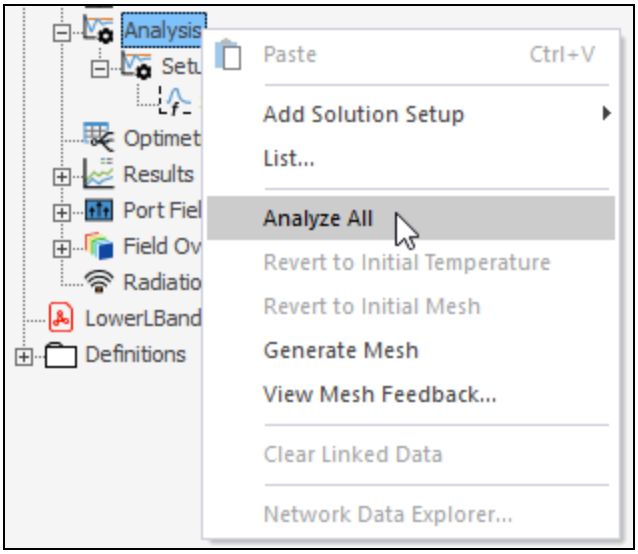


In general, the **Analyze** command on the shortcut menu applies to the selected setup and associated sweeps, if any, or to a selected sweep. To use this command, right-click on a setup or sweep in the Project Manager and click the command on the context menu.



The **Analyze All** command applies to all enabled setups and sweeps at or below the level invoked in the Project Manager. To use this command, either click **Icepak** > **Analyze All** or right-click the **Analysis** icon in the *Project Manager* and select **Analyze All**.

Ansys Electromagnetics Suite 2026 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?

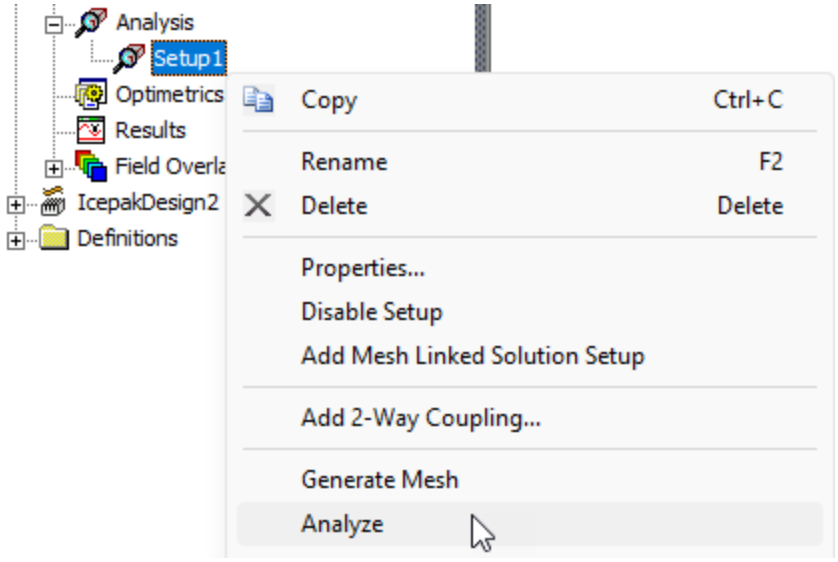
- Local Analysis
- Solve a single setup with or without sweeps (when applicable)
- Enable a queue so that multiple simulations can run sequentially as resources become available.
- Run more than one simulation, whether multiple setups, or multiple sweeps under a single setup, or setups with dependencies.
- 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

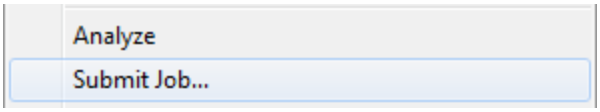
Before solving, the solver checks the design for the following issues:

- Mesh regions intersecting one another
- Sheets assigned a thin conducting plate boundary condition intersecting a mesh region
- Polygonal or inclined geometry intersecting a mesh region
- Network faces blocked by a hollow object (an object with **Solve Inside** disabled)
- 2D sources placed over sheets assigned thin conducting plate boundary conditions

If a mesh region intersects another, the analysis is aborted with an error message. If any other issues are found, a warning message is displayed in the **Message Manager**, and the analysis proceeds.

The 3D field solution is computed inside the structure for a solution. For a select sweep, it is computed for the sweep variables.

When you right-click a Setup the shortcut menu also includes the **Submit Job...** command.



Ansys Electromagnetics Suite 2026 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.

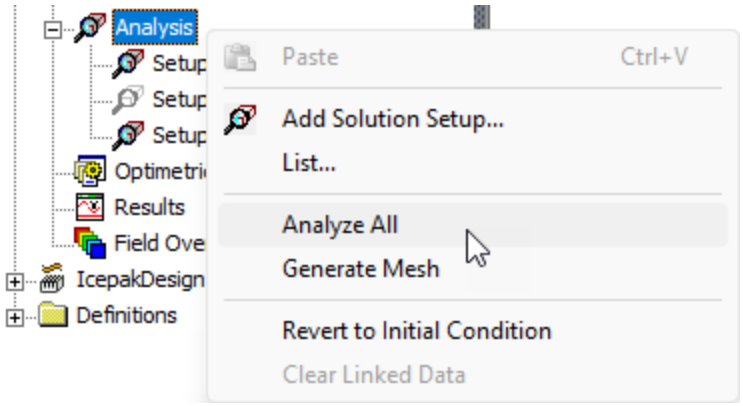
Running More than One Simulation

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 **Icepak** > **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

Note:

You can simulate all designs in a project by clicking **Project** > **Analyze All**.

Submitting a Job

Ansys Electromagnetics Desktop supports its own Remote Simulation Management (RSM) and Ansys Cloud Burst Computing along with other High Performance Computing (HPC) software

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 that 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 postprocessing 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.

> **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).

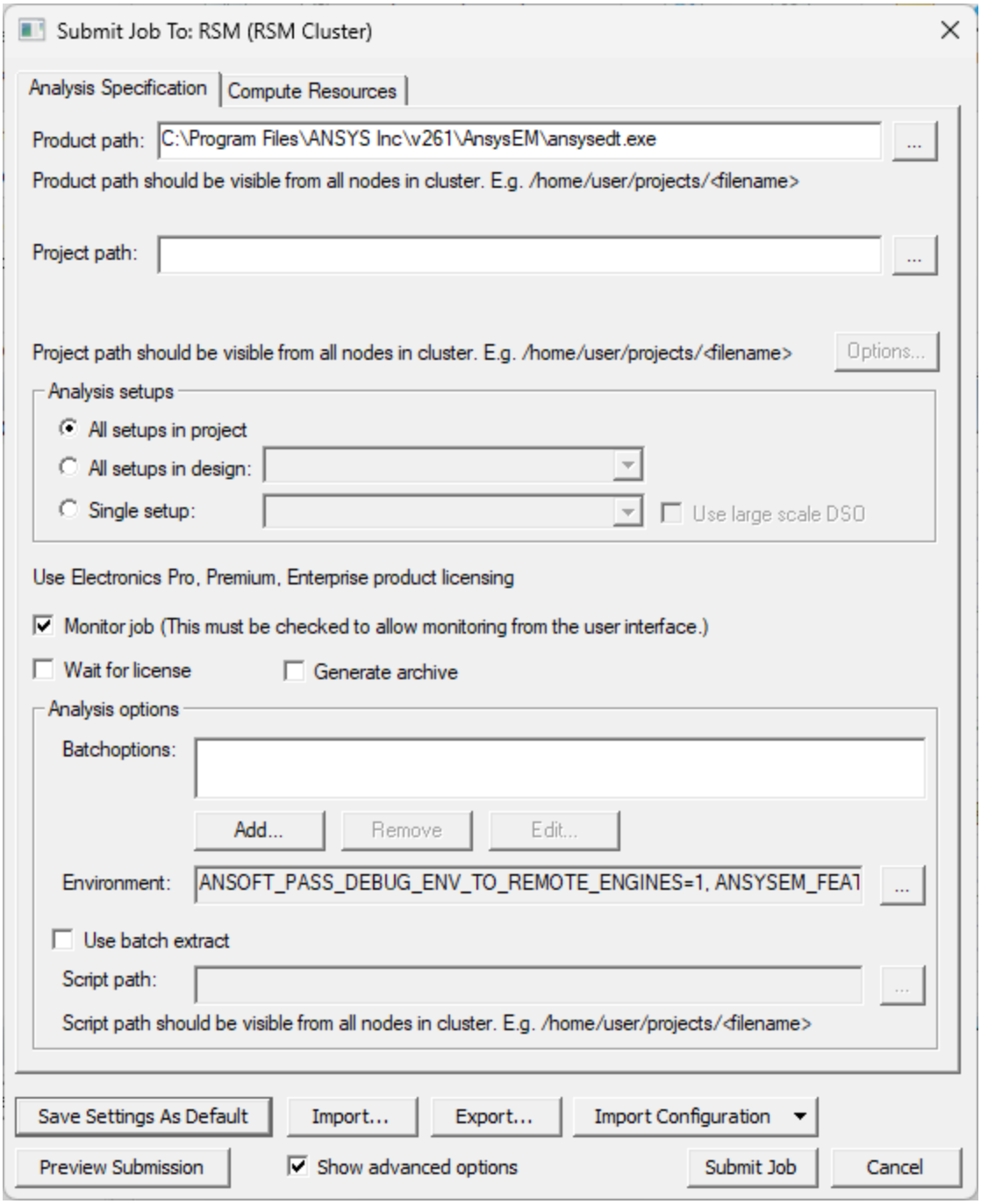
To submit a job:

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 **Icepak** > **Submit Job**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on which scheduler is selected, 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*.

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.

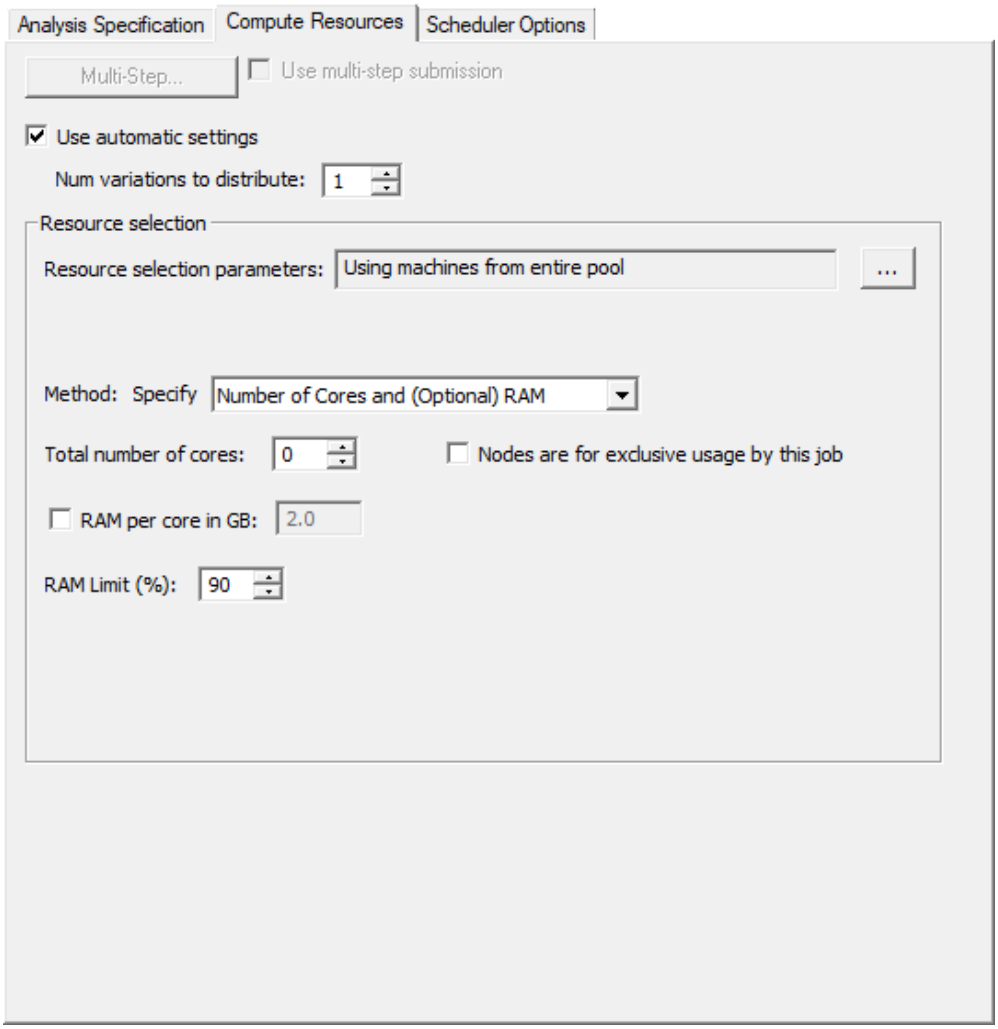
   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See Large Scale DSO for Parametric Analysis.

   > **Note:**
   >
   > Depending on which scheduler is selected (RSM, Windows HPC), the Submit Job window will include a **Generate archive** option, which is used to create an archive for the submitted job when it is complete. The project file, batchinfo directory, any external files, and the results directory are combined into the archive. If there already exists an archive file with the name <project>.aedtz, then <project>_1.aedtz, <project>_2.aedtz, *etc.*.. will be written instead. This function will not overwrite an existing archive.
   >
   > Note that if an archive file is selected, this option will be renamed **Repackage results into archive**.

Ansys Electromagnetics Suite 2026 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.

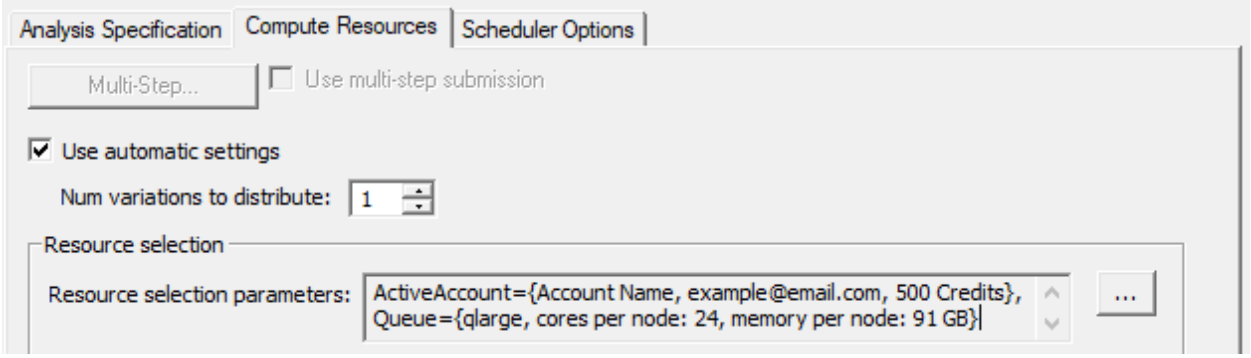
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **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 2026 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 Burst Computing 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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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 a profile of the simulation as well as convergence data during or after the solution process.

- The solution profile includes meshing information (memory, times, and mesh counts per region), solver times (input, initialization, and solve), surface and volume EM loss data, exported AC loss data from HFSS 3D Layout, and total solution process.
- The solution residuals include those for continuity, velocity, energy, and joule heating (if enabled).
- The solution monitor includes data plotted for any defined thermal and flow monitor points.
- The solution overview includes thermal and flow data for boundary conditions and fan components.

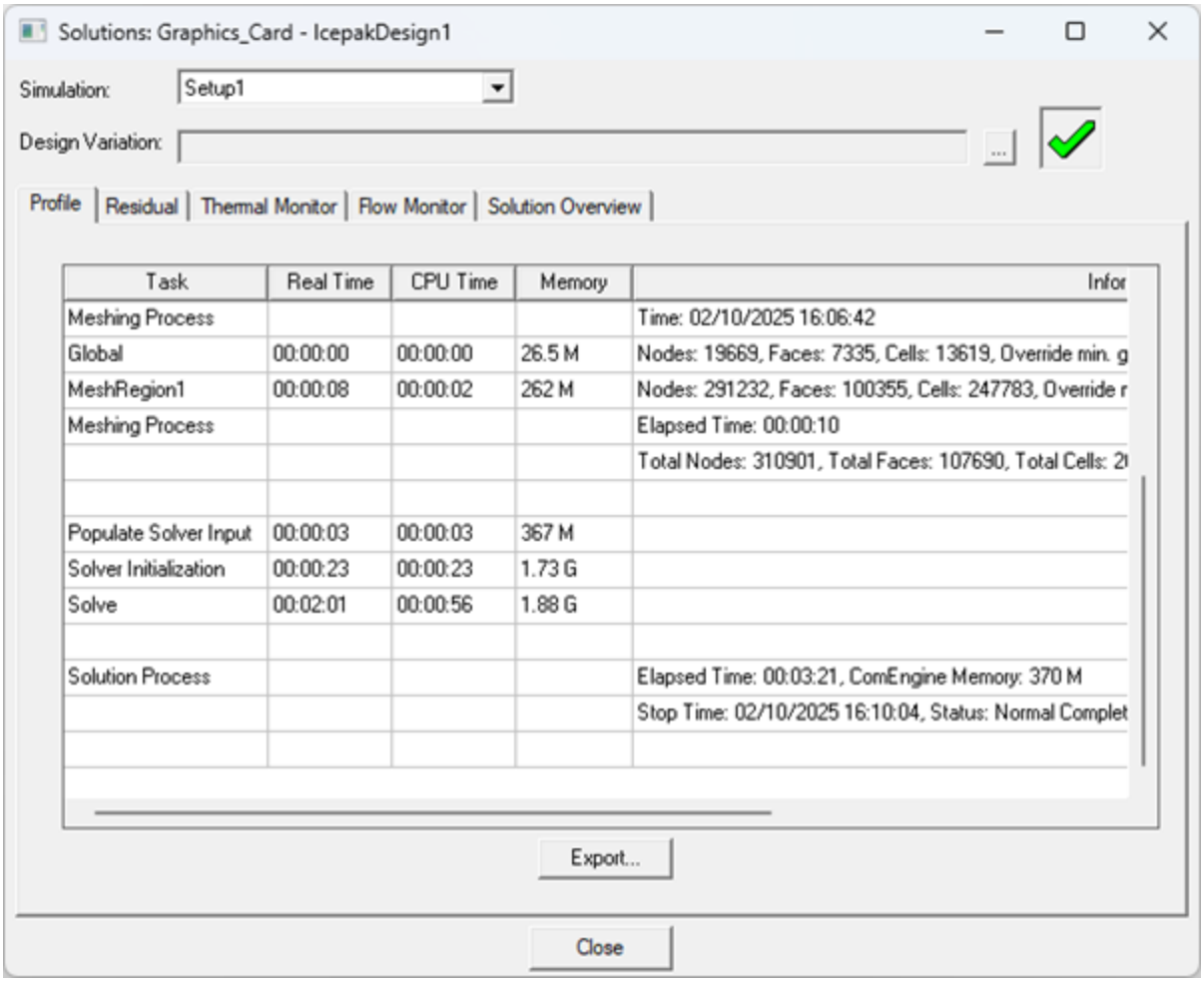
> **Note:**
>
> For solution overviews, Heatsink, PCB, CTM, and 3D-IC components are not currently supported.

To view the solution data:

1. In the **Project Manager**, expand **Analysis**.
2. Right-click the solution setup and select **Profile**, **Residual**, **Thermal Monitor**, **Flow Monitor**, or **Solution Overview**.
3. In the **Solutions** dialog box, toggle between the **Profile**, **Residual**, **Thermal Monitor**, **Flow Monitor**, and **Solution Overview** tabs to view the information. See "Solution Overview" on page 16-15 for details on the data displayed on the **Thermal**, **Flow**, and **Fan** tabs.

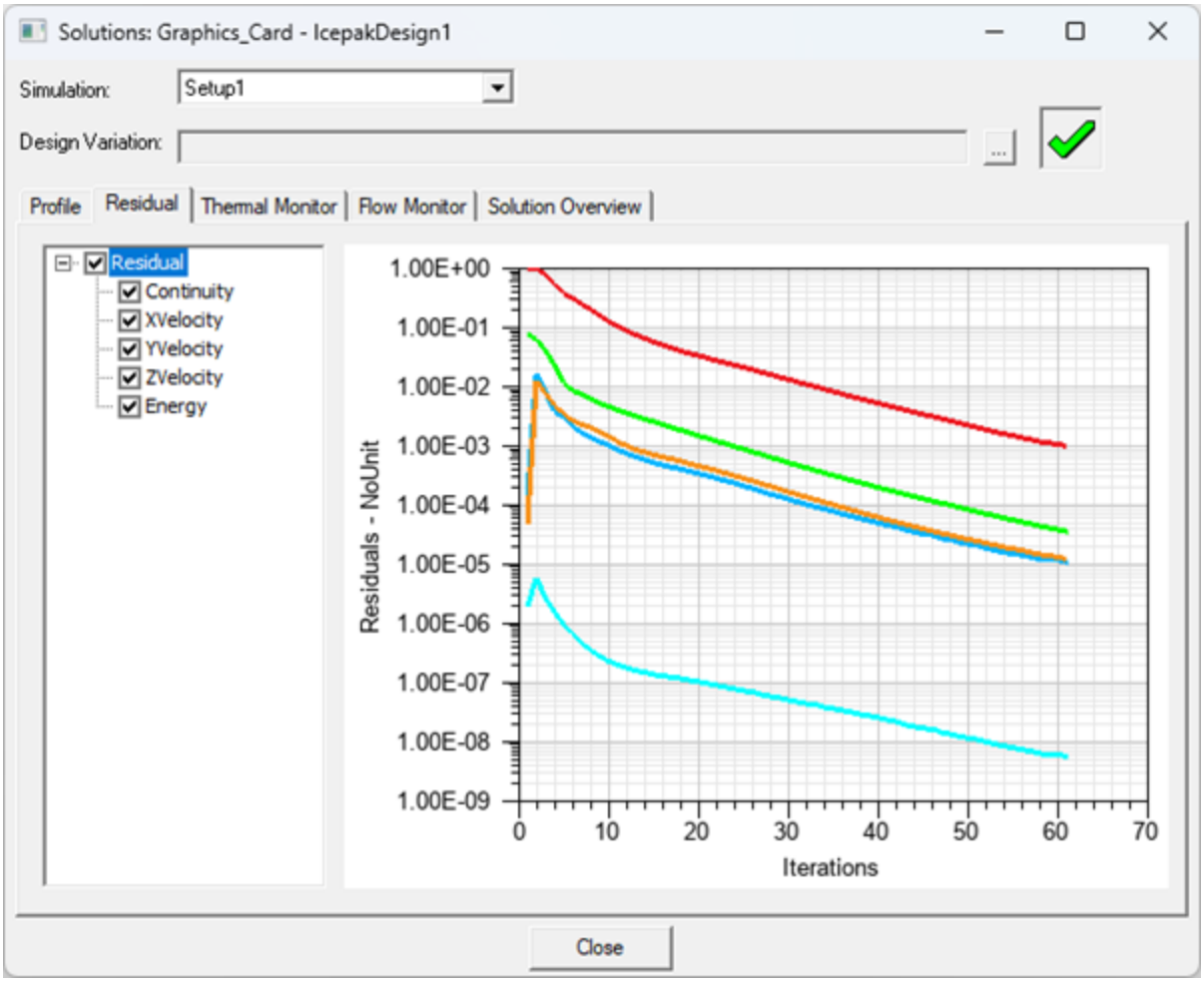
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. For Parametric trials, click [**...**] next to **Design Variation** to open the **Set Design Variation** dialog box and choose the variation for which to display data.

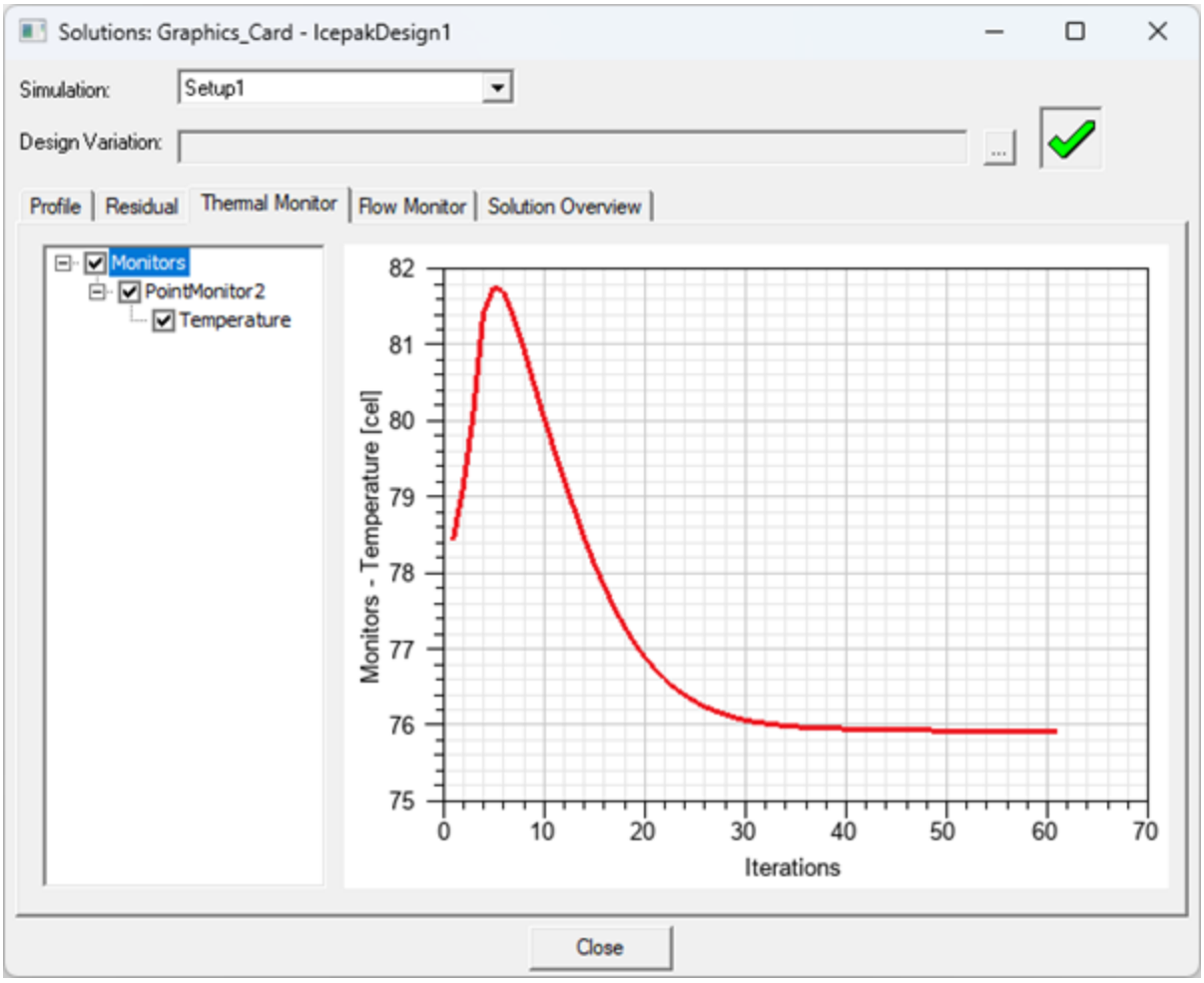


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

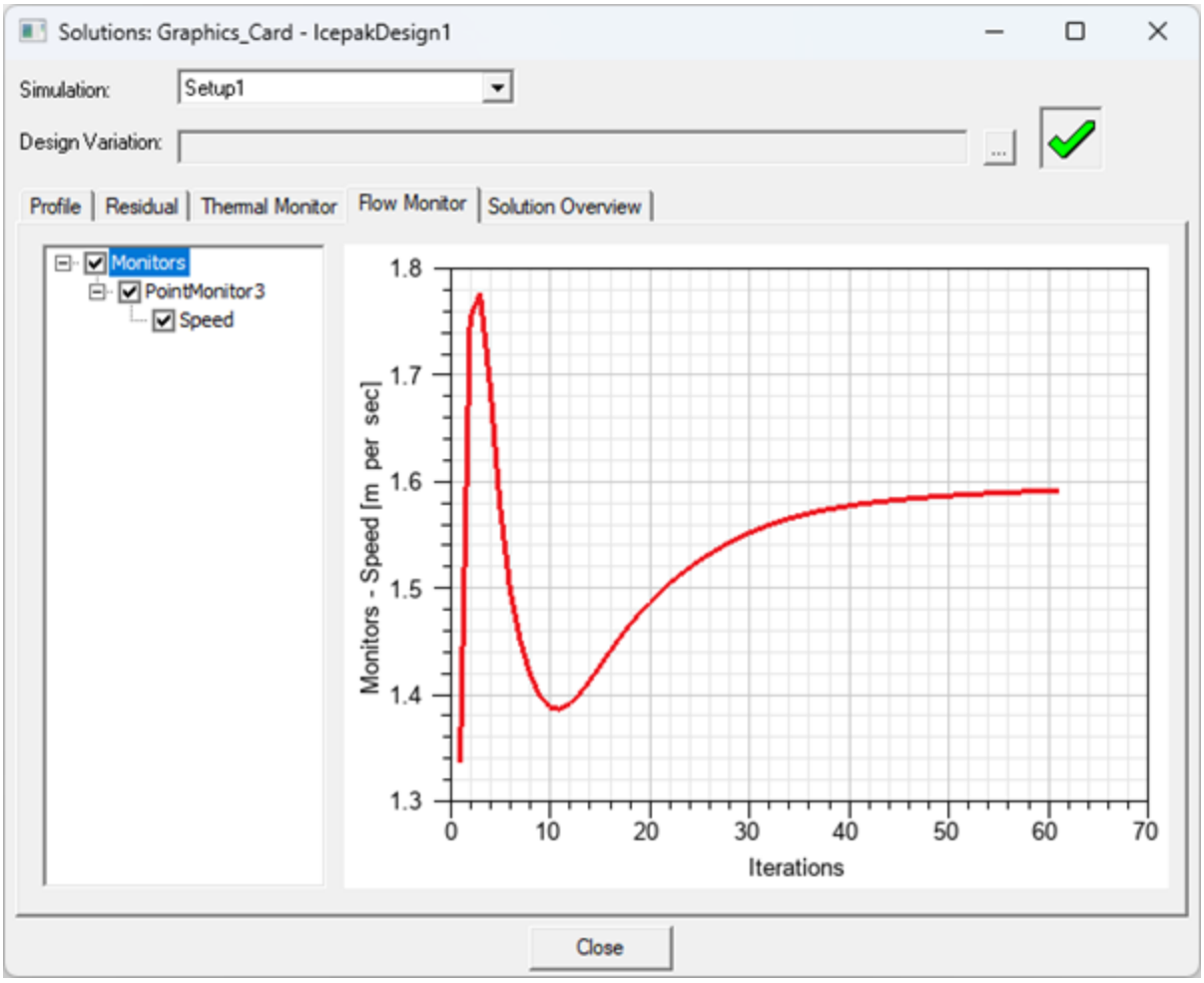
Tip: Toggle the top check box to display or hide all residual and monitor curves. To select multiple curves, press and hold **Shift** and click.



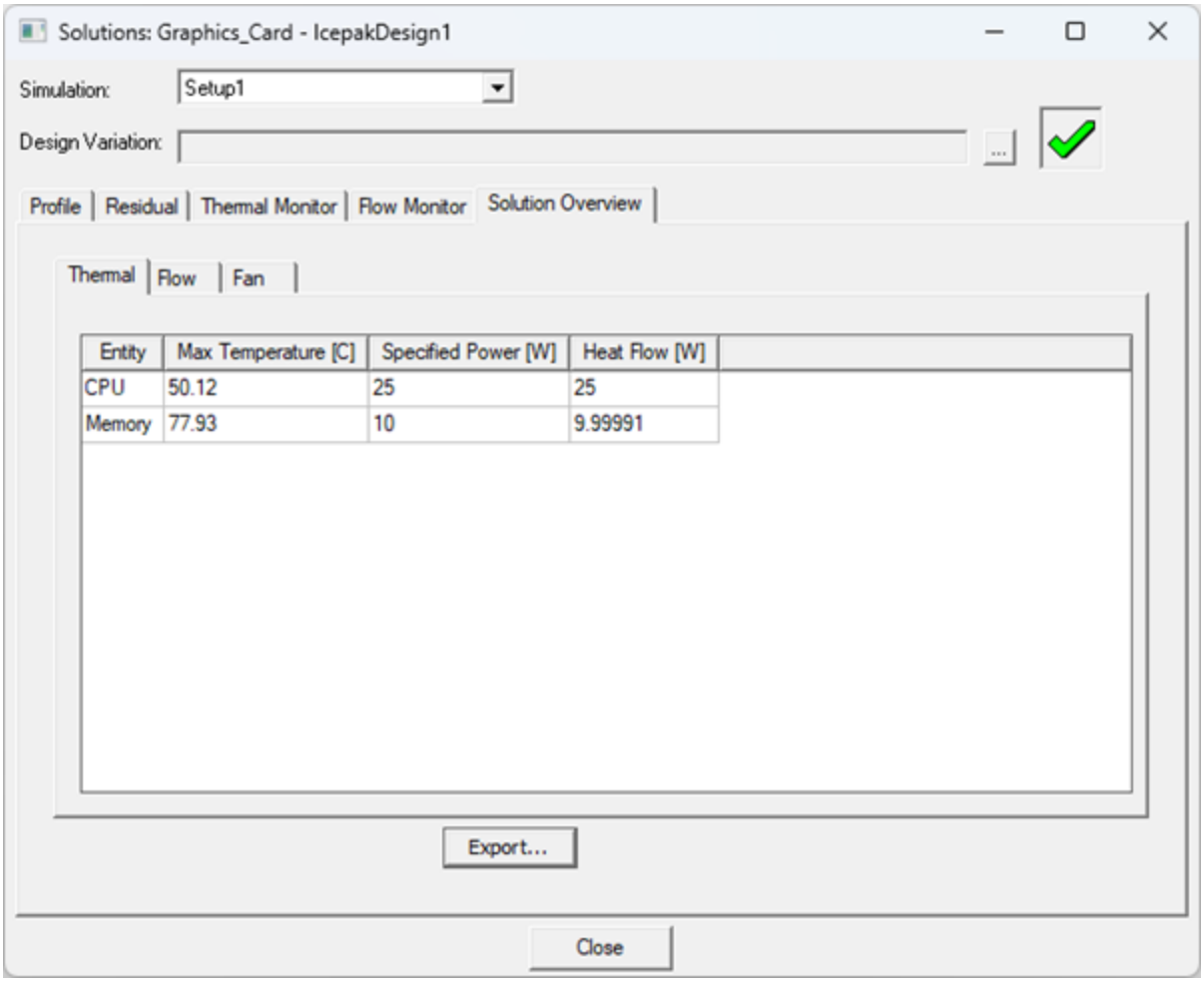
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Solution Overview

Note:

The solution overview has the following limitations:

- When **Mesh Fusion** is enabled, the solution overview may report incorrect values (partial results) for some or all boundary conditions.
- The solution overview does not report values based on full mesh. It reports values based on reduced mesh, so results may be incomplete in some instances. It is recommended to use a Fields Summary report to review the results in such instances.

The solution overview has three tabs that display thermal, flow, and fan data. Toggle between the **Thermal**, **Flow**, and **Fan** tabs to view the information. Click **Export** to generate a plain text

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(.txt) file containing the data. For parametric trials, click [**...**] next to **Design Variation** to open the **Set Design Variation** dialog box and choose the variation for which to display data.

> **Note:**
>
> Data in the solution overview is cleared if you delete a boundary condition or component in the report or if you clean up solution data.

Thermal Tab

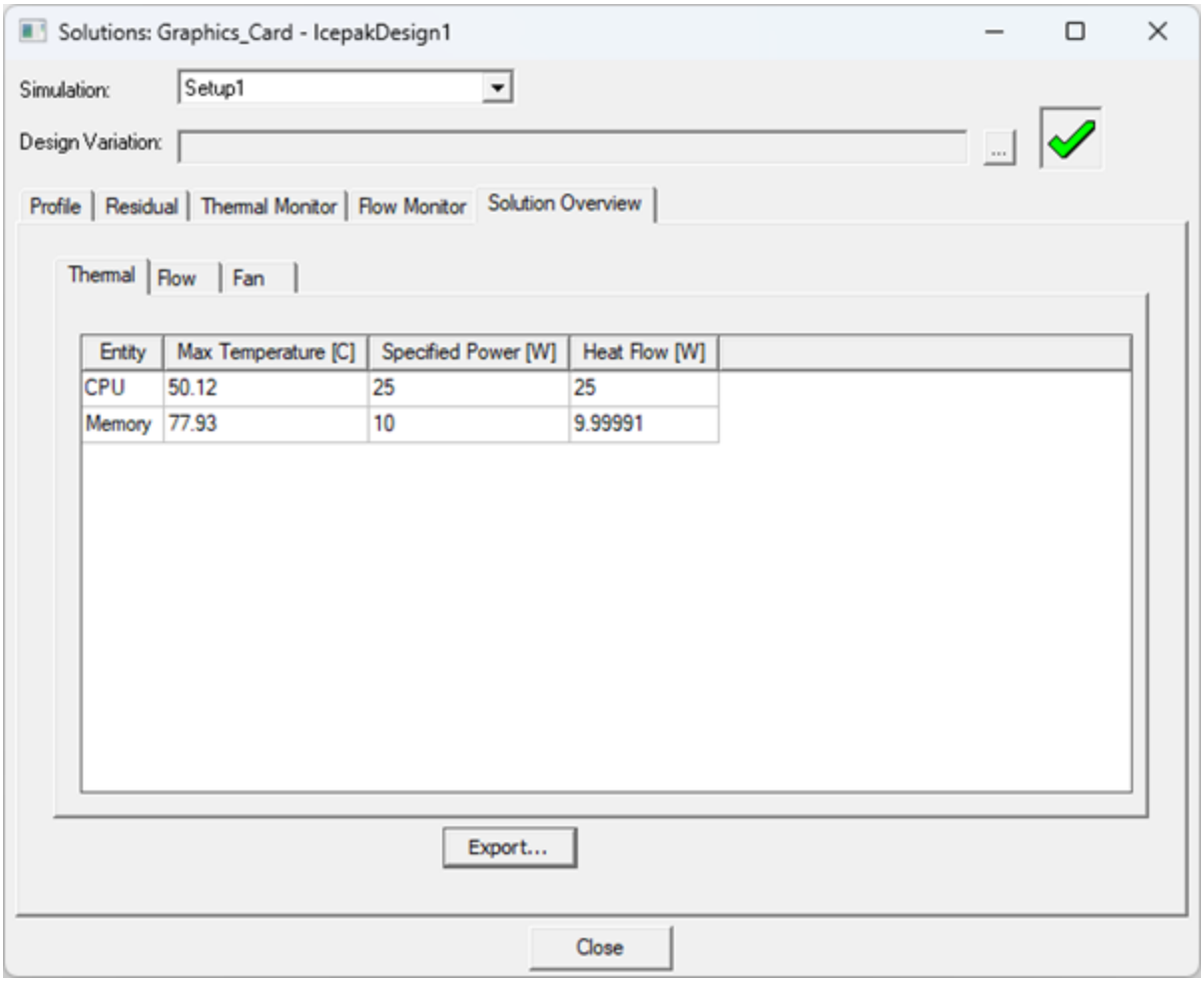
The **Thermal** tab displays data for boundary conditions, including Blocks, Networks, Plates, Sources, Em Losses, Resistances, and Walls. Data is also displayed for external 3D components with supported thermal boundary conditions.

- Max Temperature
- Specified Power

  > **Note:**
  >
  > If a dataset is used to specify power, a default value of zero is displayed.

- Heat Flow

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



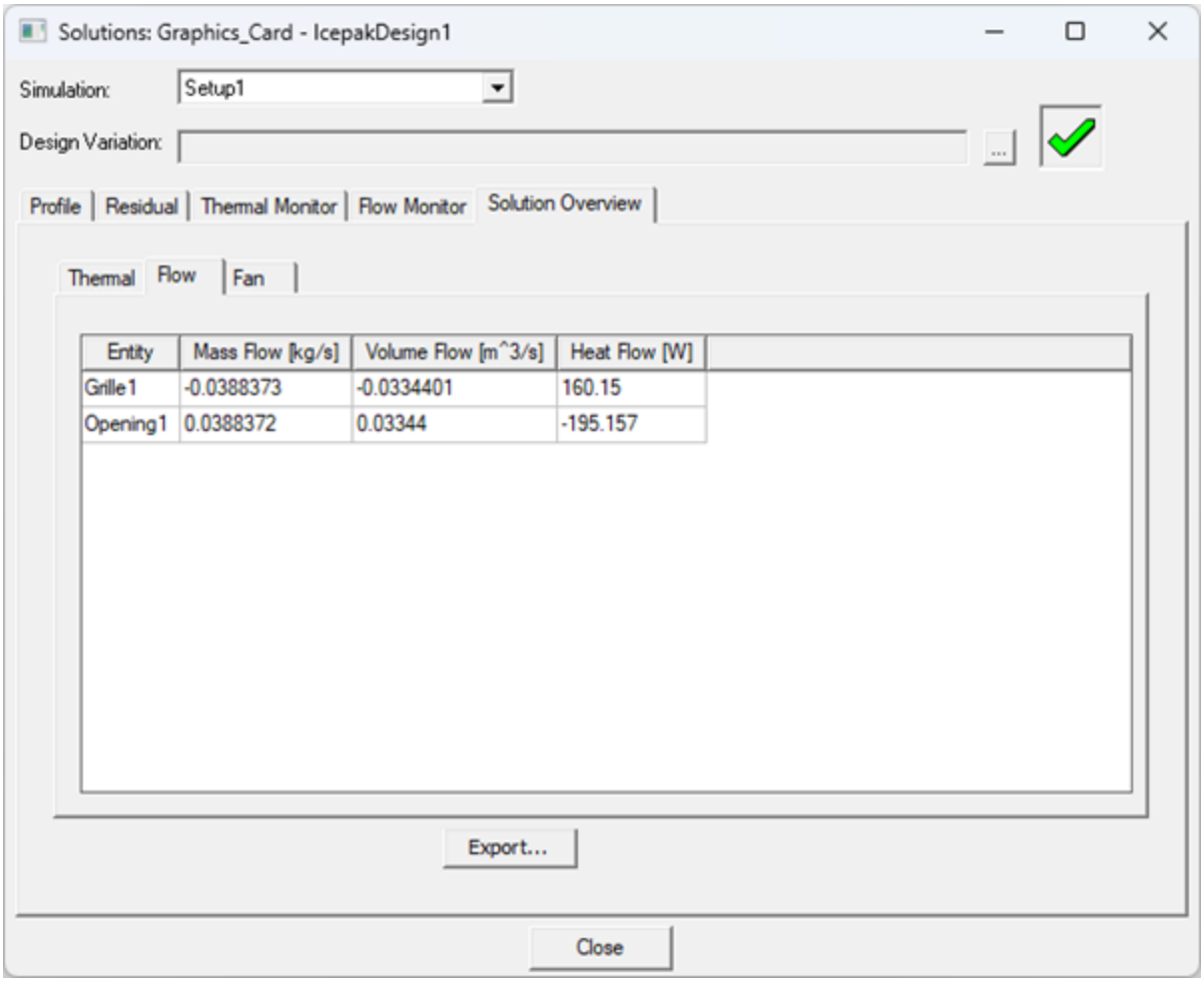
Flow Tab

The **Flow** tab displays data for boundary conditions, including Openings, Grilles, and Blowers. Data is also displayed for external 3D components with supported flow boundary conditions.

- Mass Flow
- Volume Flow
- Heat Flow

> **Note:**
>
> For openings on the external boundary of the computational domain, the heat flow rates are reported at reference temperature in Fluent (25°C).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

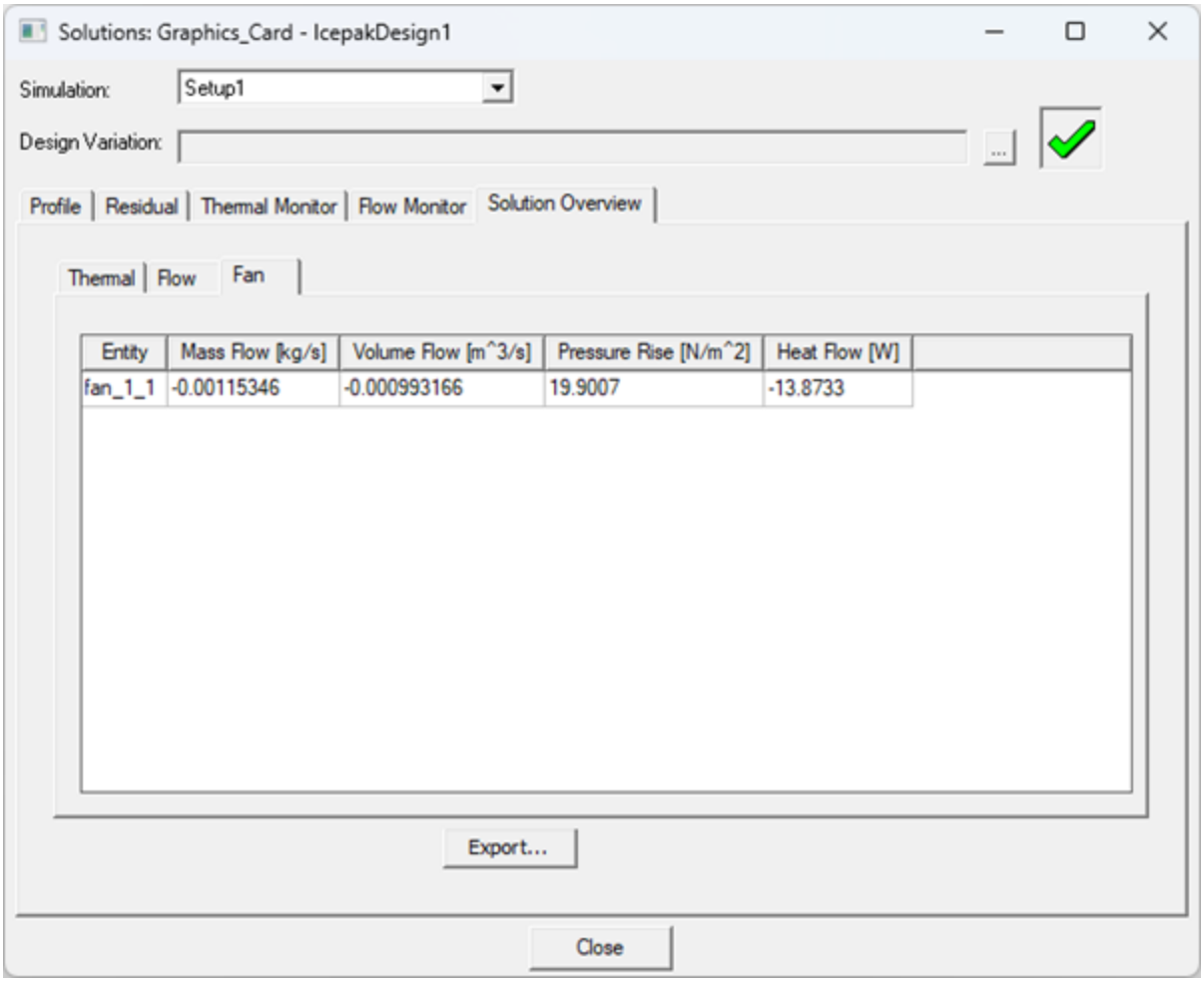


Fan Tab

The **Fan** tab displays data for fan components in the design (if any). Data is also displayed for external 3D components with fan components.

- Mass Flow
- Volume Flow
- Pressure Rise
- Heat Flow

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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	
Above Normal	
Normal	Default
Below Normal	
Lowest Priority	

3. Click **OK**.

Related Topics

Monitoring Queued Simulations

Reverting an Icepak Simulation

When analyzing an Icepak solution setup for a subsequent time, the analysis resumes from the last iteration completed normally. A simulation is completed normally when it:

- Reaches convergence
- Reaches maximum iterations
- Is stopped with a Clean Stop operation

A previously solved solution that ended with one of the ways listed above can be continued unless the following has occurred within the project:

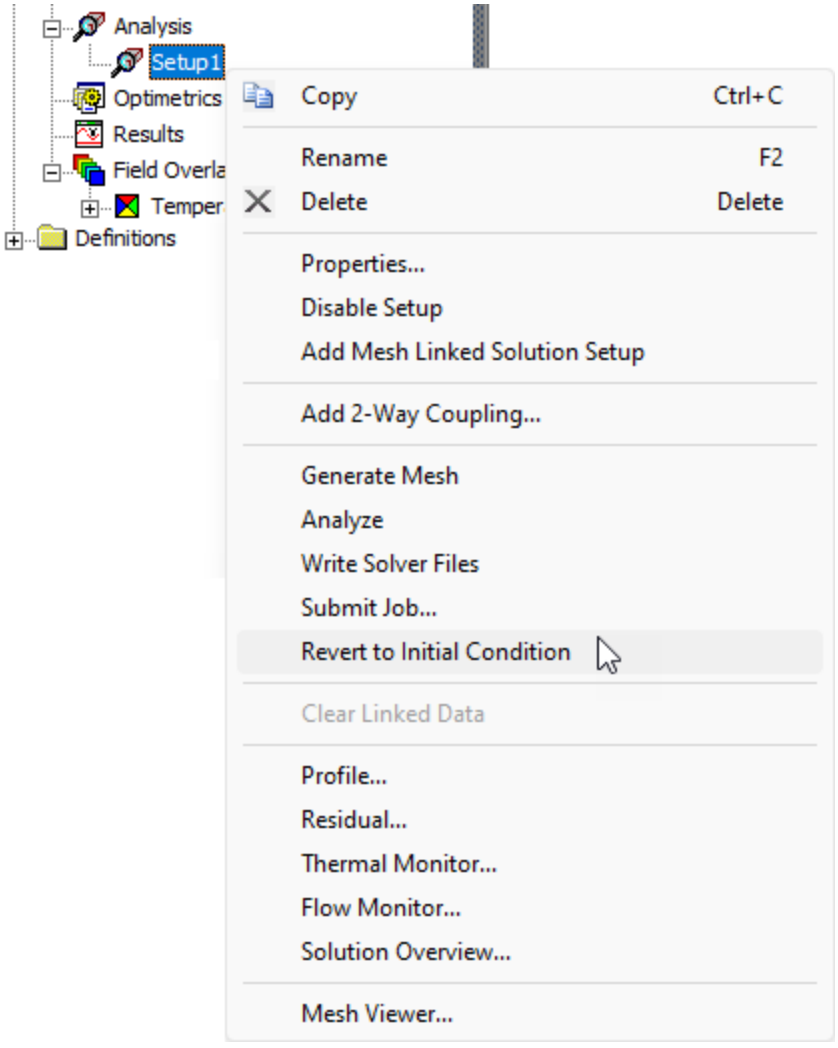
- Geometry changes
- Mesh region changes

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reassignment of boundary conditions
- Reassignment of materials

> **Note:**
>
> Iterations completed before an **Abort** operation are not saved and must be solved again.

If you want to start the simulation over without continuing from the last iteration solved, you can right-click the solution setup in the **Project Manager** and select **Revert to Initial Condition**. This option removes the current solution data so that the subsequent analysis of the solution setup begins at the first iteration.



Ansys Electromagnetics Suite 2026 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 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. Solutions completed before the stop request are retained.

Linux

For Linux, you can use TERM commands. Sigterm handling is done in Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

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.

Writing Solver Files

In Icepak designs, you can export solver files based on a solution setup without initializing a simulation. You can review the solver files in the export directory:

[ProjectDirectory]*ProjectName*.aedtexport*DesignName**SetupName*\SolverFiles

1. In the **Project Manager**, expand **Analysis**.
2. Right-click a solution setup and select **Write Solver Files**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17 - Post Processing and Generating Reports

As Ansys Electronics Desktop completes a solution, you can display and analyze the results:

- View solution data including profile data and solution monitor data. Solution data can also be viewed while Ansys Electronics Desktop is generating a solution.
- View analysis results for Optimetrics solutions.
- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Create 2D or 3D reports.
- Create animations of field quantities, the finite element mesh, and defined project variables.

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.
- Solution monitor data, including residuals and point and surface monitors.

To access the **Solution Data** window, in which the information above can be accessed, do one of the following:

- Click **Icepak**> **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

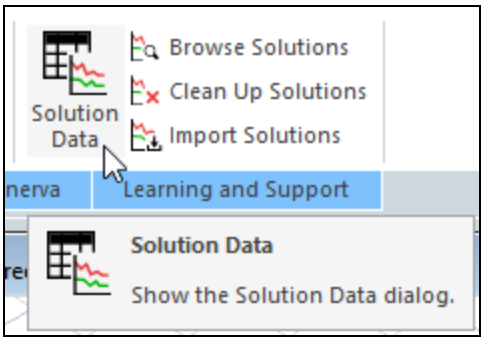
At any time during or after the solution process, you can examine the computing resources or profile data that were used by Ansys Electronics Desktop solvers during the analysis. The profile data is essentially a log of the tasks performed by Ansys Electronics Desktop during analysis.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The log indicates the length of time each task took and how much physical and disk memory was required.

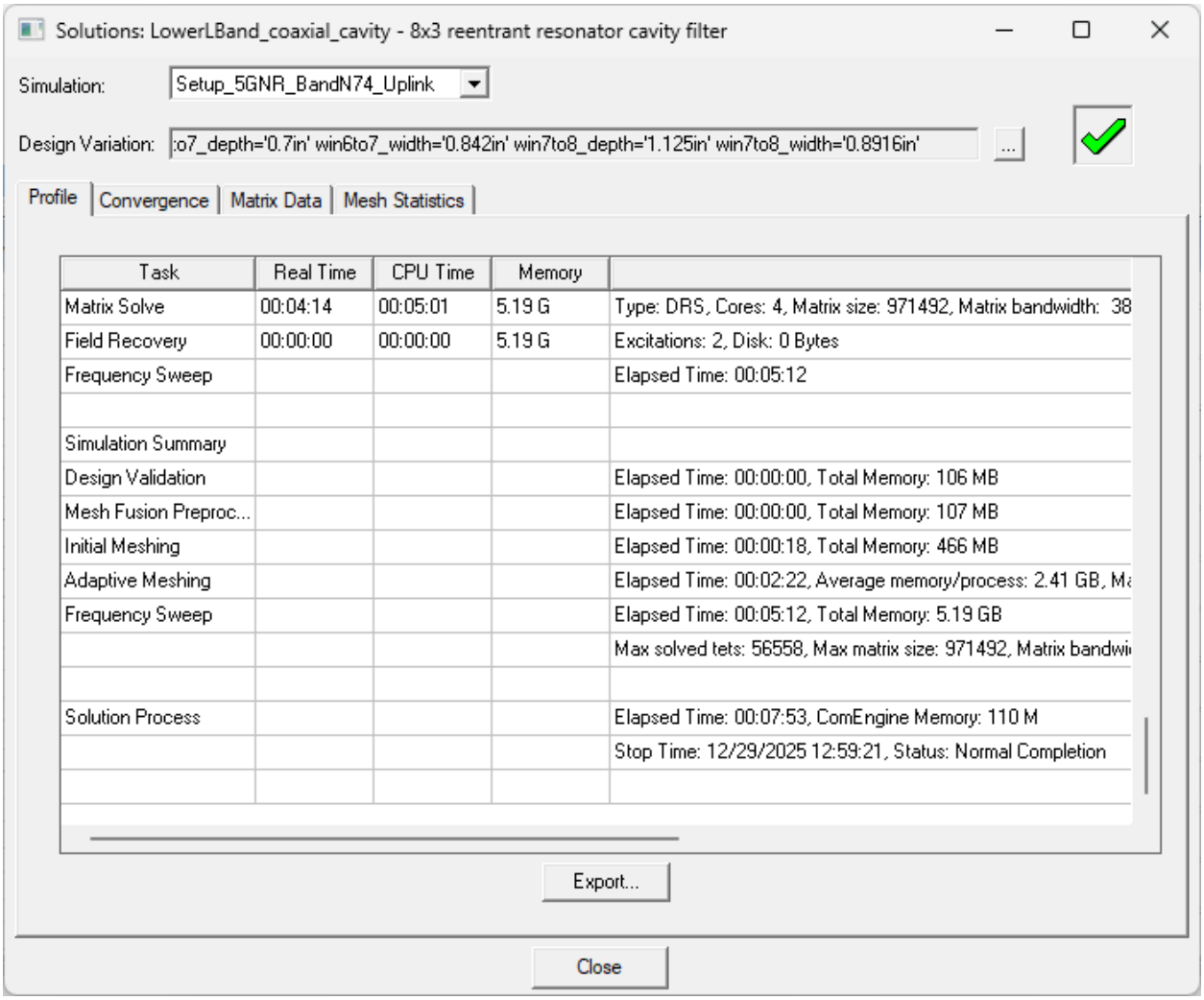
1. In the project tree, right-click a solution setup, and select **Profile**.

   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.



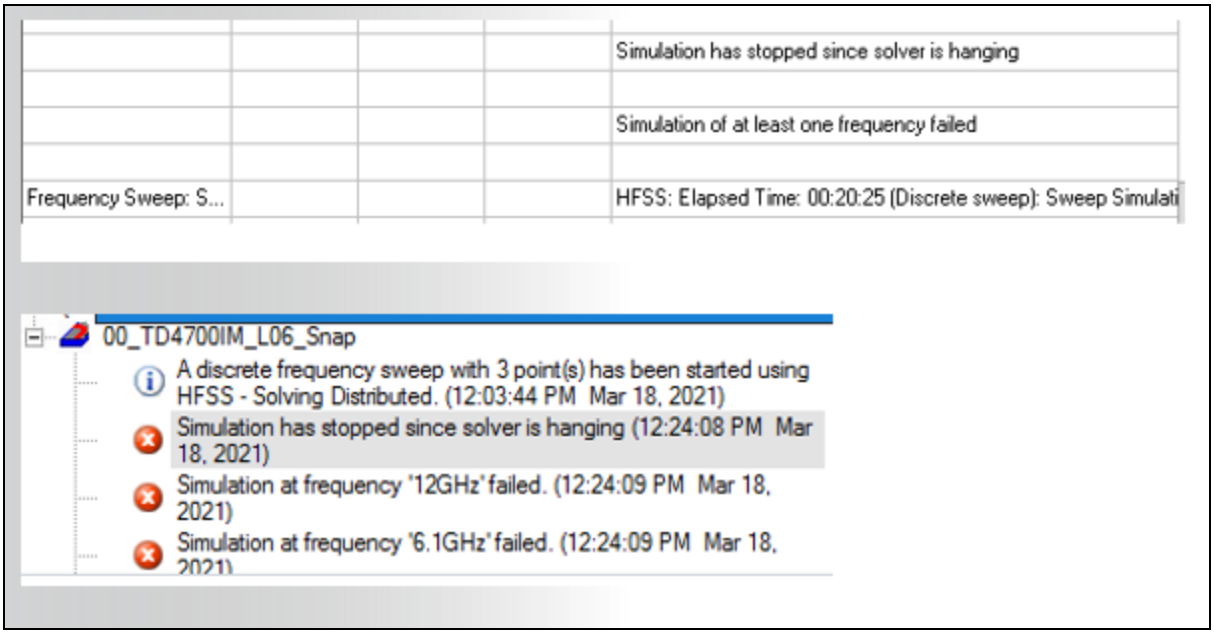
   The displayed data depends on the type of problem and solution setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the simulation hangs, equivalent messages appear in the Message window and the Solution **Profile** tab.

Ansys Electromagnetics Suite 2026 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 Solver Input, Solve, Solution Process, and Start.
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:

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Save**.

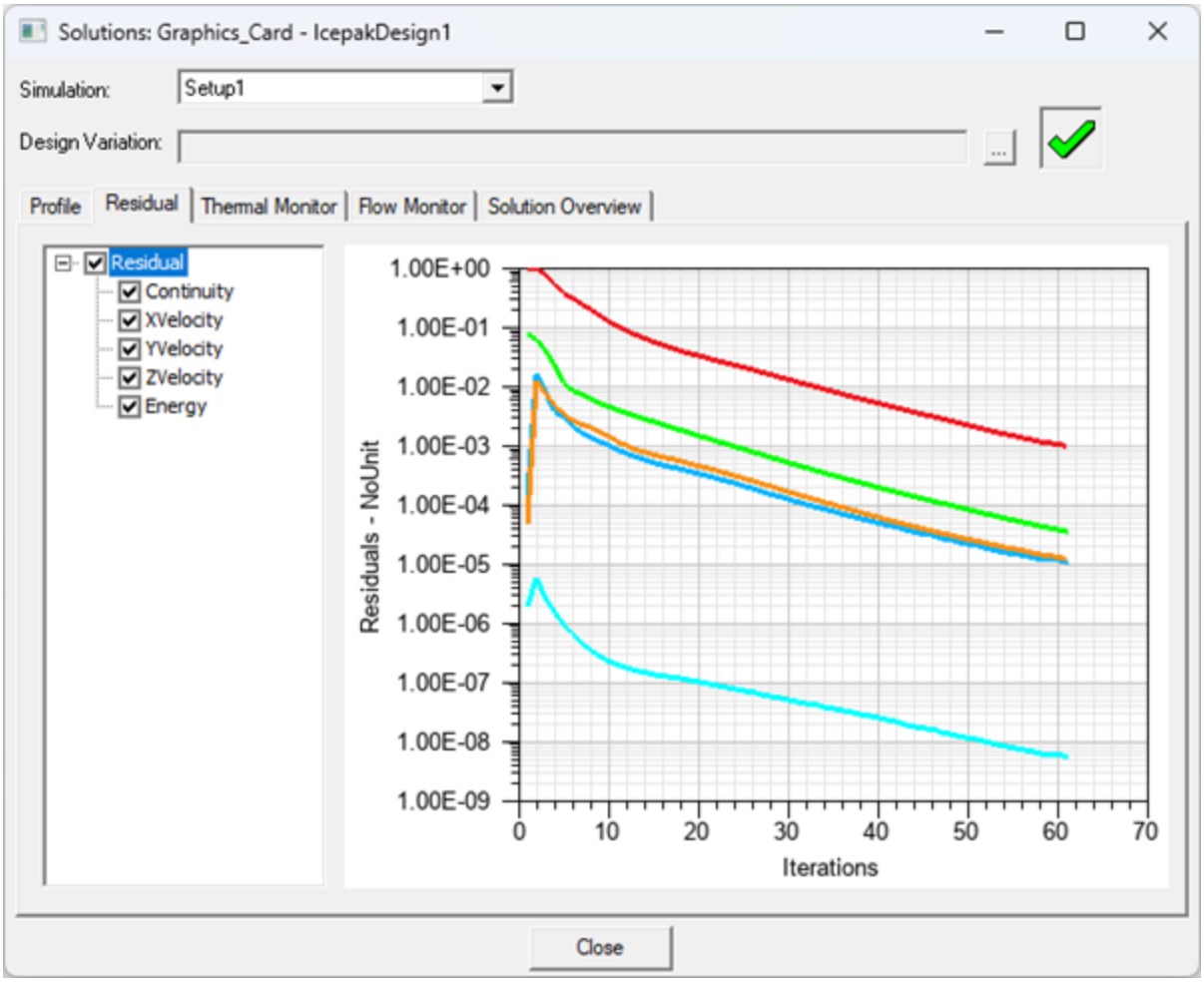
   The data is saved in a text file with a .prof extension.

Viewing Solution Residuals

At any time during or after the solution process, you can examine the solution residuals and any surface or point monitors you created.

In the project tree, right-click the solution setup of interest, and then click **Residual** on the shortcut menu.

The **Solutions** dialog box appears. The **Residual** tab is selected.



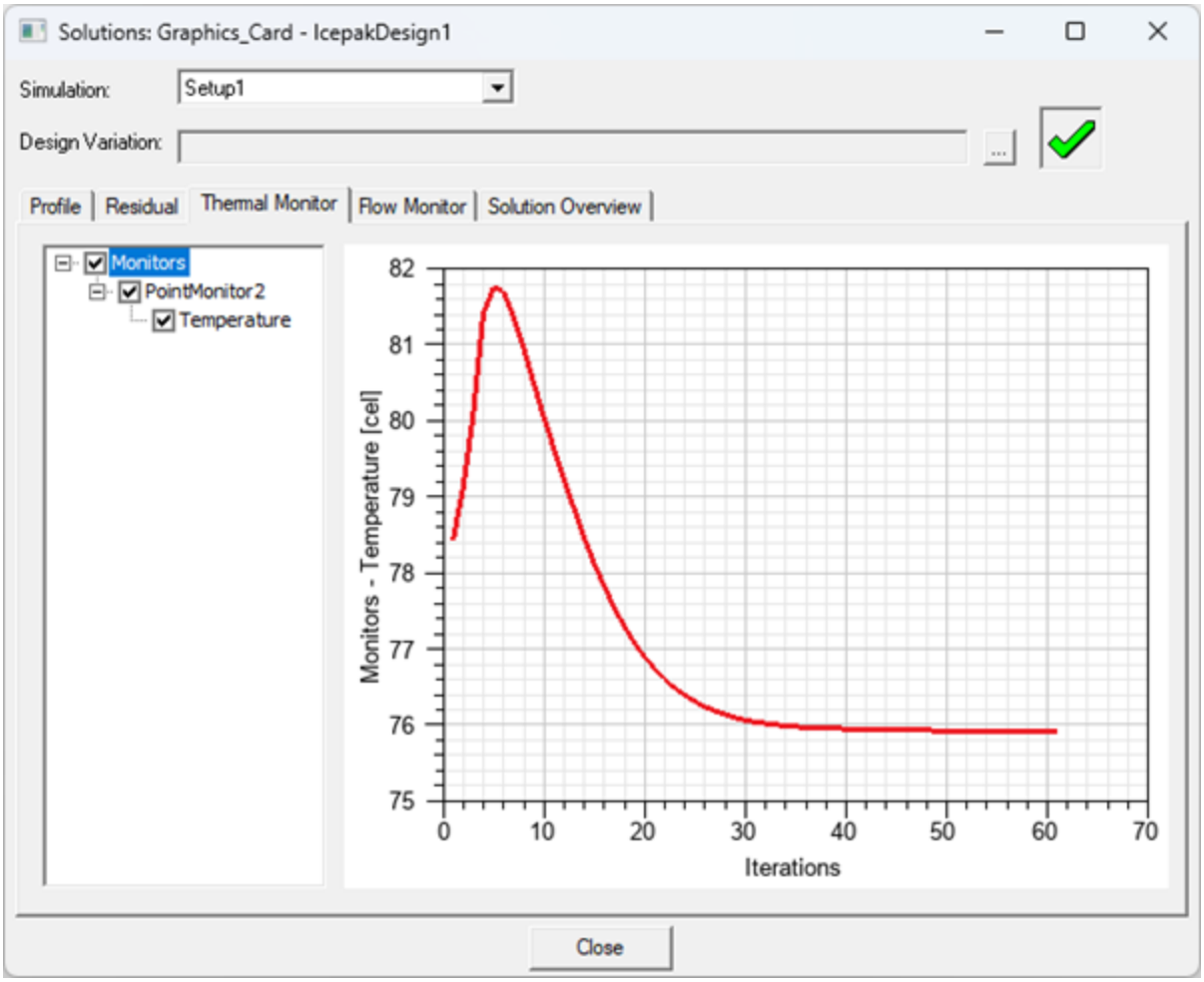
Viewing Thermal and Flow Monitors

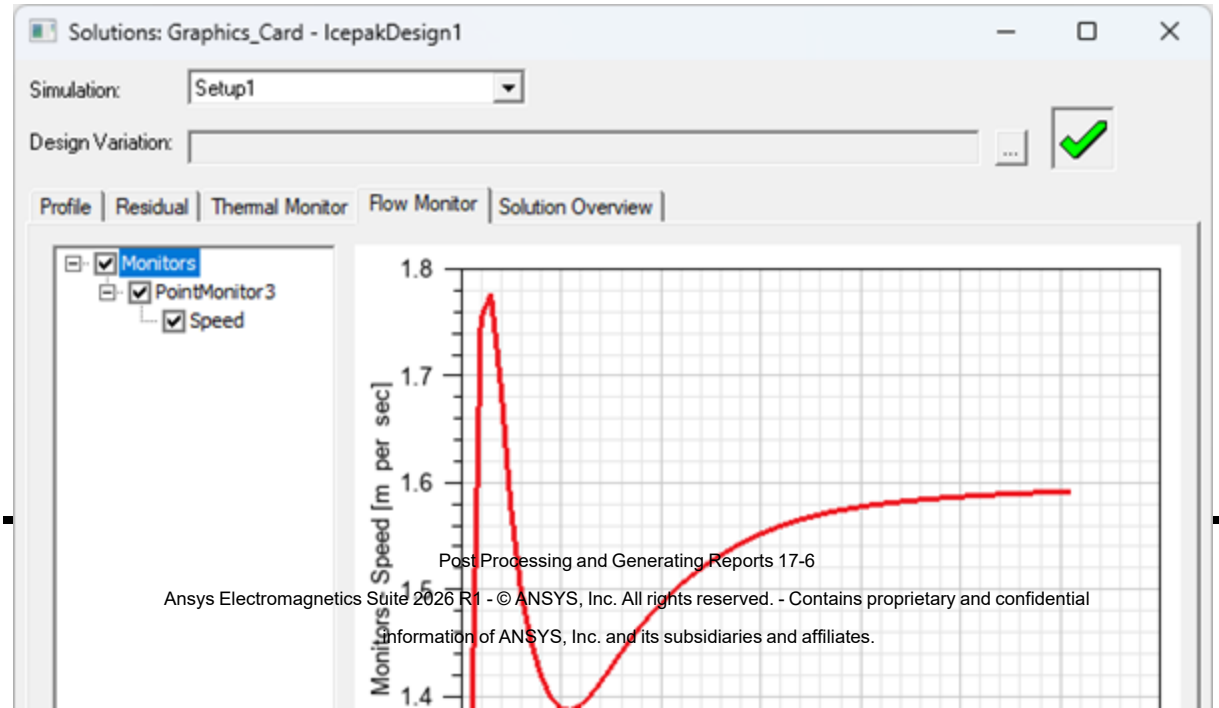
At any time during or after the solution process, you can examine the thermal and flow monitors you created before analyzing the solution setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the project tree, right-click the solution setup of interest, and then click **Thermal Monitor** or **Flow Monitor** on the shortcut menu.

The **Solutions** dialog box appears. The **Thermal Monitor** or **Flow Monitor** tab is selected.





Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

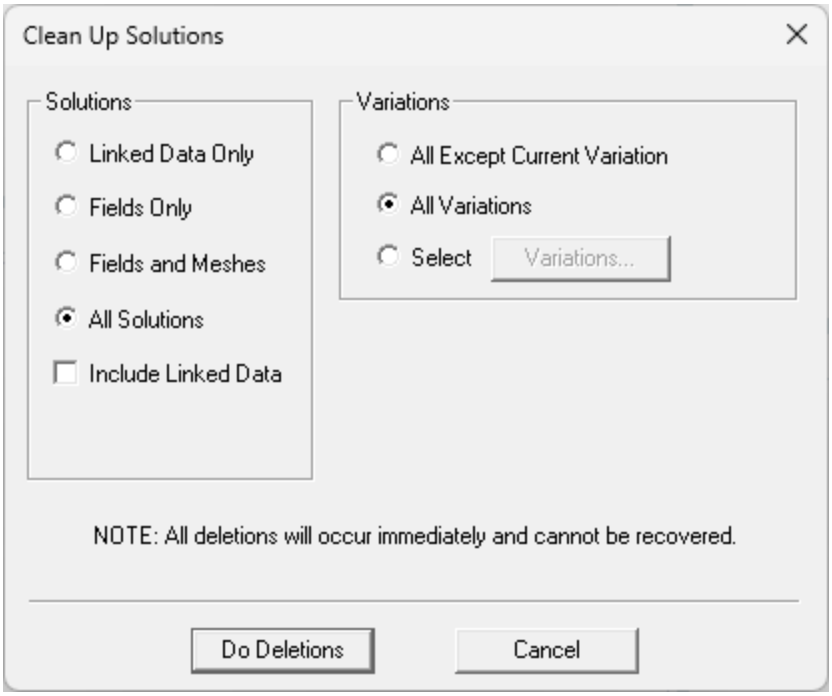
Deleting Solution Data

You can use **Clean Up Solutions** to selectively make deletions, or remove all solutions from the results.

To use **Clean Up Solutions**:

1. Click **Icepak** > **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 **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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Deleting Reports

To use **Delete All Reports**:

1. Click **Icepak** > **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:

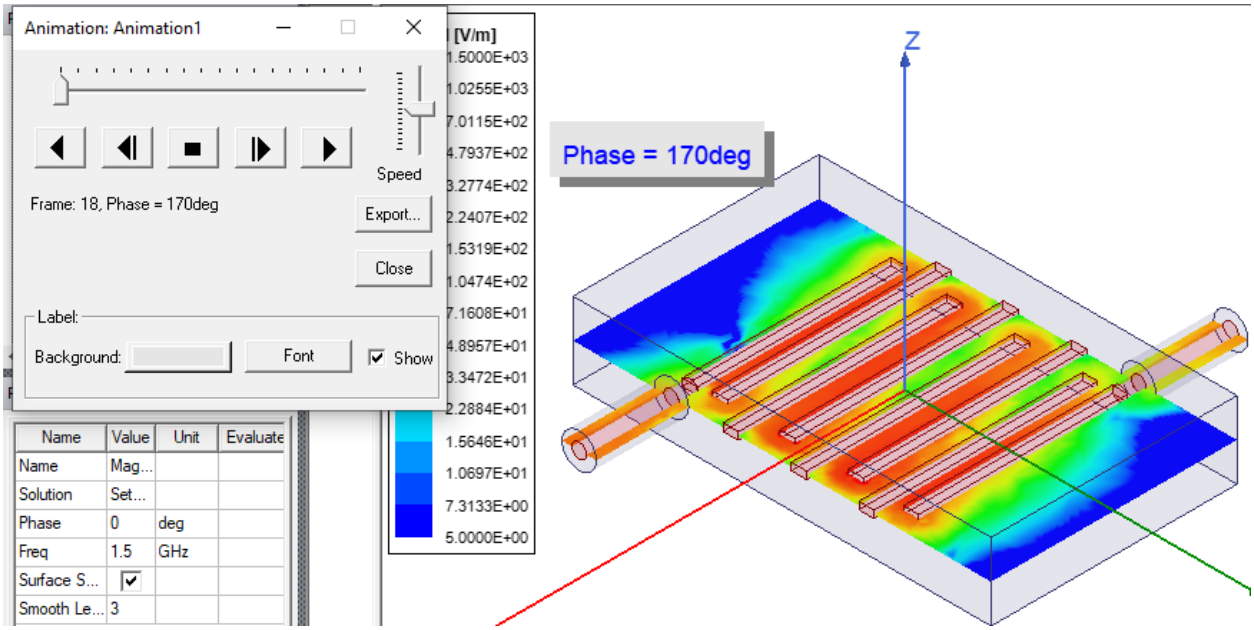
1. Right-click a report icon in the Project Manager to display the shortcut menu.
2. Click **Delete** to remove this report.

> **Warning:**
>
> Solution data that has been deleted cannot be recovered!

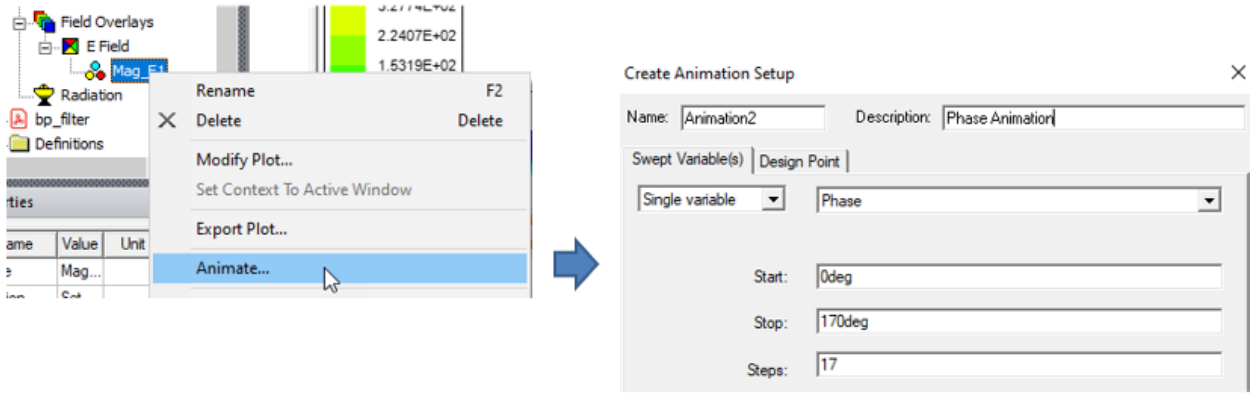
Creating Animations

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 2026 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 2026 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 Design of Experiments Animations

Animations can be based on Design of Experiments setups. Following is the general procedure for creating an animation of a geometry model using a Design of Experiments setup.

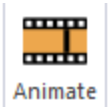
Prerequisites

- For animation of geometry solving Design of Experiments setup is not required.

Procedure:

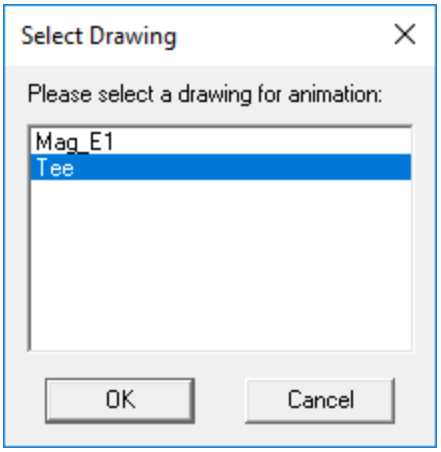
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the **Modeler** window, and click **View**> **Animate...**, right click on the Design of Experiments setup in the Project tree and click **Animate...** in the short-cut menu, or select the **View** tab of the ribbon, and click the **Animate** icon.



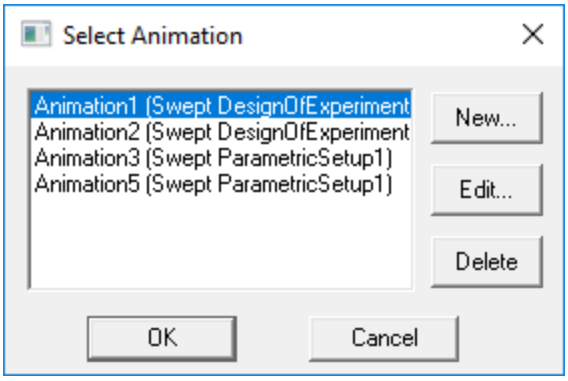
   If multiple geometries can be varied in the design, the **Select Drawing** dialog box appears, proceed to step 2. If only one geometry is assigned variable(s), proceed to step 3.

2. In the **Select Drawing** dialog box select the drawing for animation. The drawing should be appropriate for geometry animation:



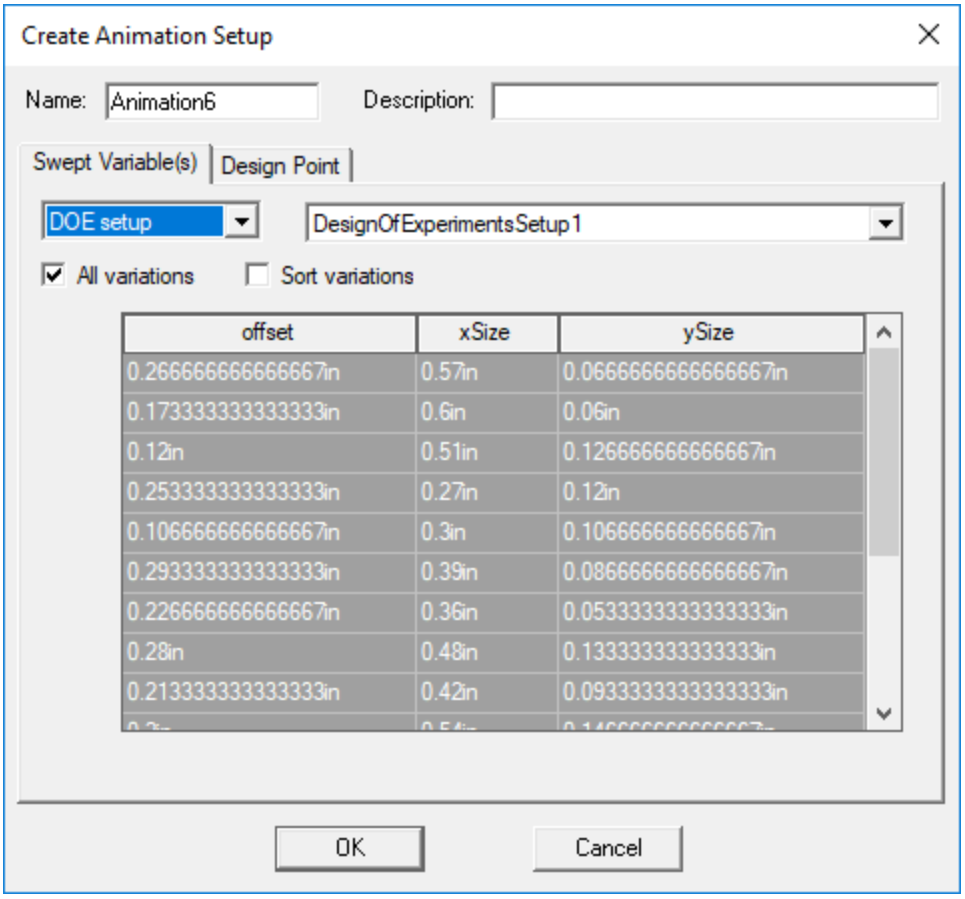
> **Note:**
>
> If previous animations have been created for this project, the **Select Animation** dialog will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest and the animation will start.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

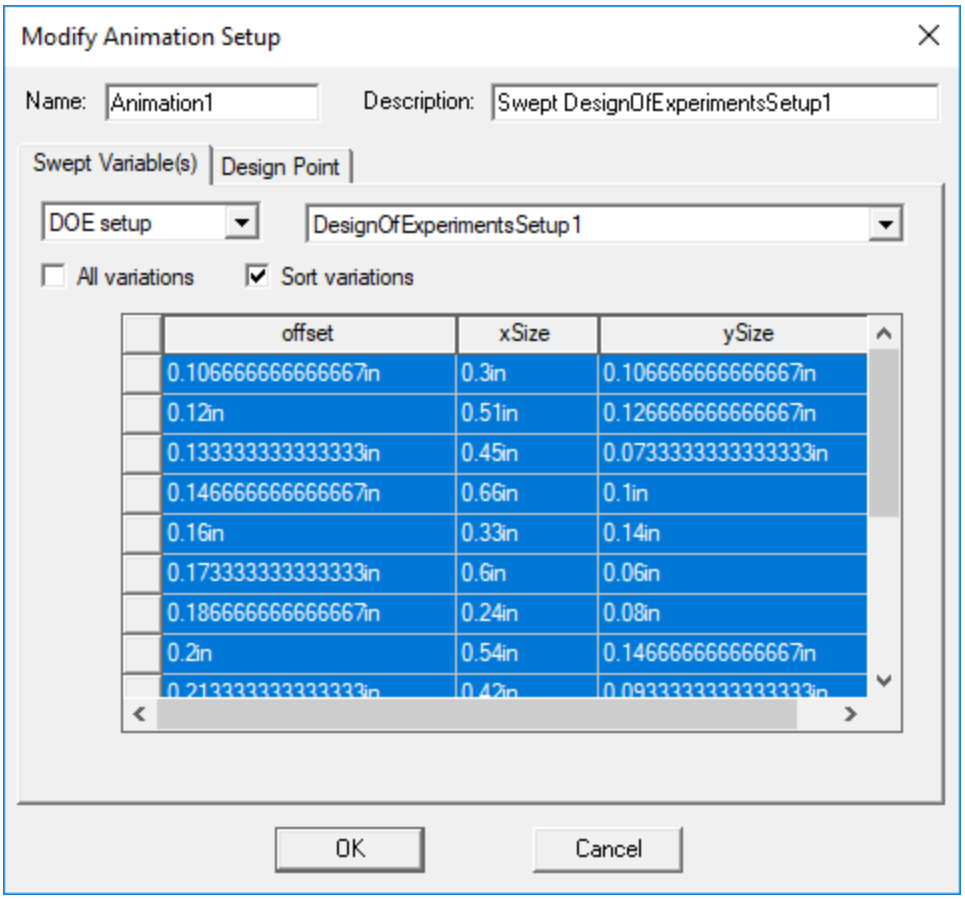
The **Create Animation Setup** dialog box appears.



3. If your have created one or more Design of Experiments setups for one or more geometric variables, you can select, **DOE setup** to filter existing DesignOfExperimentsSetups, and the setup to use, and the variable values to animate. With All variations checked, the

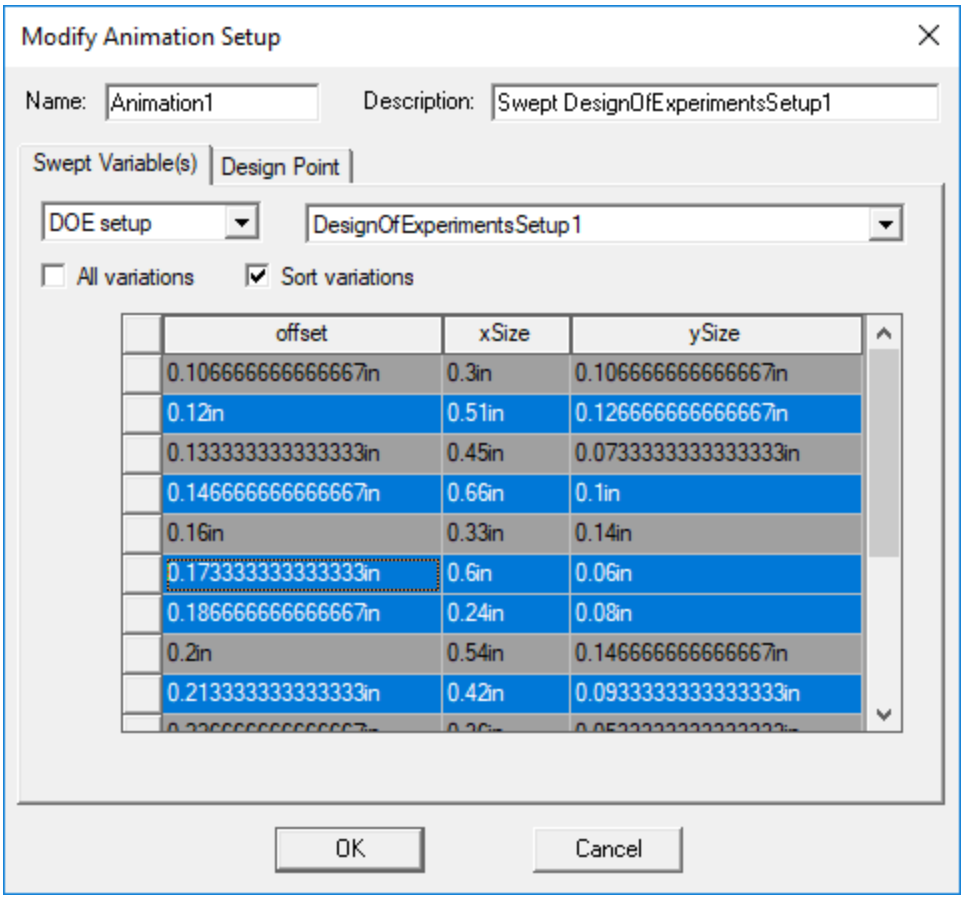
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable table is not selectable. If you uncheck All variations, you can use select from the available setup values. All variations are selected [highlighted] when you uncheck All variations.



If you check Sort variations, the values are sorted. You can click to select the variations you want.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

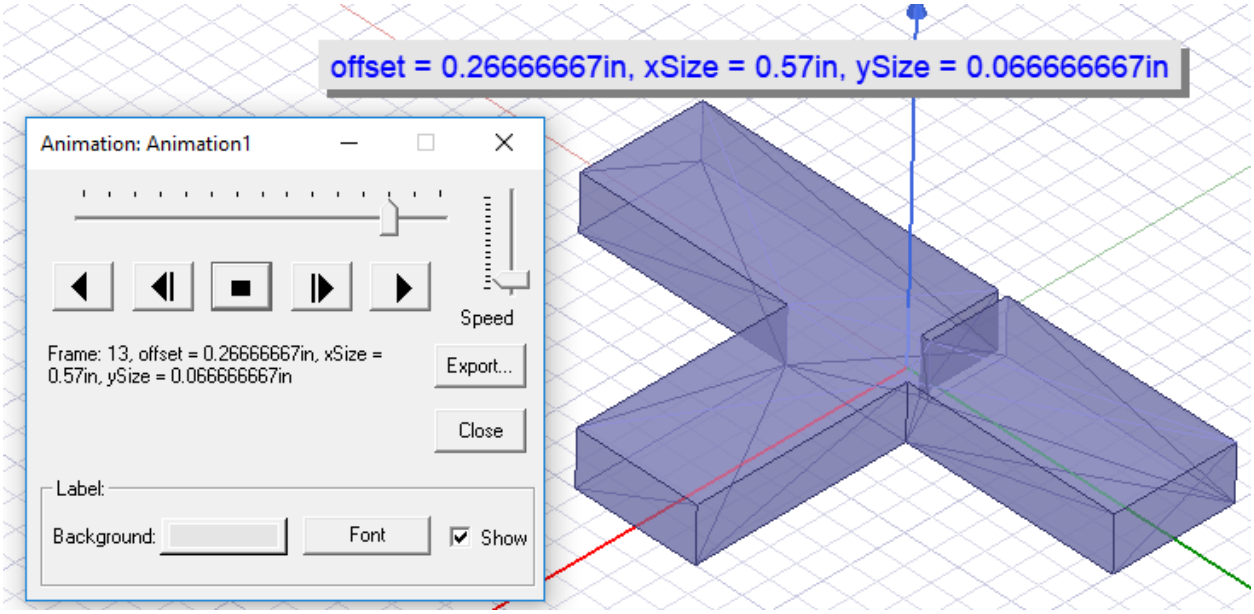


4. Click **OK**.

   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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

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.

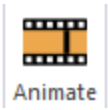
Procedure:

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

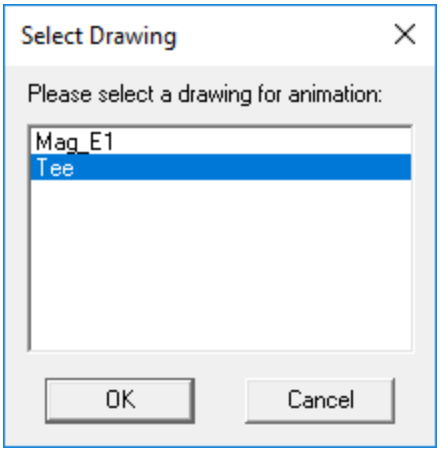
   Select the **View** tab of the ribbon, and click the **Animate** icon.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

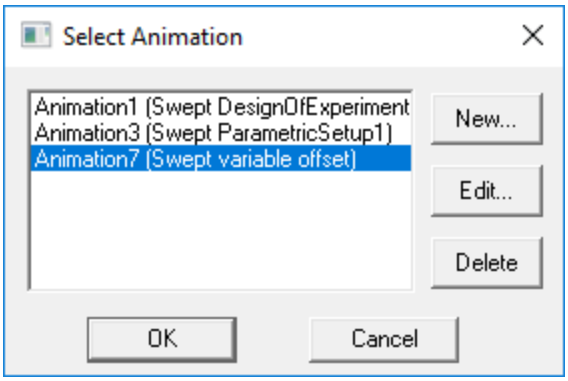


If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

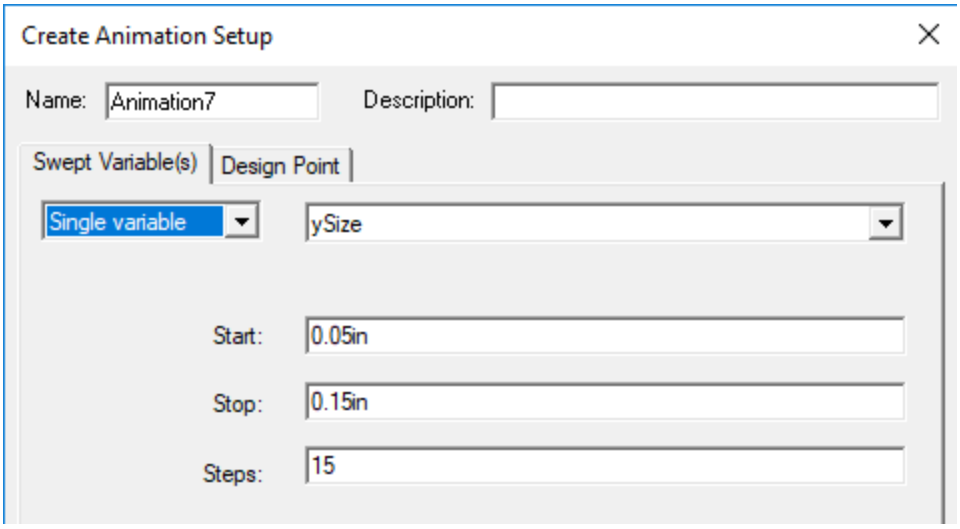


If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. You may choose an animation setup from the list 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.

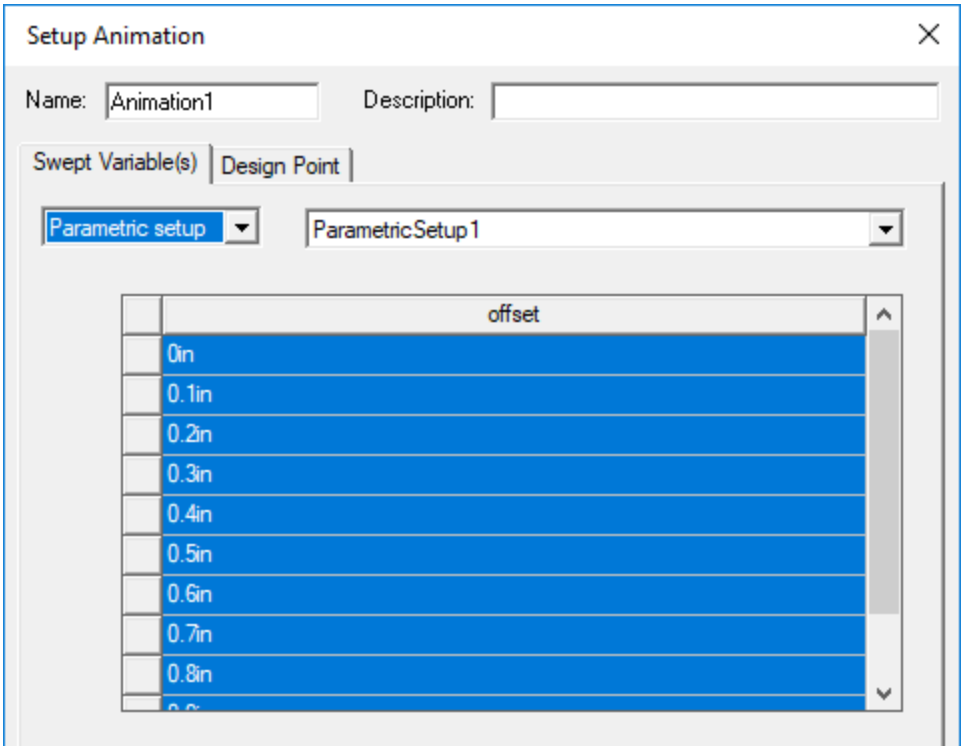


The *Create Animation Setup* dialog box appears:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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

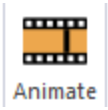
Parametric animations can be based on parametric sweeps. Following is the general procedure for creating an animation of a plot based a parametric sweep.

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.
- 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.

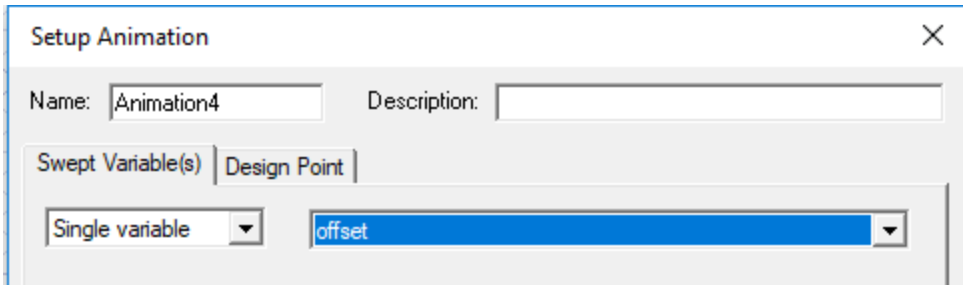
2. In the *Select Drawing* dialog box:
   a. Select the geometry variable to vary in the animation.
   b. Select the object you want to animate.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

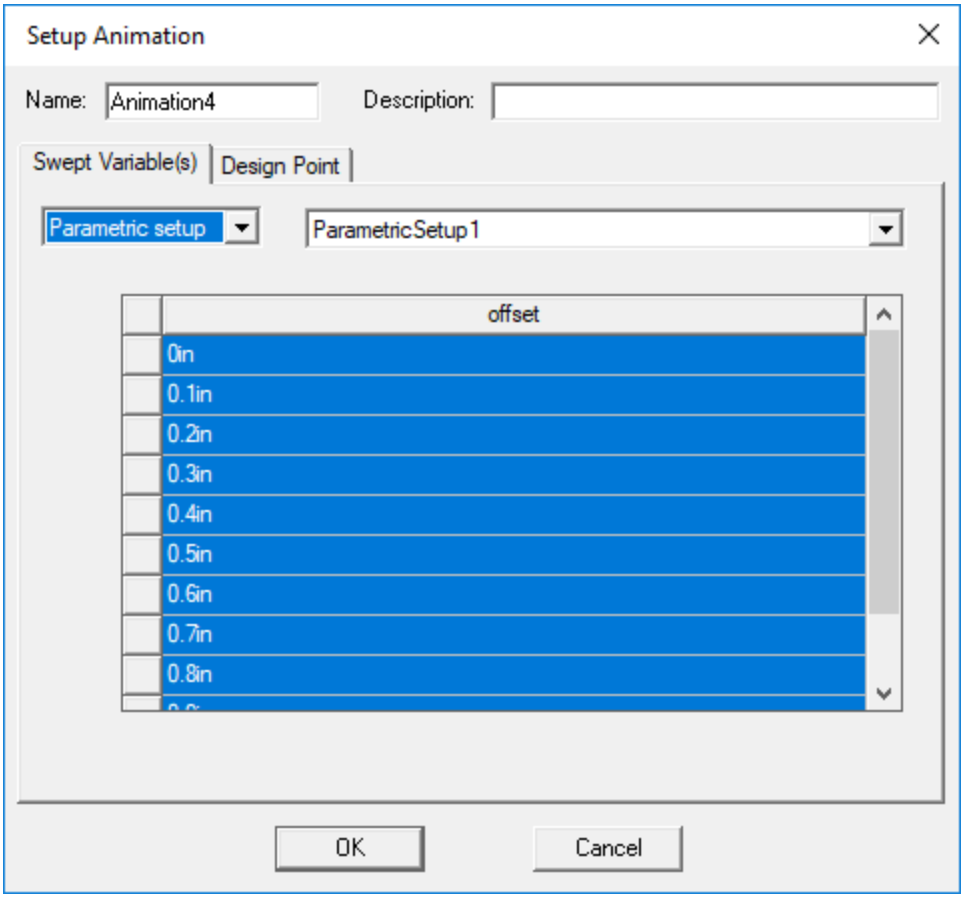
If previous animations have been created for this project, the *Select Animation* dialog box will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

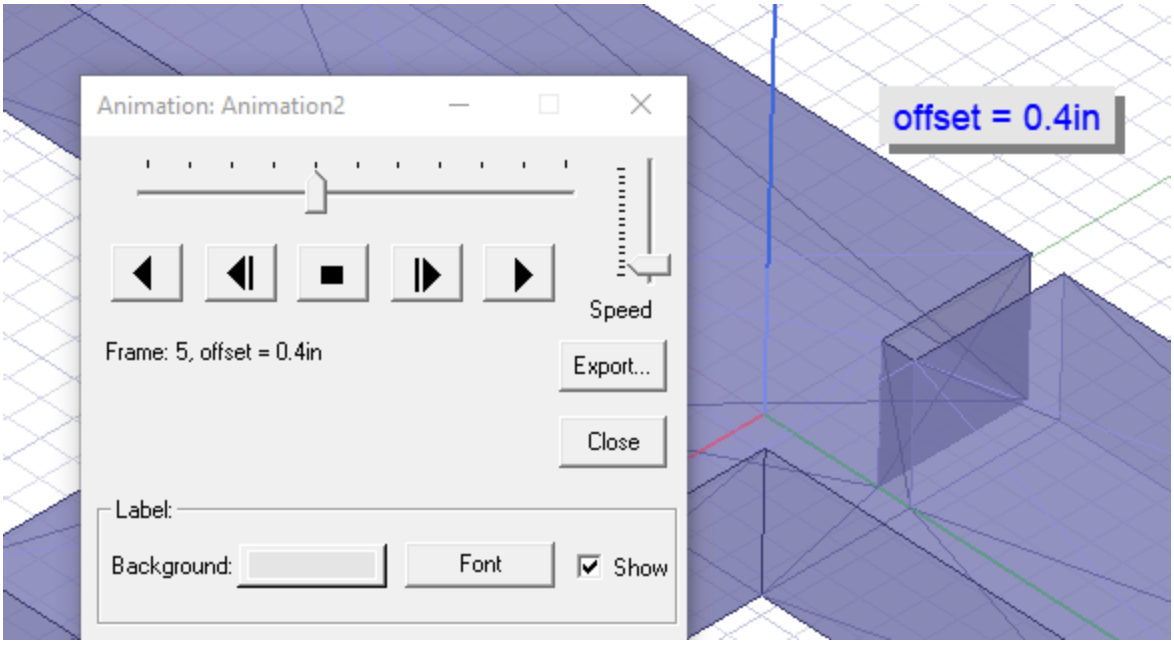


4. Click **OK**.

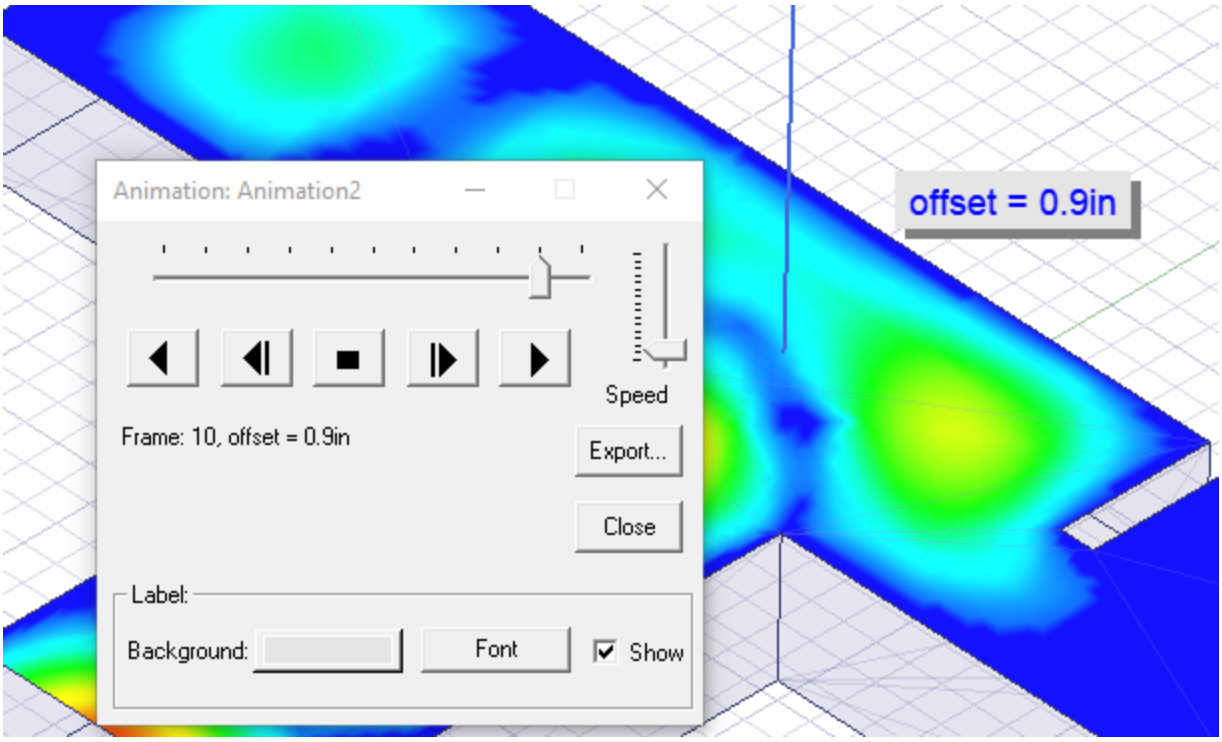
   The animation begins in the view window. It will display one frame for each variable value.

   The **Animation** control panel lets you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



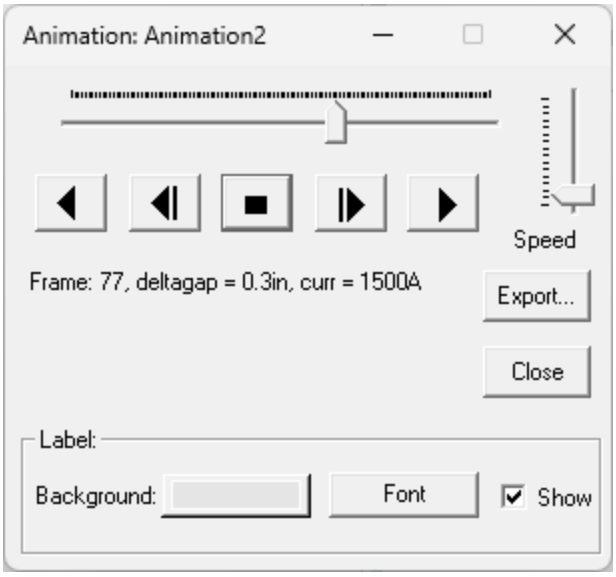
While a parametric animation is running, you can also turn on other field plots.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controlling the Animation's Display

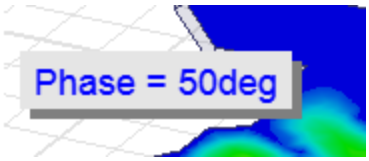
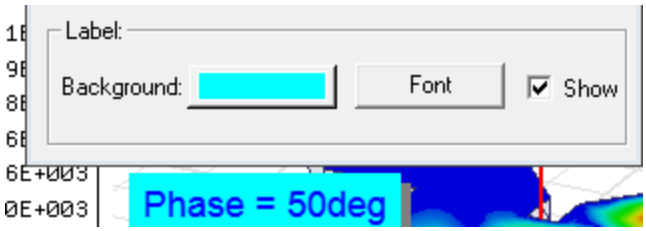
When an animation is displayed in the view window, the **Animation** window (also called the *play panel*) appears in the upper-left corner of the desktop. It has buttons that enable you to control the speed and sequence of the frames, start and stop the animation, and export the animation.



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.
	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	The current frame and phase at which the plot is being displayed is listed below the control buttons.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

information

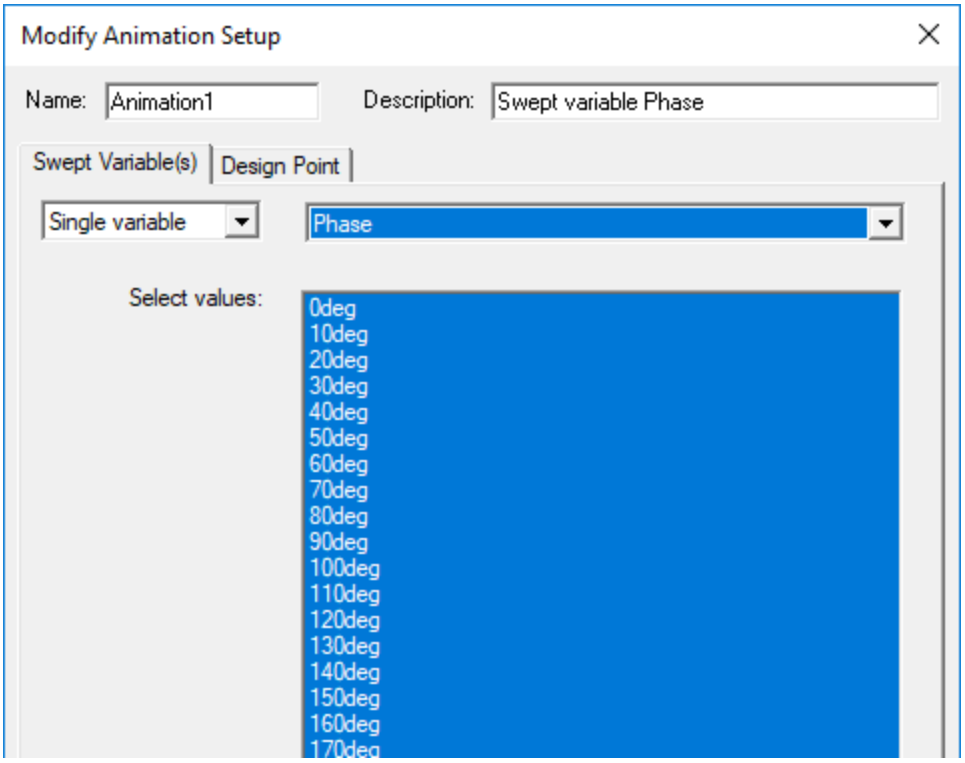
Export...	Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.
Close	Closes the animation window.
Show label check box	If you select the Show check box, a label showing the swept variable value appears in the animation. You can 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.
	
Font	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.

Modifying an Animation Setup

The **Edit** button on the *Select Animation* dialog box opens the *Modify Animation Setup* dialog box.

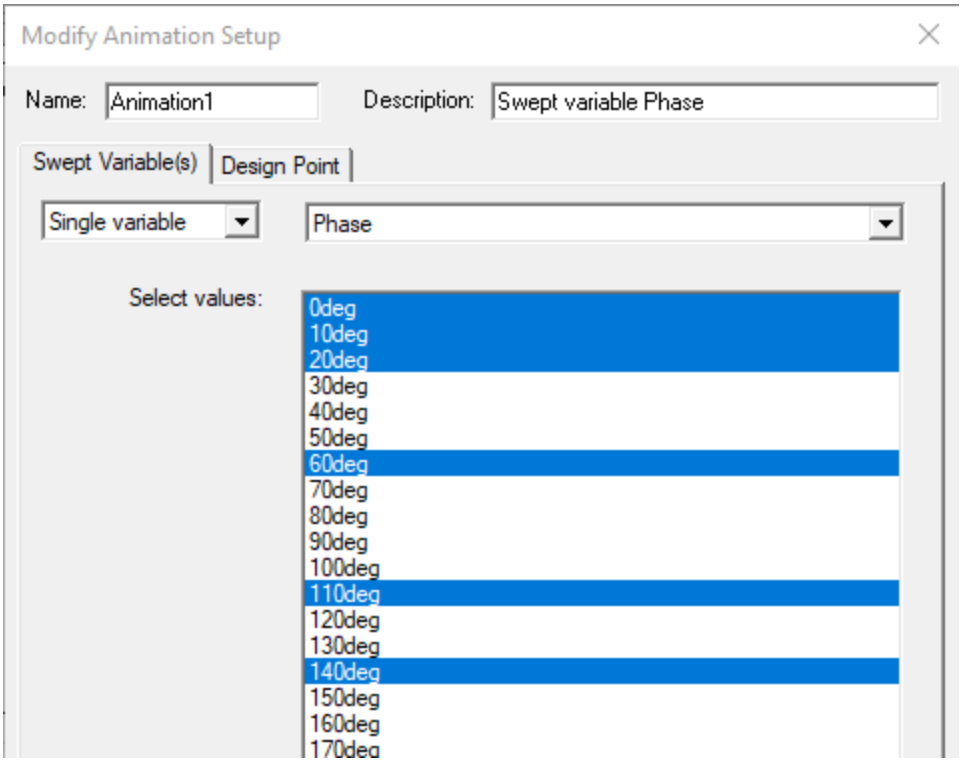
It opens showing the selections for variables and the selected values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

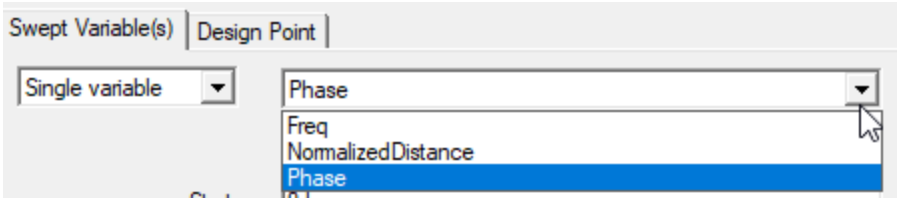


You can deselect or select any values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

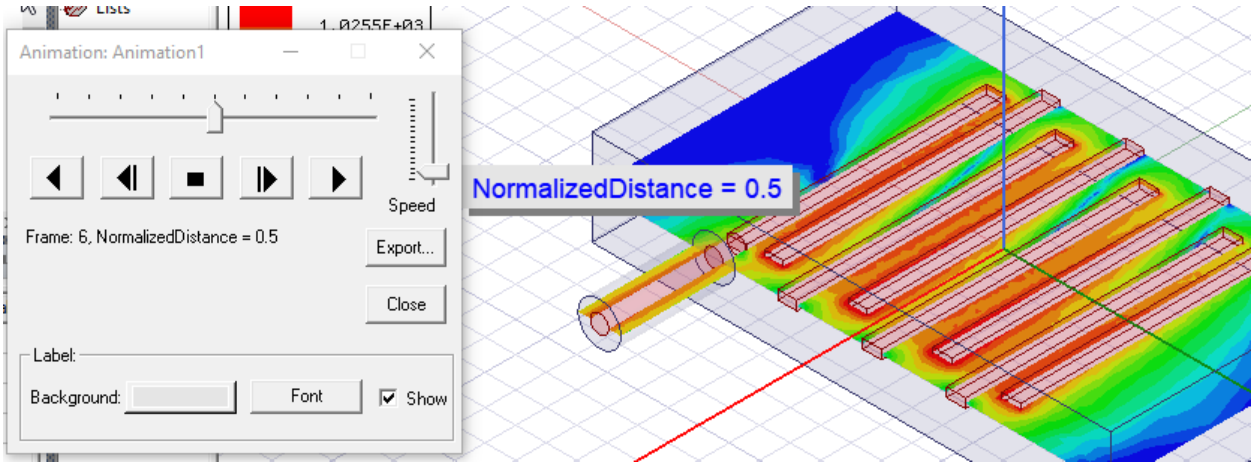


If other variables have been defined, you can select from the drop down list.

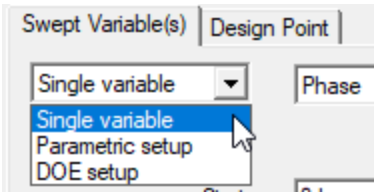


If the other variables have or can be set to appropriate values, you can modify the animation.

Ansys Electromagnetics Suite 2026 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

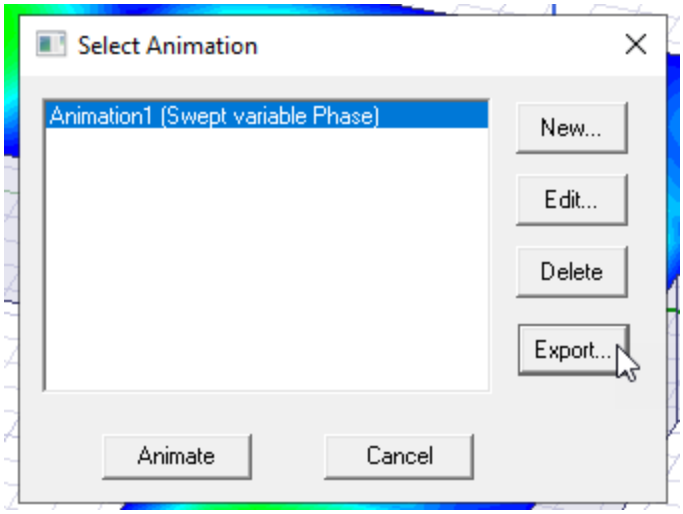
Exporting Animations

You can export animations from the **Animation** control dialog box, or, for Ensight format, from the **Select Animation** dialog box.

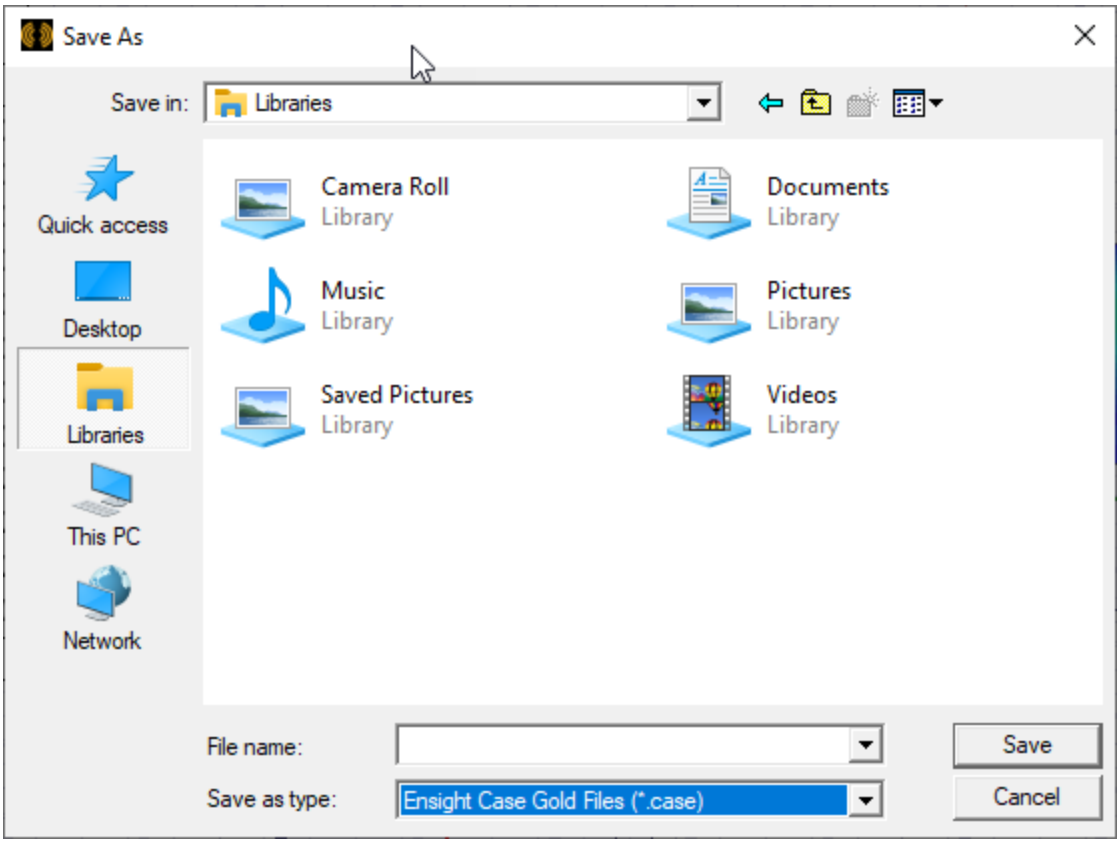
Exporting Ensight from the Select Animation Dialog Box

1. Create the animation you want to export.
2. In the **Select Animation** dialog box, select **Export...**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



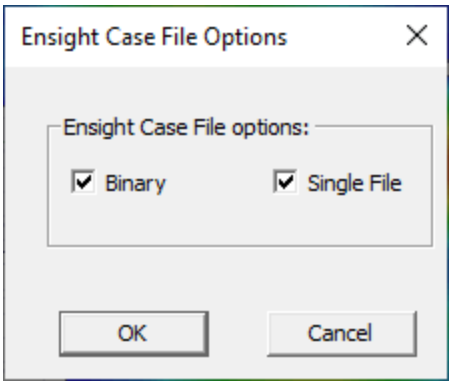
3. This opens the Browser window.



Navigate to the desired location and provide a file name. The format will be Ensight .case files.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

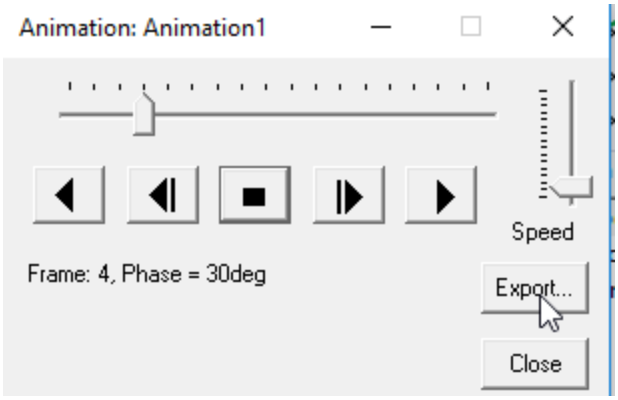
4. When you select **Save**, the **Ensight Case File Options** dialog box opens.



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"

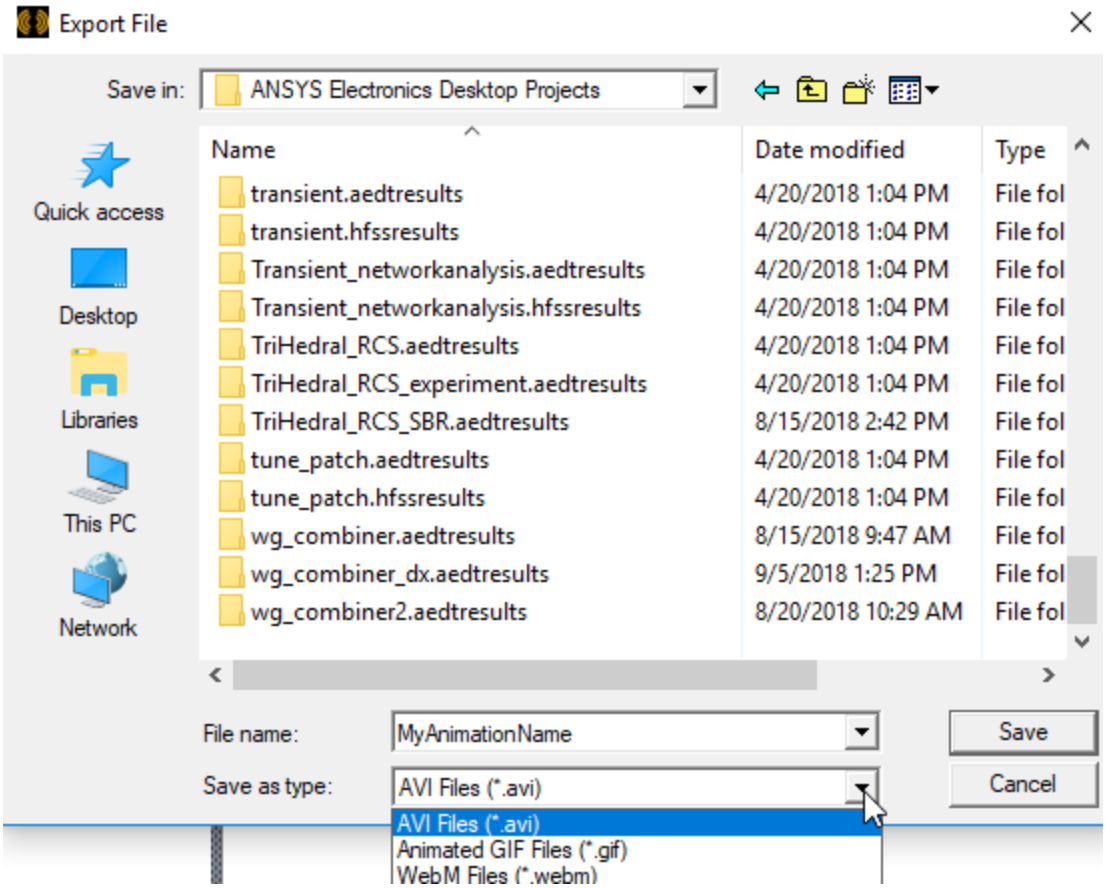
Exporting from the Animation Control Dialog Box

1. Create the animation you want to export.
2. In the play panel, click **Export...**.



The *Export File* dialog box appears.

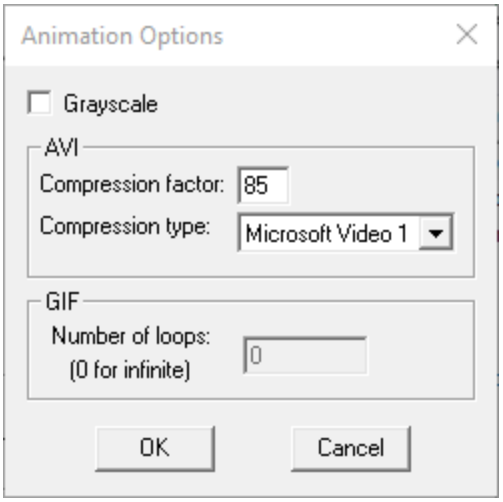
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

Creating Reports

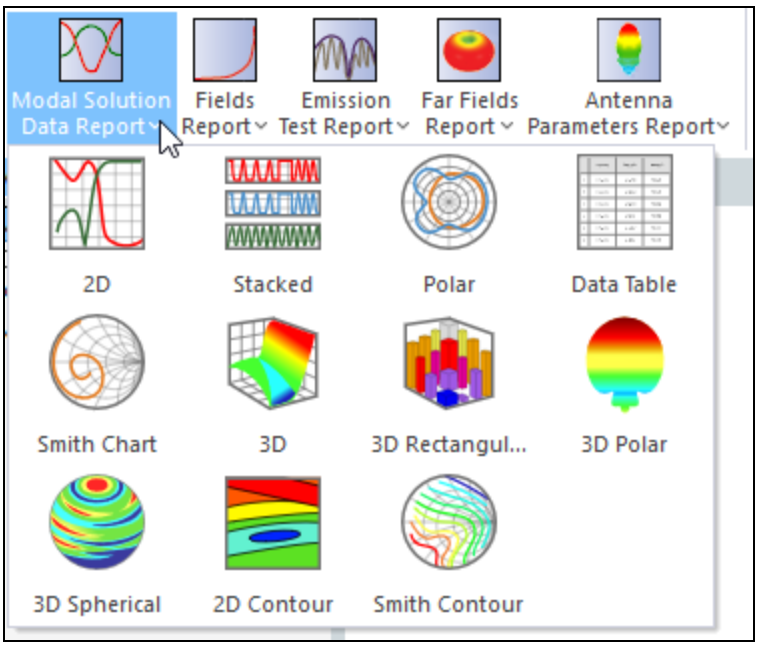
After a solver generates a solution, you can analyze all the results for that solution. Ansys Electronics Desktop lets you create 2D or 3D plots. A 2D or 3D plot shows the relationship between a design's values and the corresponding results of the analysis. You can create reports using either the **Create Quick Report** option, or **Create *<type>* Report** commands. The **Quick Report** feature lets you select from a list of predefined categories (such as S-parameters) from which to create a rectangular plot.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



> **Note:** You can use the Reporter Options to configure the elements of a 3D or 2D report.

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 **Icepak** > **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.

Creating a Quick Report

Following is the procedure for creating a quick report.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the Project tree under Analysis, select a **setup** or **sweep icon**, or the **Results** icon.
2. Right-click to display the shortcut menu and select **Create Quick Report**.

   The **Quick Report** dialog box appears.

   The list of available reports differs depending on the Solution type.

3. Select the one or more categories for the report from the list and click **OK**.

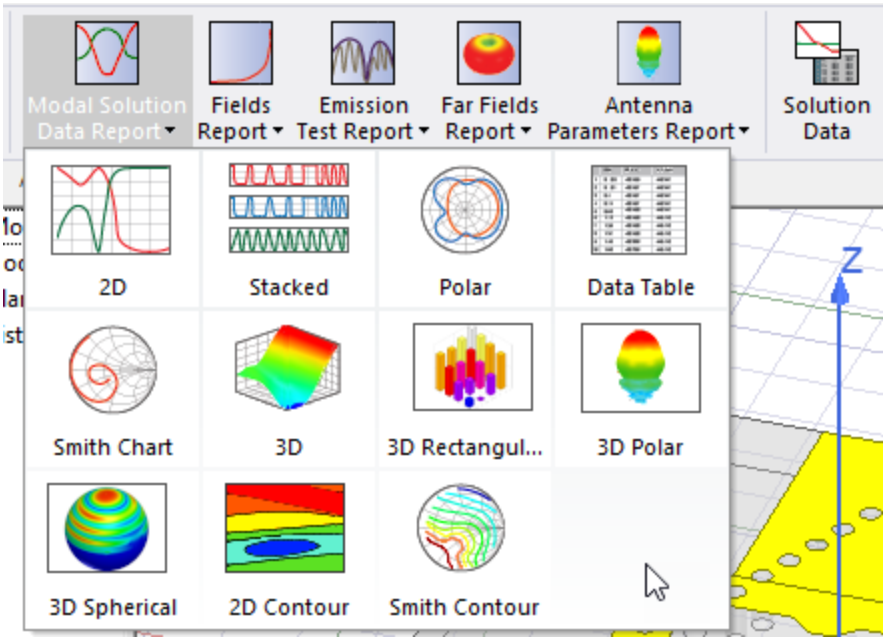
A rectangular plot for each selected category displays. The new plot or plots appear in the Project tree under the Results icon. The default Report Name that appears derives from the report type specified in the **Quick Report** dialog box.

Creating a New Report

Following is the general procedure for creating a new report:

1. On the **Icepak** menu or the Project Manager, select **Results**, then 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.



   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 **Icepak** > **Results** > **Report Templates** > **PersonalLib** > *<templateName>*. You can also

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   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.

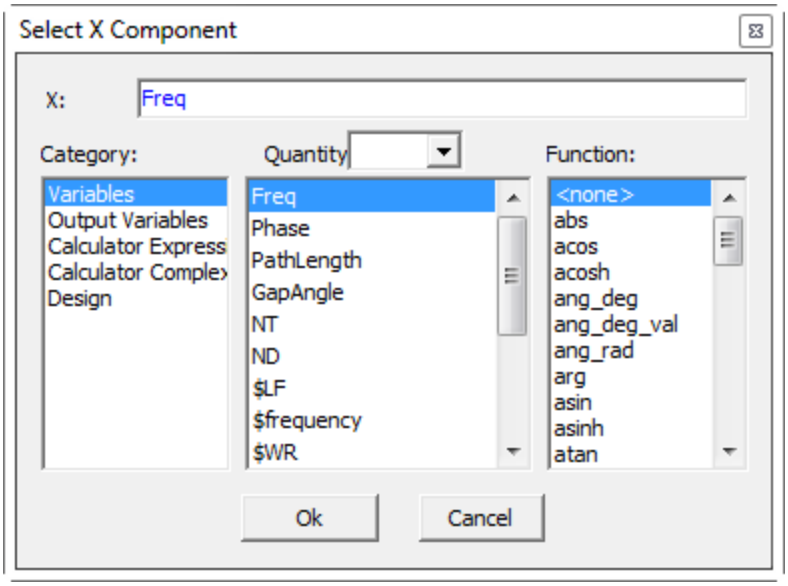
> **Note:**
>
> Color shows valid expression.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



      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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
   - **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 2026 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.

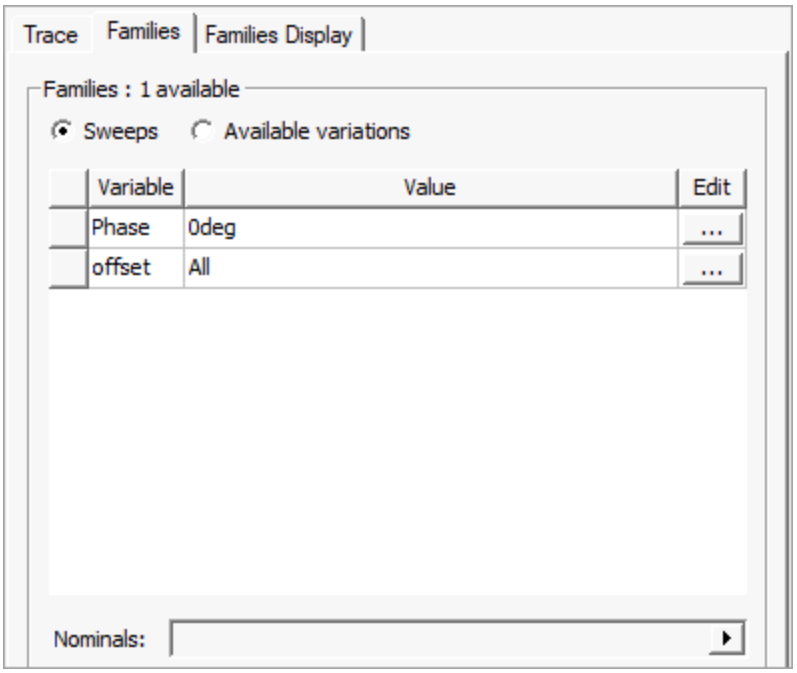
1. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
2. Geometry field with a drop-down selection list.
3. A Sources combo box appears in the Reporter when you have specified at least one source in the **Edit Sources** dialog box. For a usage example, see: *User Defined Solution for MIMO Calculations*.

Using Families Tab for Reports

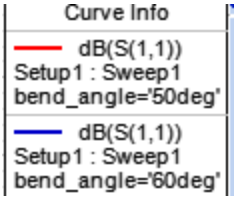
The **Families** tab of the **Report** dialog box provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined and solutions exist for multiple variable values (for example, for a parametric sweep or re-running an analysis with a different variable value). 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set any solved variables as Nominal, All, or select from values provided for Available solutions.



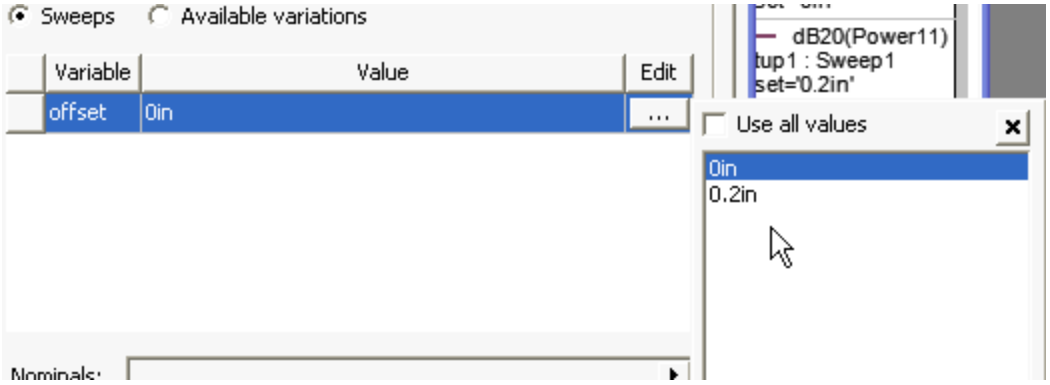
When you select a variable with multiple solved values, a trace for each solved value appears in the Report, with the variable value appended to the trace name in the Report legend.



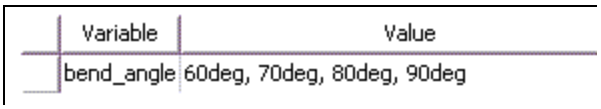
When families are available, you can make selections for the following:

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

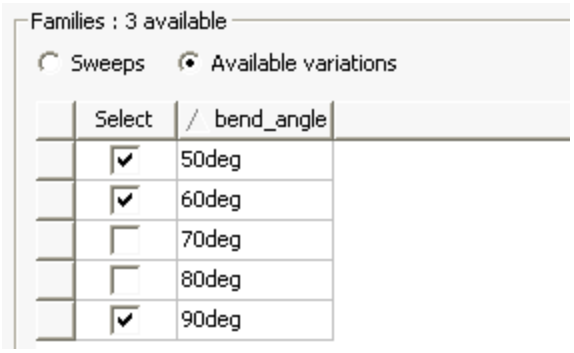


- To select all values, click the check box for **Use all** values. This writes "All" in the value field for that variable. You can also select individual values by clicking on them.
- To select a range of values, hold down the shift key, and click again.
- To select intermittent additional values, hold 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.

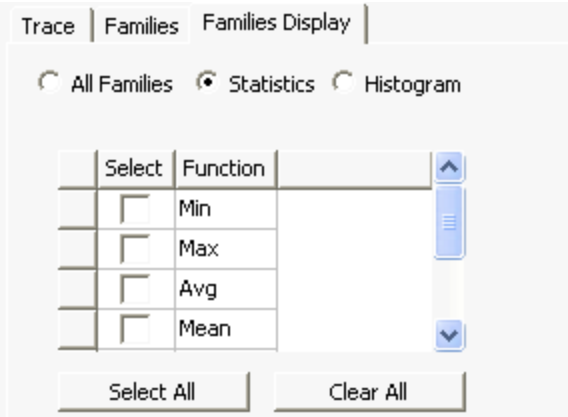


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



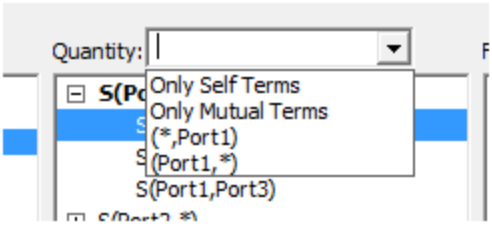
c. **Histogram** lets you select the number of bins to use for a histogram plot, and the sampling frequency to use.



Filtering Quantity Selections for the Reporter

When a two-port quantity Category is selected, four predefined filters are added to the combo box. "Port1" is the first matrix element name found in the quantity list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Only Self Terms – Only display quantities when the first and second port are same.
- Only Mutual Terms – Only display quantities when the first and second port are different.
- (*,Port1) – Only display quantities when the second element name is "Port1". You can edit the element name to display quantities for other elements.
- (Port1,*) – Only display quantities when the first element name is "Port1". You can edit the element name to display quantities for other element.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.
   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.

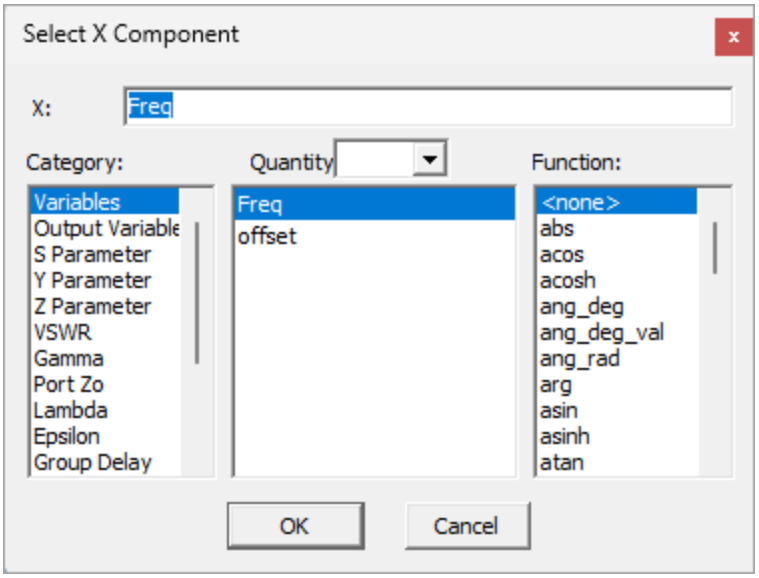
> **Note:**
>
> Color shows valid expression.

   e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



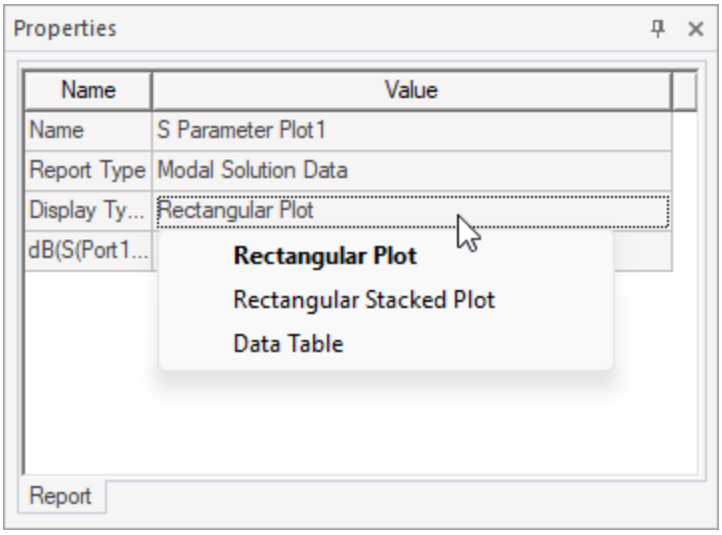
      This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

   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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



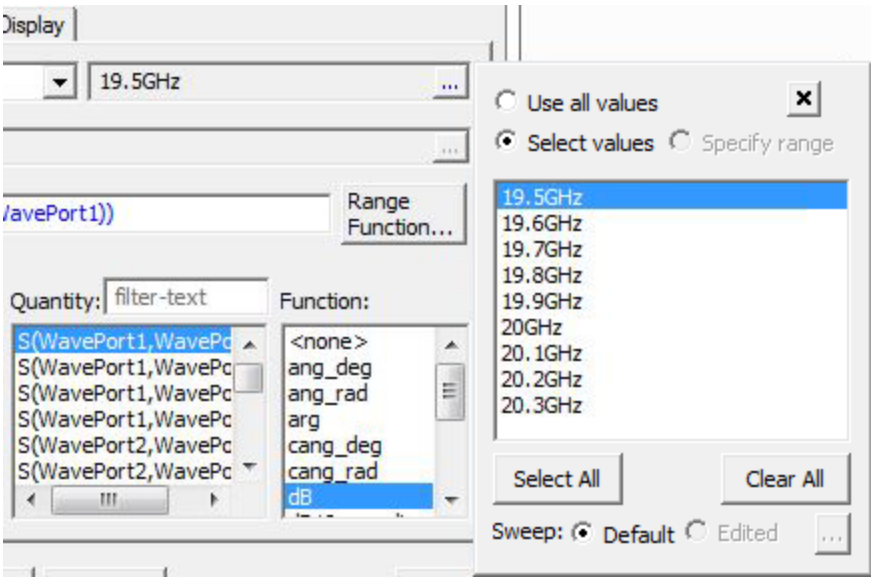
Once you make a selection, the plot display updates for the current selection.

> **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 2026 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.

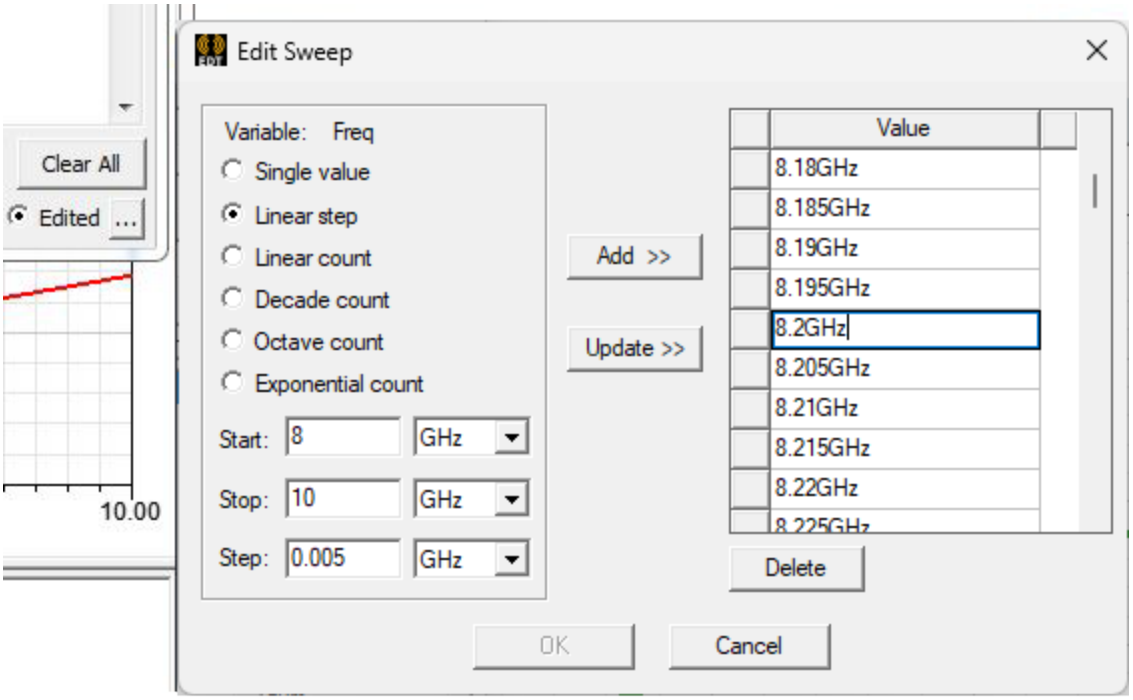
You select either the Sweep radio button for Default or Edited selection.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 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 **Icepak** > **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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. See Color Key Settings.
   - **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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.
  - **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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 Icepak treats as zero, in the data, for example, dB Gain.

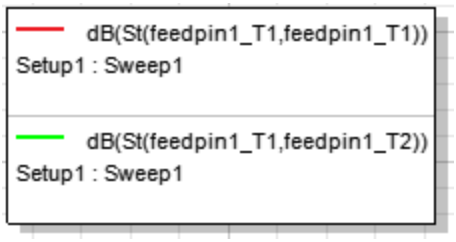
  - Map Infinity To – enter a real value for the Map Infinity Mode.

- **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.

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:

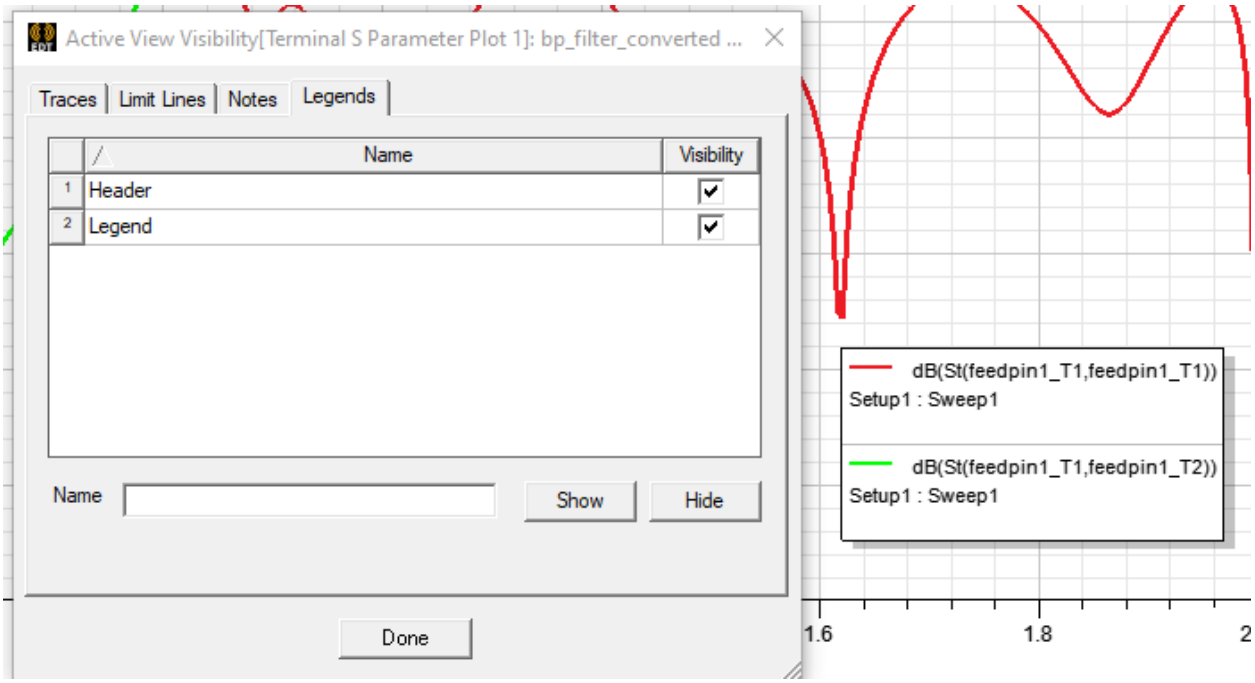
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



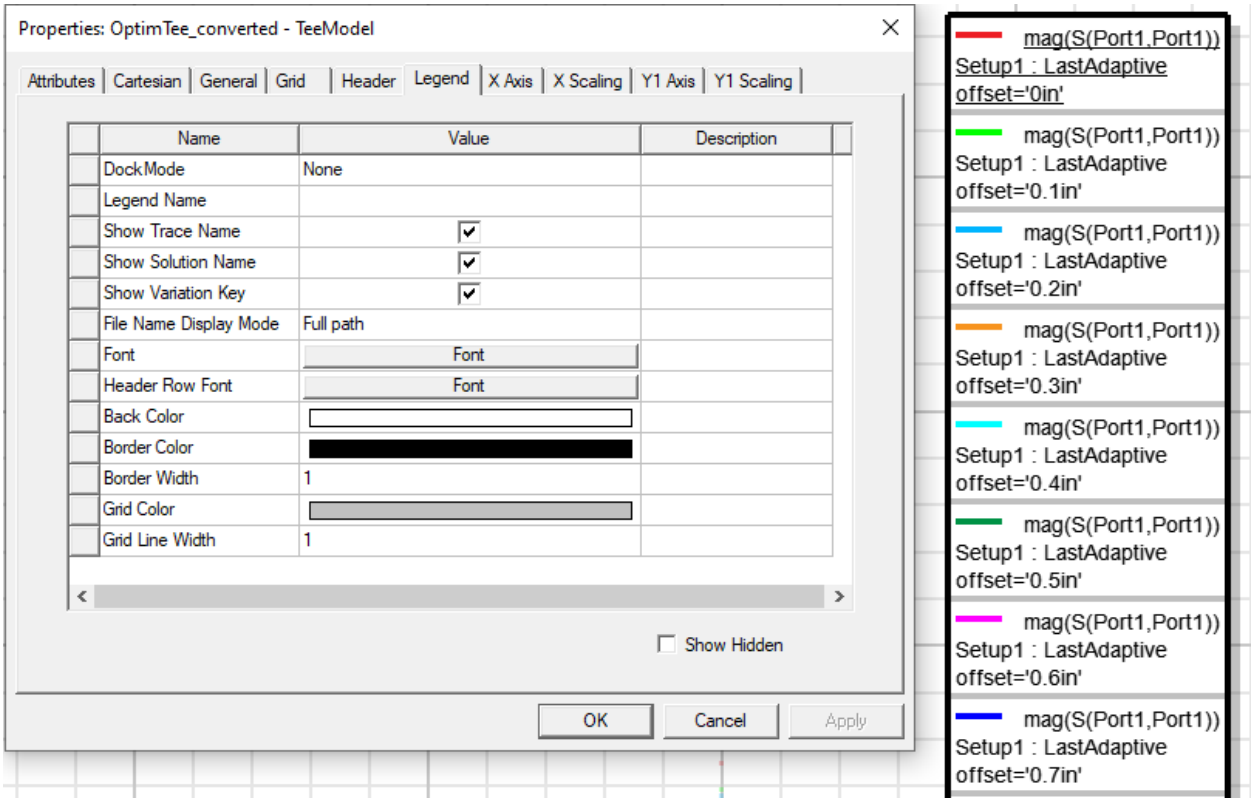
4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

To edit the display properties of a legend:

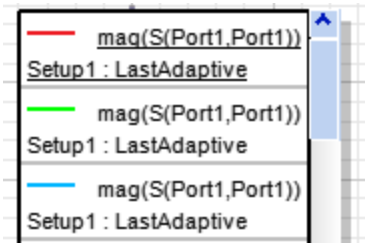
1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here is an example with Variation Key off.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

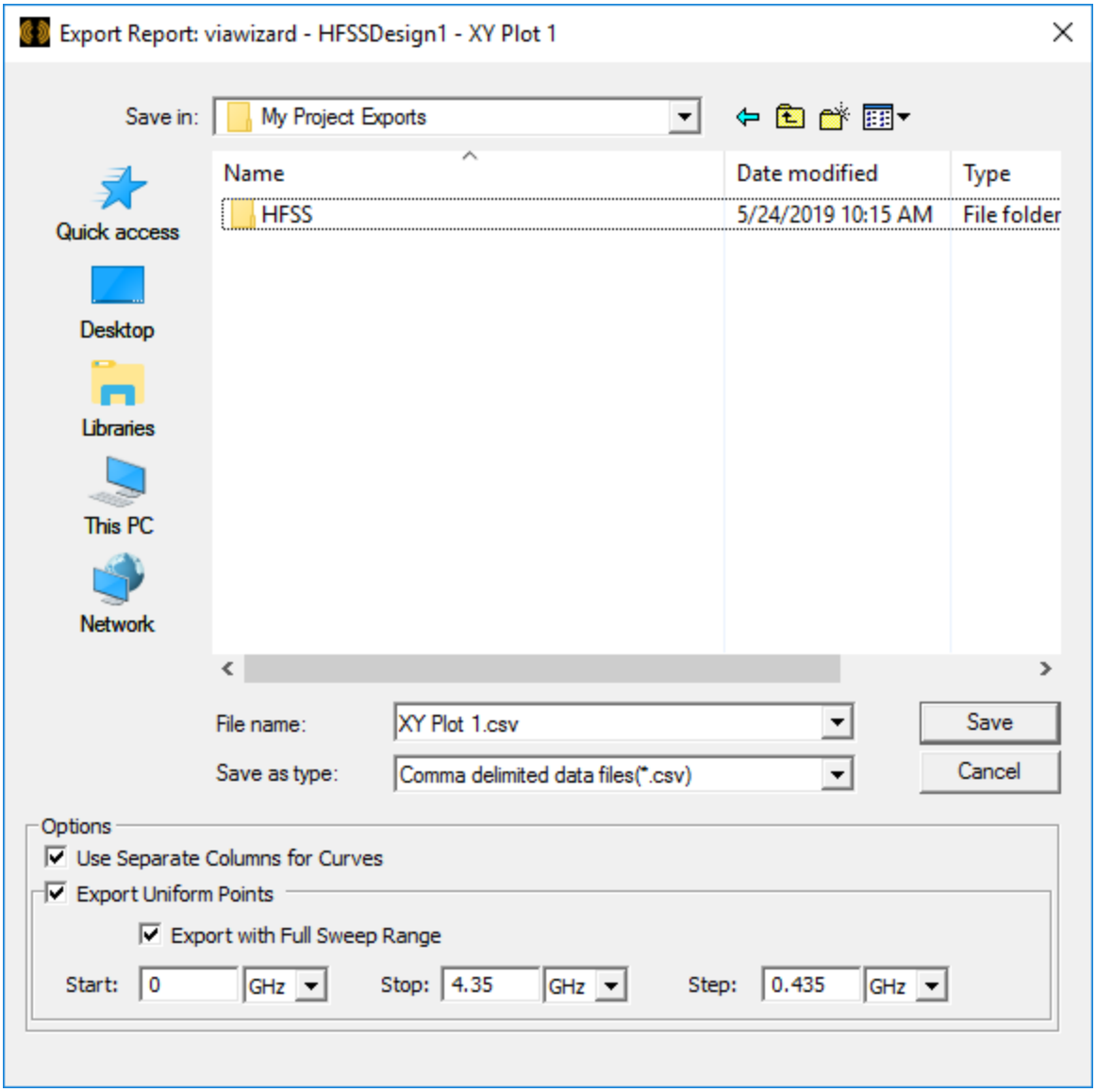
Exporting Ansoft Report Data Format Files

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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

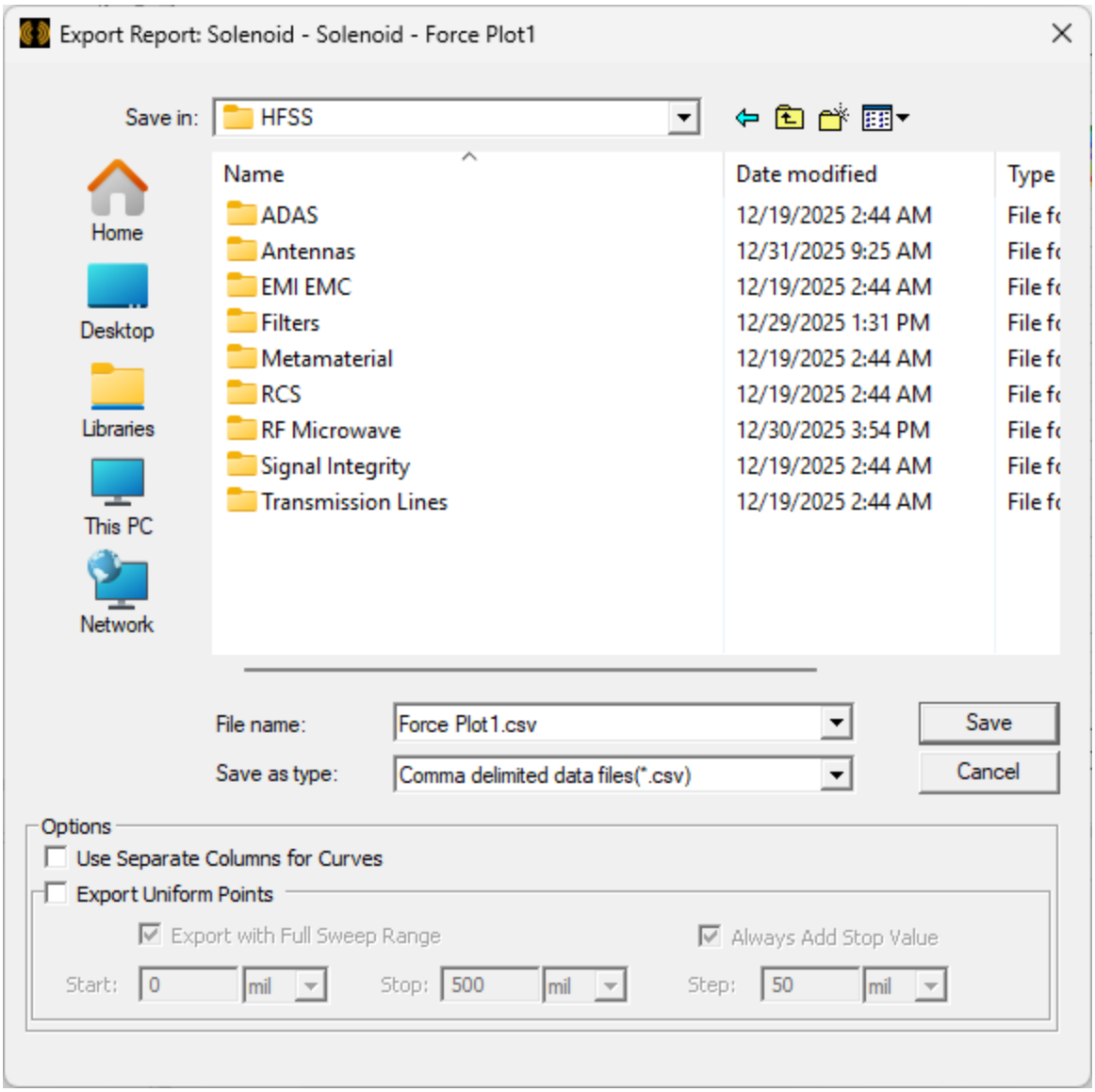
The file will then be available for import using Create Report from file.

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...**

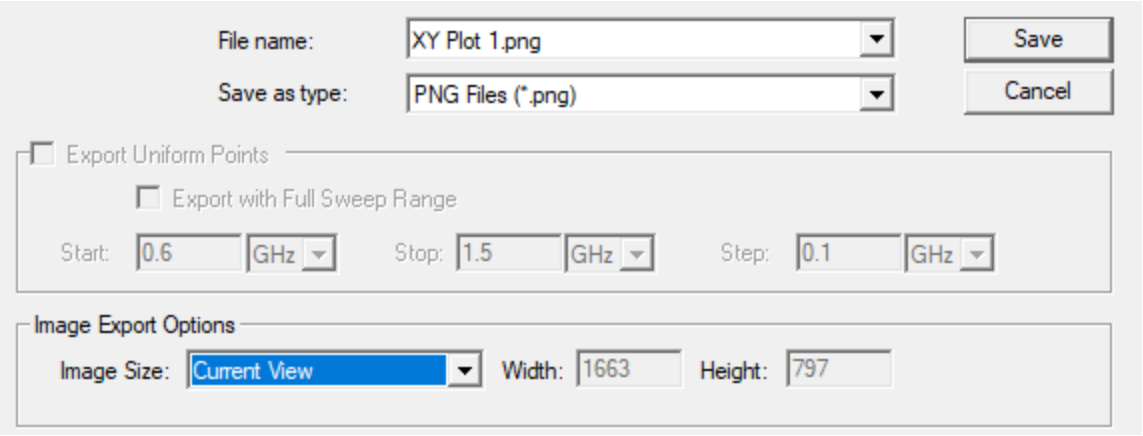
   The **Export Report** dialog box appears:



2. Click **Browse...** to open the **Export Report** browser window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the file location and name, and select a graphics format from the drop-down menu:
   - .bmp
   - .gif
   - .jpeg
   - .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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Report File Formats

The following table provides information about the file formats that can be imported into plot data reports:

File Extension	Information
csv	• Uses a comma (,) as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
tab	• Uses the tab character as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
txt	The txt file must begin with a header that contains the following: • A first line made of 92 equal signs (=) • A second line that begins with the company name and ends with the date in MM/DD/YY format • A third line that begins with a plot name and ends with the time in hh:mm:ss format • A fourth line, which is empty • A fifth line made of 92 hyphens (-) **Note:** Each header line must be 92 characters in length, except the empty line. • Each column has a fixed width, separated by white space. • The x-axis value is in the first column. Each curve's Y-values make up one

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Extension	Information
	column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the trace name [unit] for each curve, which is the same as the plot curve legend. • The second row is an empty column and the variable values for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
dat	DAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.
rdat	RDAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.

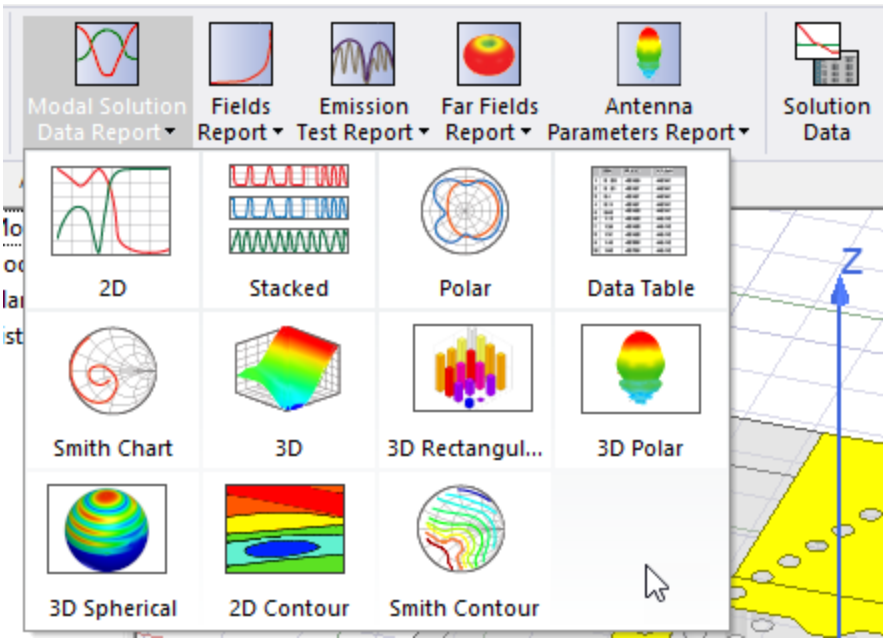
Selecting the Report Type

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.

Monitor	Data collected by surface and/or point monitors are available to plot.
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)*.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



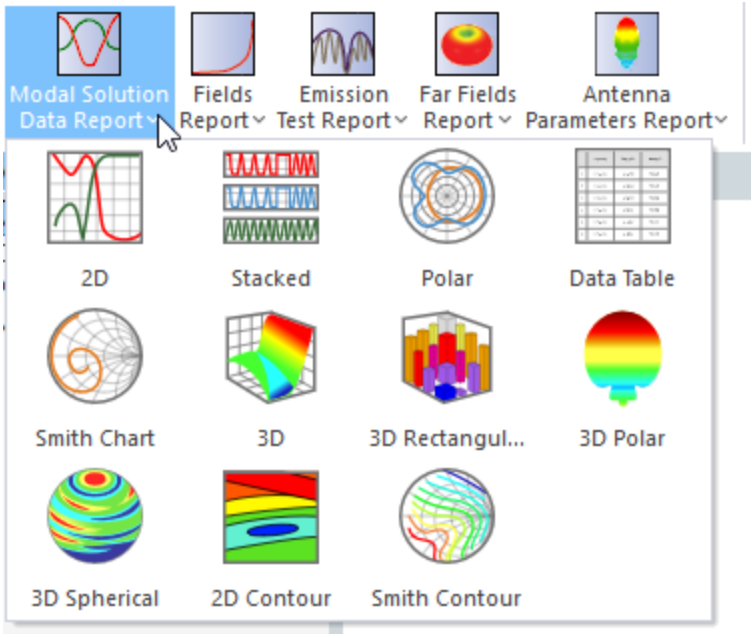
Report Type	Display Type
Create Standard Report	A Standard report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Polar Plot • Smith Chart • Rectangular Stacked Plot • 3D Rectangular Plot • 3D Rectangular Bar Plot • 3D Polar Plot • 3D Spherical Plot • Rectangular 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:



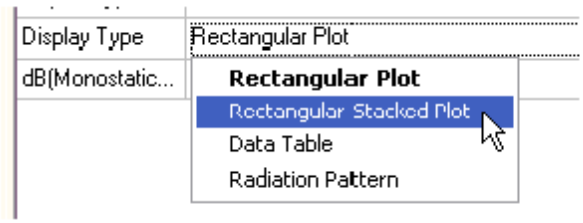
For the initial plot, you can also select from the following **Display Type** formats in the **Create*<type>* Report** submenu:

Rectangular Plot	A 2D rectangular (x-y) graph.
Rectangular	This choice puts each trace into its own 2D

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stacked Plot	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** window for that plot. Select the Report icon in the Project tree to display the Properties window. 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 Plots

A rectangular plot is a 2D, x-y graph of results.

1. On the **Results** menu (under the **Icepak** 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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**.

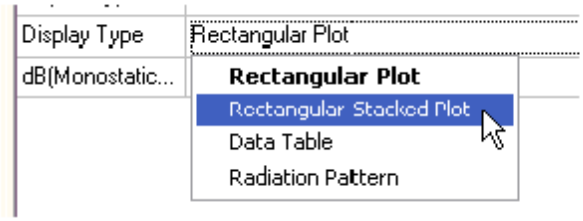
    This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog.

    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.

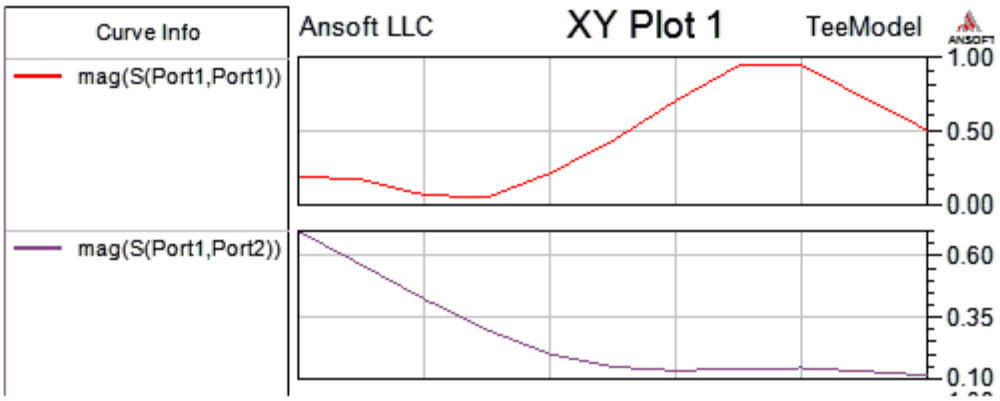
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Stacked Plots

A rectangular stacked plot is a 2D, x-y graph of results, with each trace displayed on a separate plot.



1. On the **Results** menu (under the **Icepak** 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.**

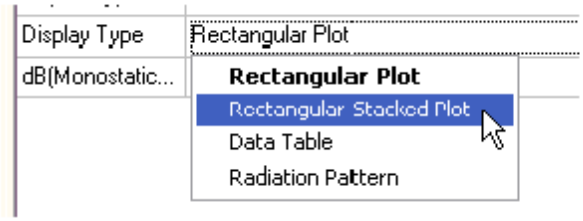
   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project 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.

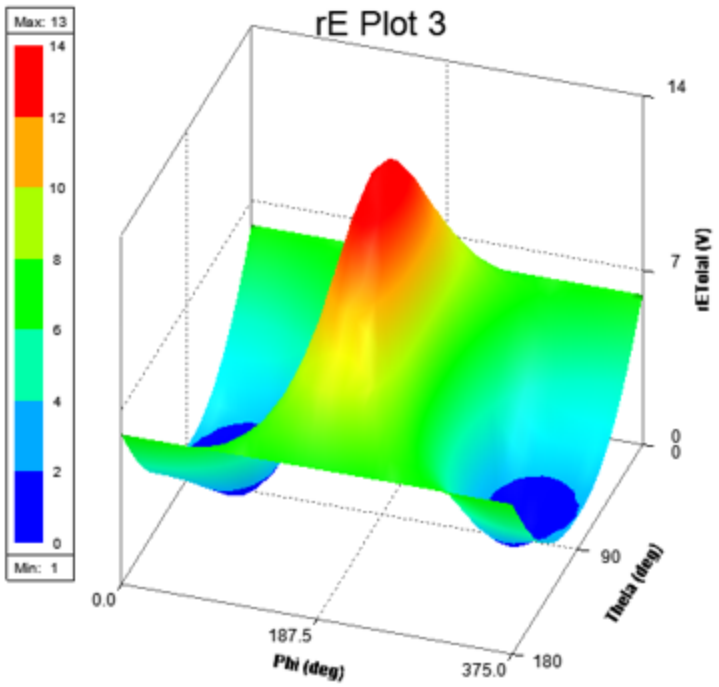
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you make a selection, the plot display updates for the current selection.

Creating 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

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 **Icepak** 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:

Ansys Electromagnetics Suite 2026 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.

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. 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.

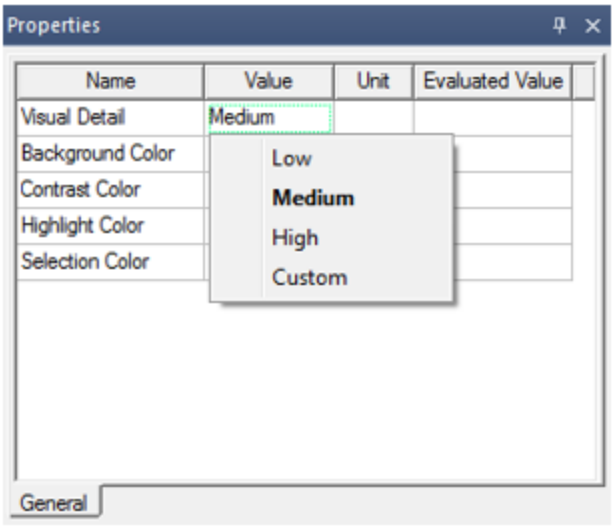
   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

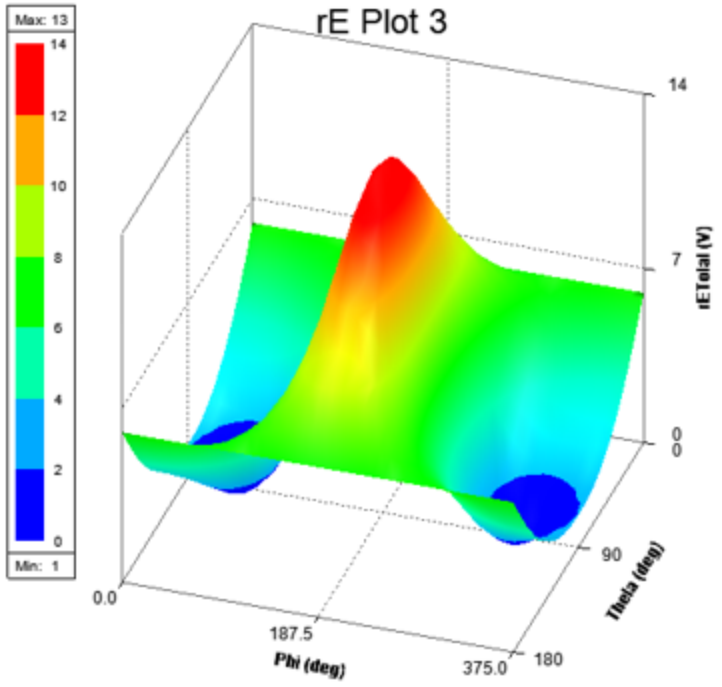


The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail 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

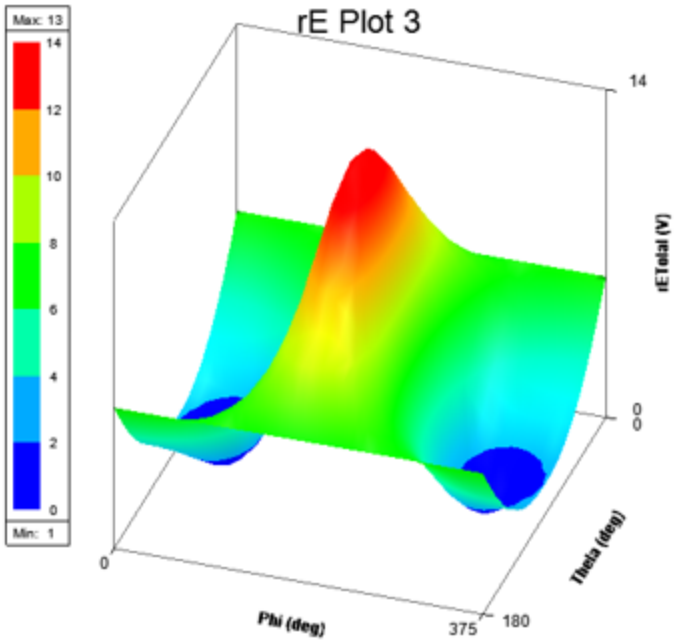
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

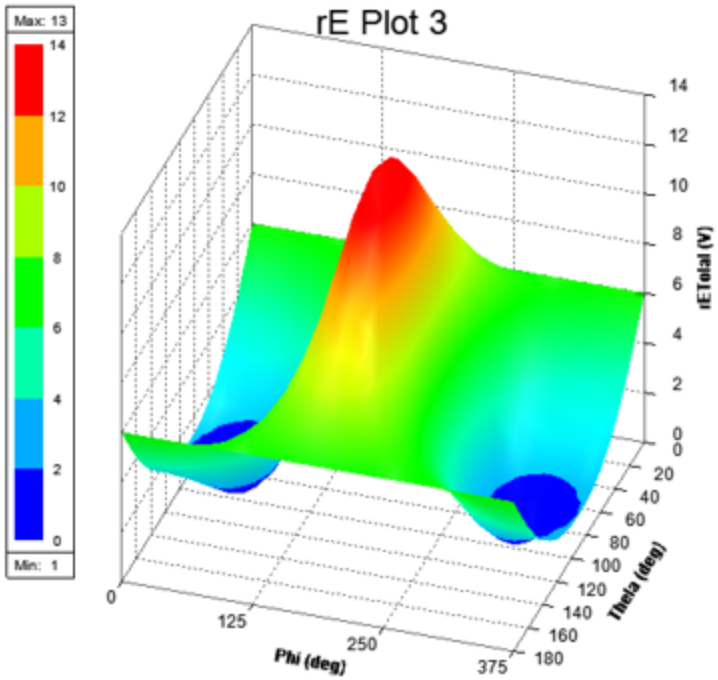
3D Rectangular Plot with Low Visual Detail

Ansys Electromagnetics Suite 2026 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

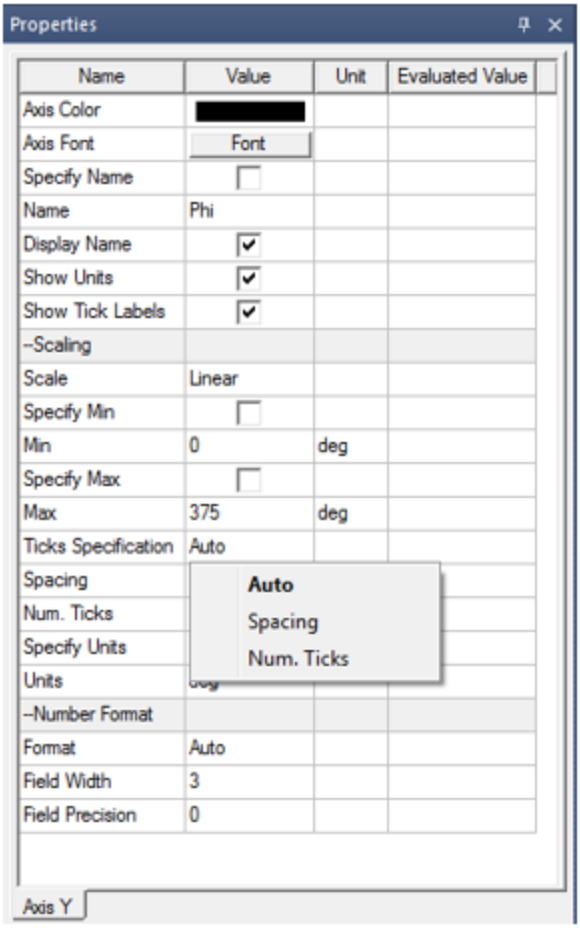


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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 (**Icepak** 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 2026 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 2026 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.

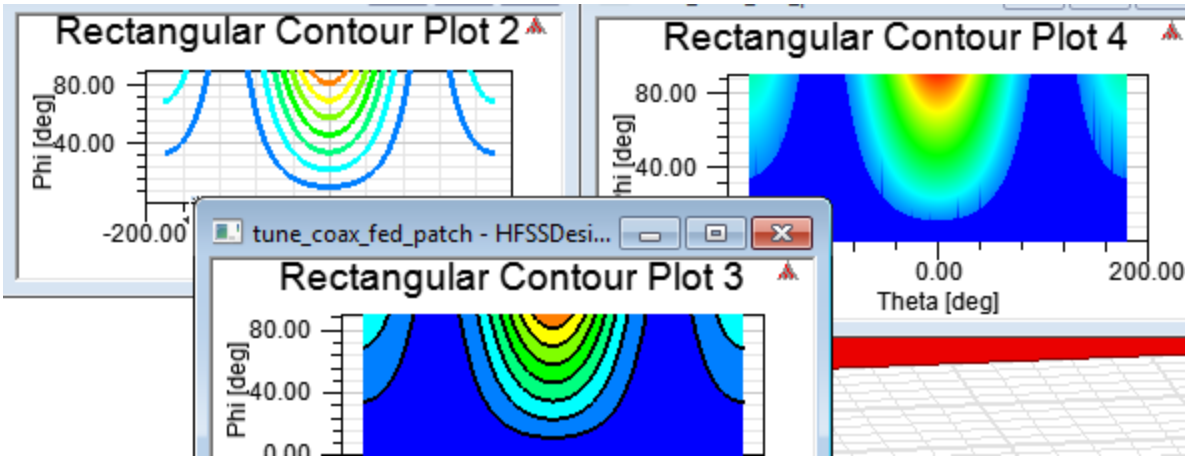
The Trace Properties for Cartesian plots include tabs for:

- 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

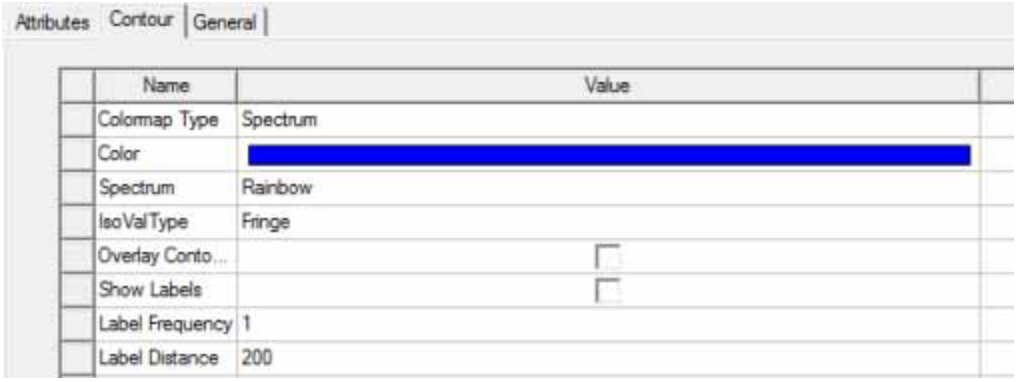
IsoValType can be Line, Fringe, or Tone.



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 2026 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 2D Polar Plots

In Icepak, a polar plot is a 2D circular chart divided by the spherical coordinates R and theta, where R is the radius, or distance from the origin, and theta is the angle from the x-axis. Following is the general procedure for drawing a polar graph of results:

1. On the **Results** menu (Icepak menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **Polar Plot** from the report type menu.

The **Report** dialog 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 ***<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.
3. In the **Trace** tab **PolarComponent** area, specify the information to plot:
    a. On the **Category** drop-down menu, click the type of information to 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.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

4. 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.

5. 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 **Icepak** > **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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

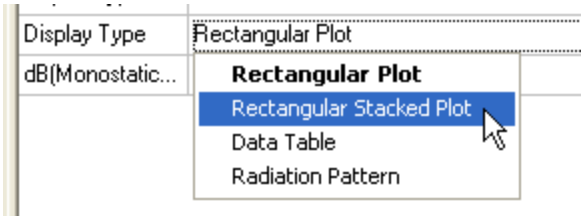
5. On the **Trace** tab **X (Primary sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values.
6. Click **New Report**.

   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.



Ansys Electromagnetics Suite 2026 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.

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).
- 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 2026 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 2026 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 2026 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.)

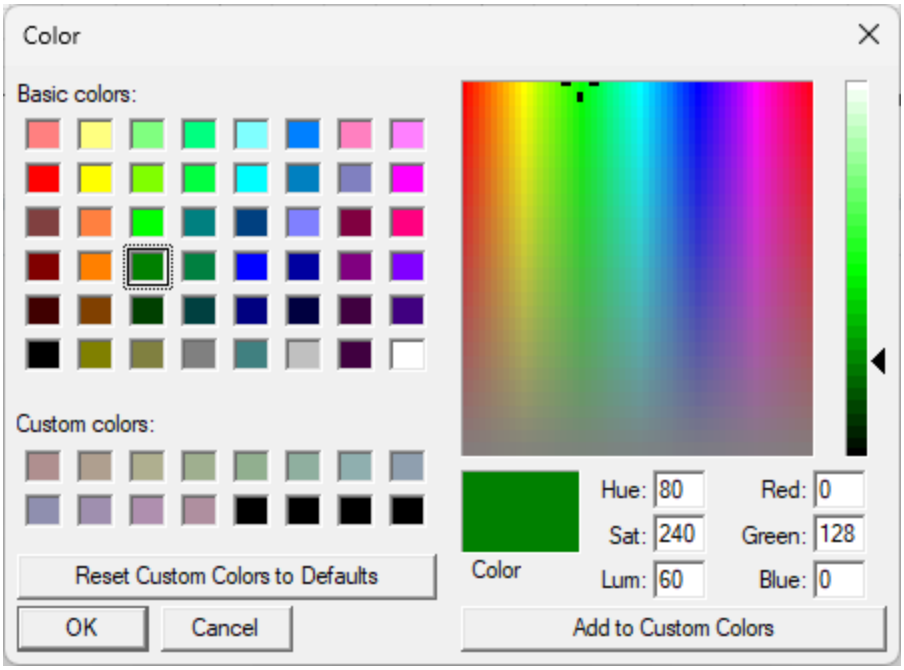
   The *Attributes* tab properties apply specifically to the trace. The defaults are set in the Report2D options. They include:

   - Name – not editable by selecting the trace from the Report. It shows the characteristics of the trace as defined in the *Report* dialog box.

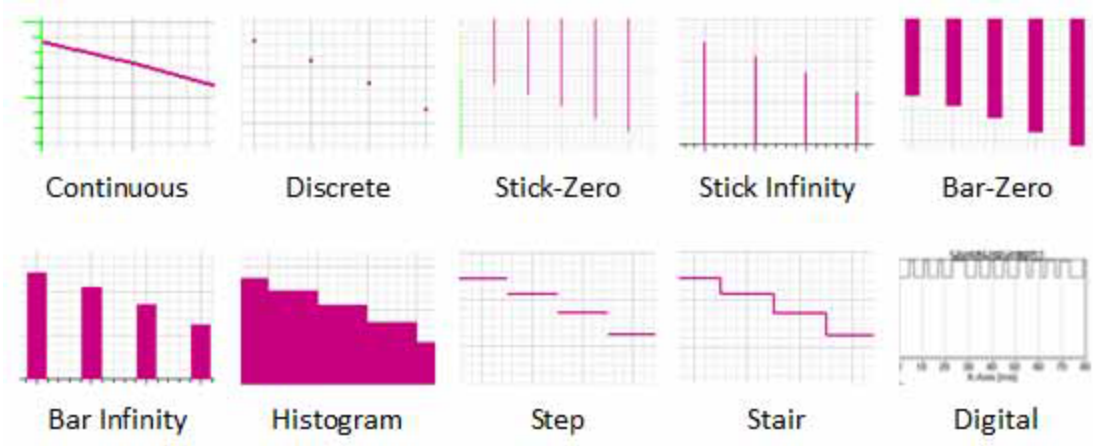
     To edit a trace name, see Editing Trace Properties.

   - 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



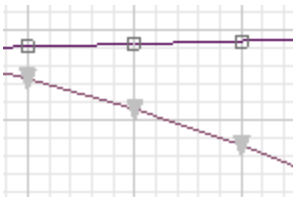
- Line style – a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width – a text field lets you edit the numeric value.
- Trace type – the drop-down menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.



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 2026 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.

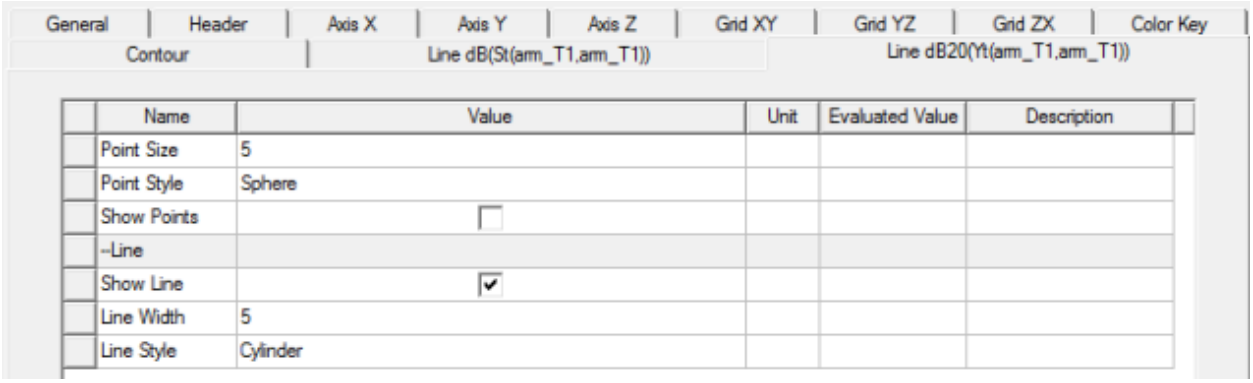
3. Edit the properties, if needed. Click OK to apply the changes and close the window.

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

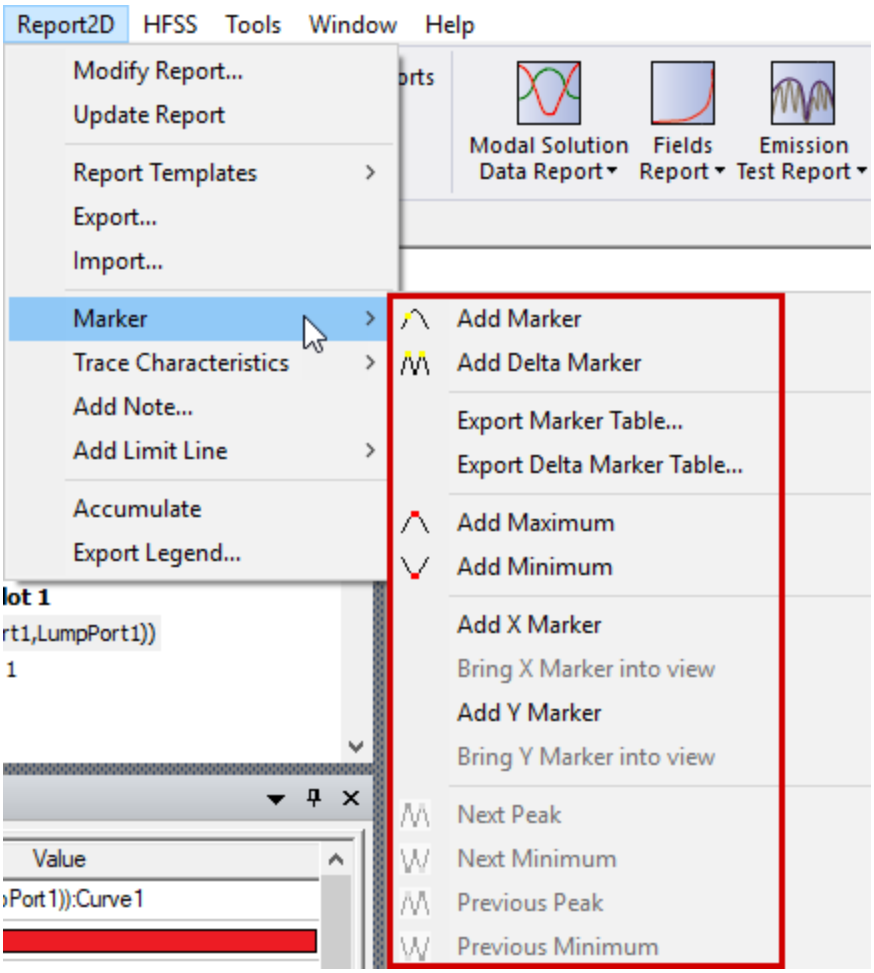
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 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 2026 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.

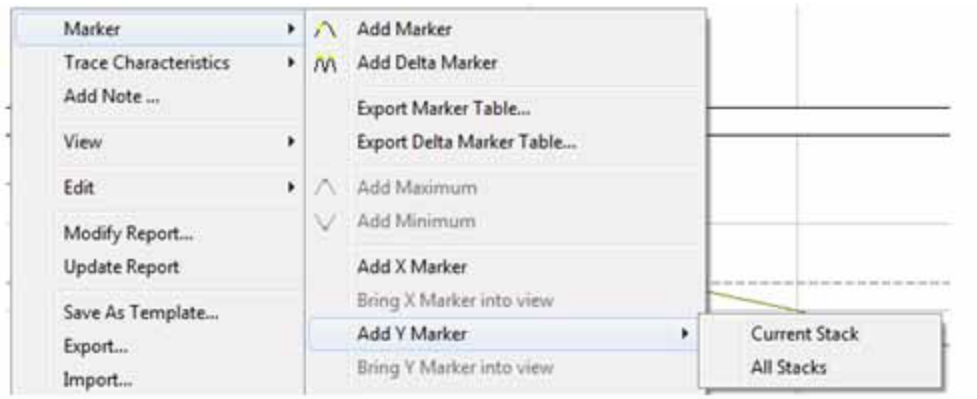
Y Markers in Stacked XY Plots

Y Markers allows for easy analysis and comparison of curves at a particular y-coordinate. Y Markers can be used to compare stacked curves.

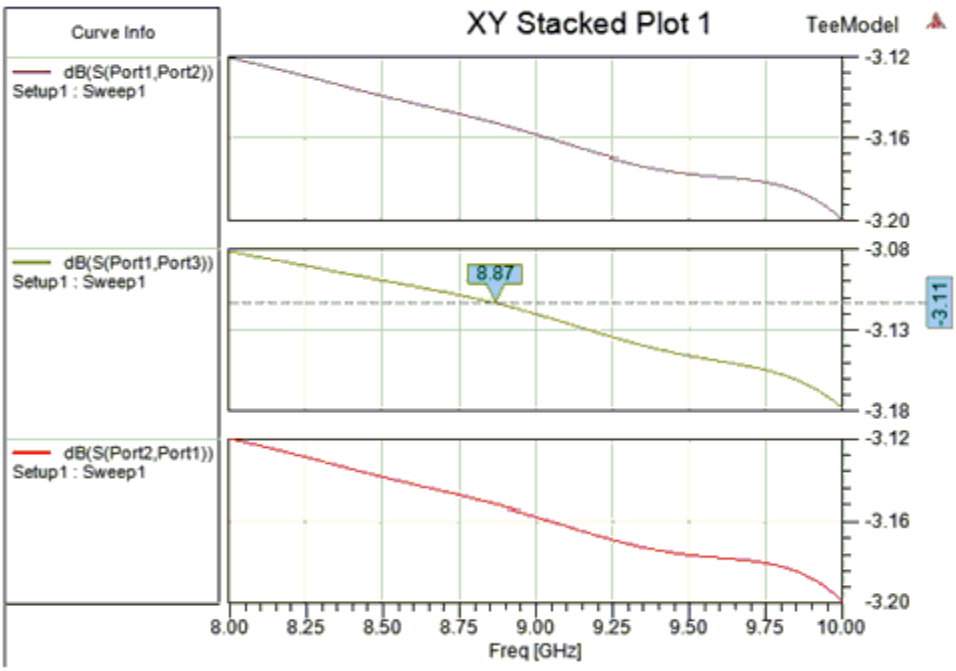
Creating Y Markers in Stacked Plots

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

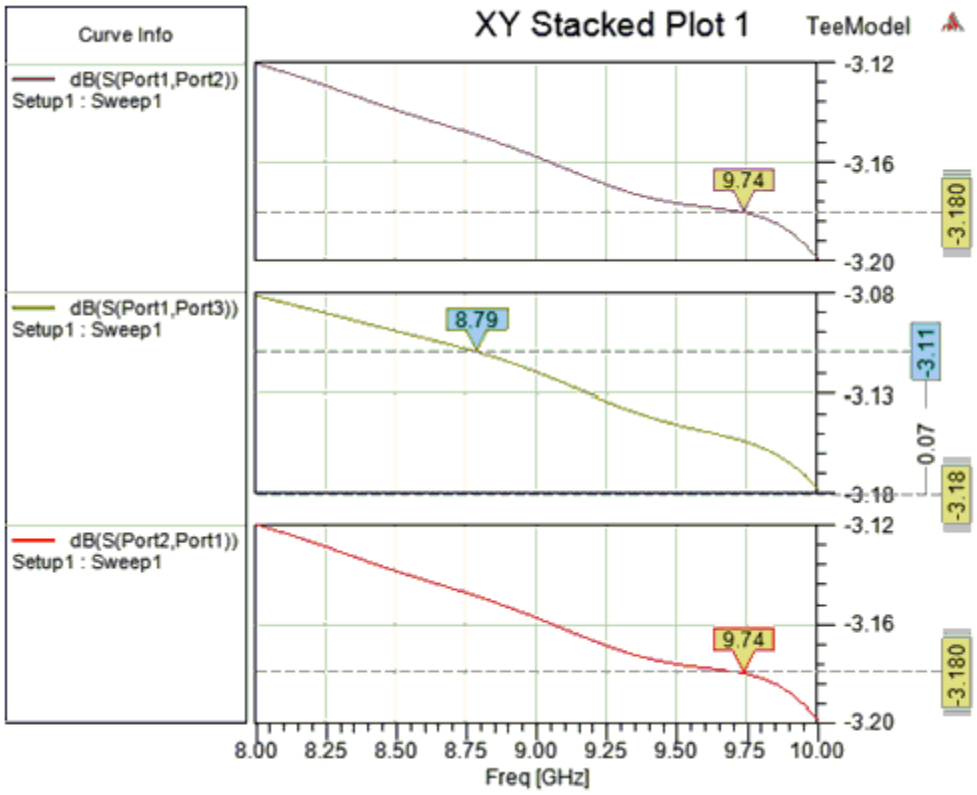


Add Y Marker > **Current Stack** creates a Y marker for the stack on which user performed right mouse button click. The following figure shows that a Y Marker was added to second stack only:



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

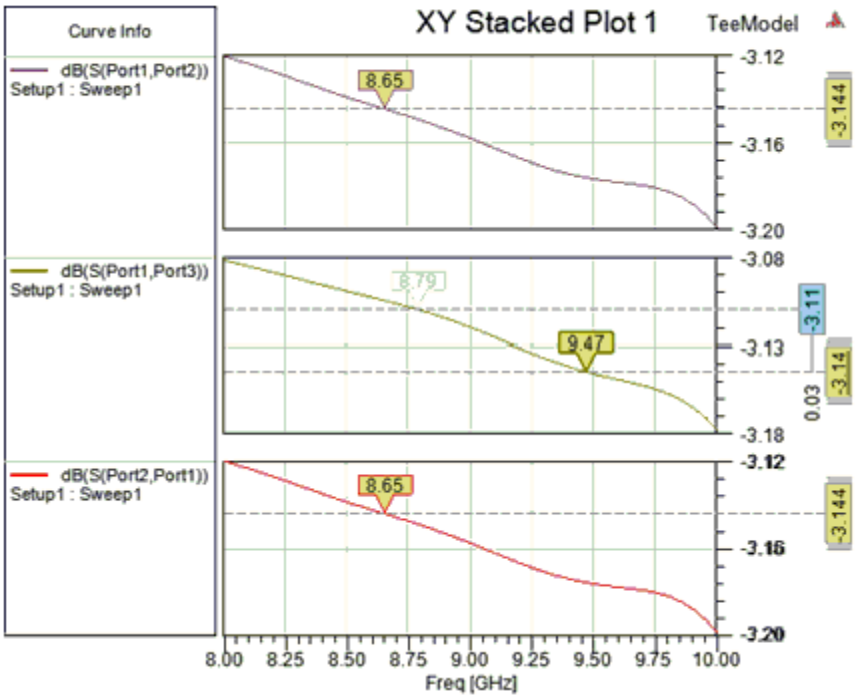


Notice that the Y Marker for All Stacks has a different appearance than the Y Marker for a particular stack, that is, it has double parallel lines above and below the Y Marker textbox.

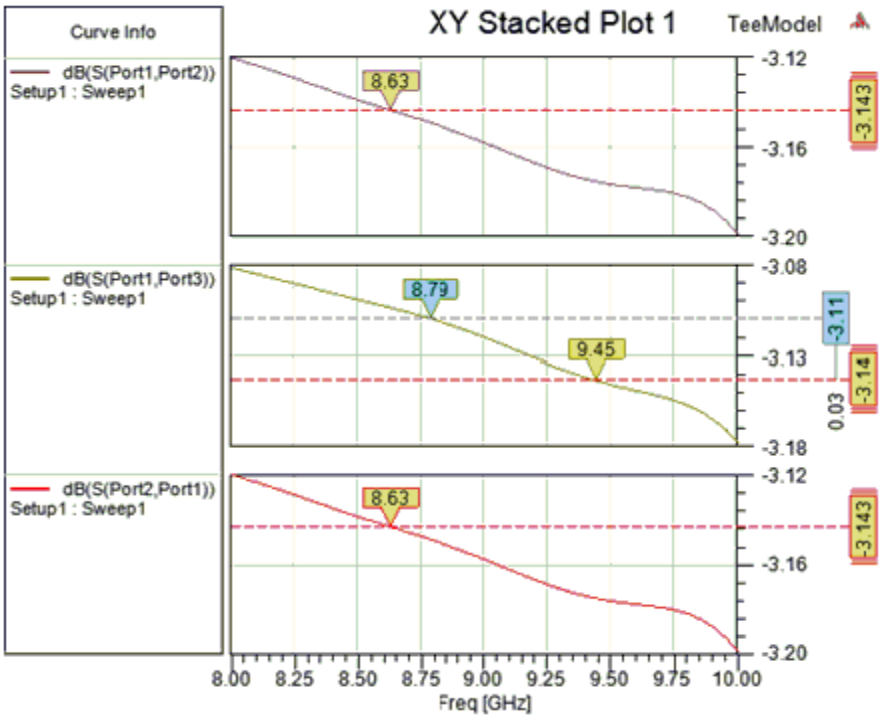
Synchronized Y Markers

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



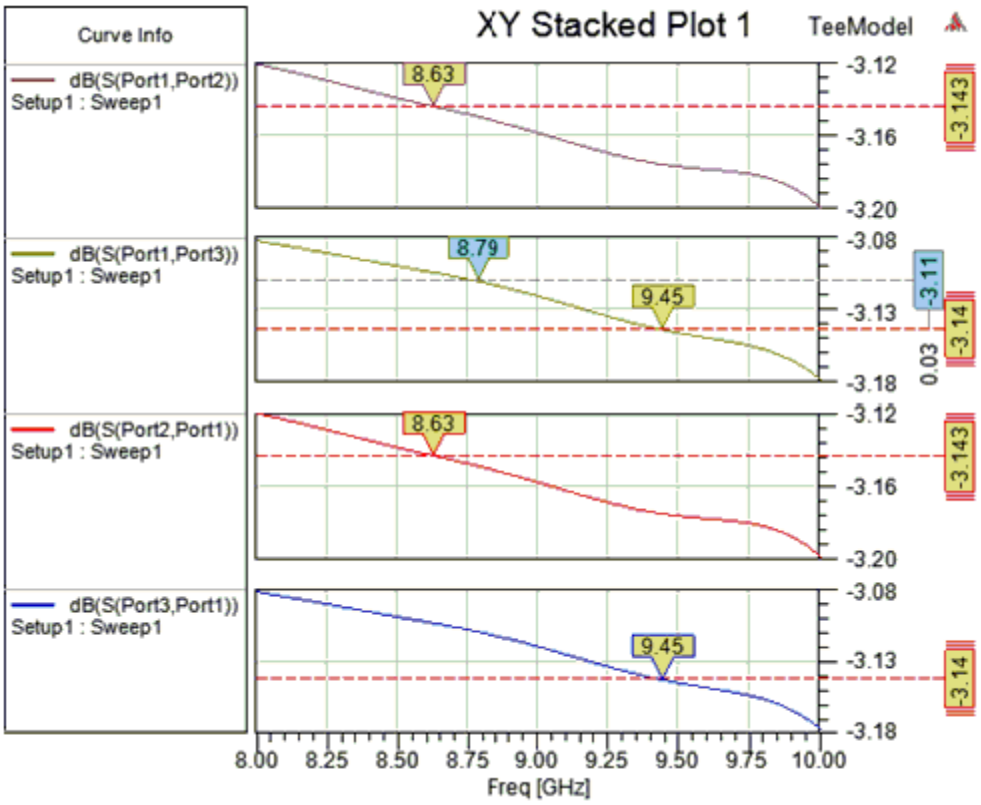
Also if a property of any one Y Marker is changed, all the "same" Y Markers show the change in property as well. For example the following figure shows that when the line color of a Y Marker in the top stack was changed to red color, a Y Marker in bottom stack show the same line color as well:



Ansys Electromagnetics Suite 2026 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

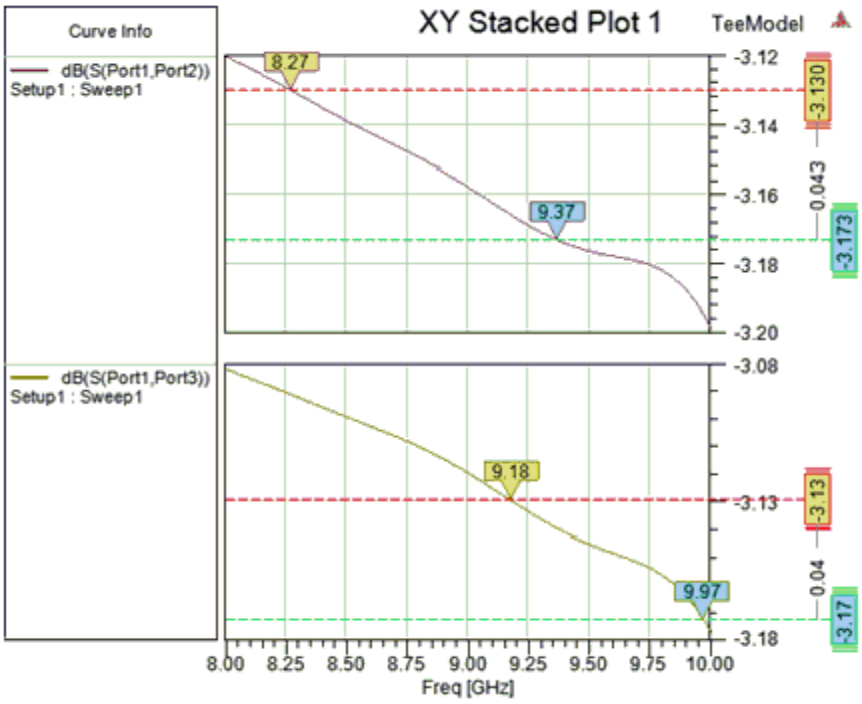
When a new curve is added to the plot, it gets all the Y Markers for all stacks in other stacks, excluding the Y Marker for particular stacks. The following figure shows that when the new curve "dB(S(Port3, Port1)" was added, a Y Marker was added to it with value -3.14 and it has all the same properties as other "same" Y Markers in other stacks:



Y Marker Delta Annotations

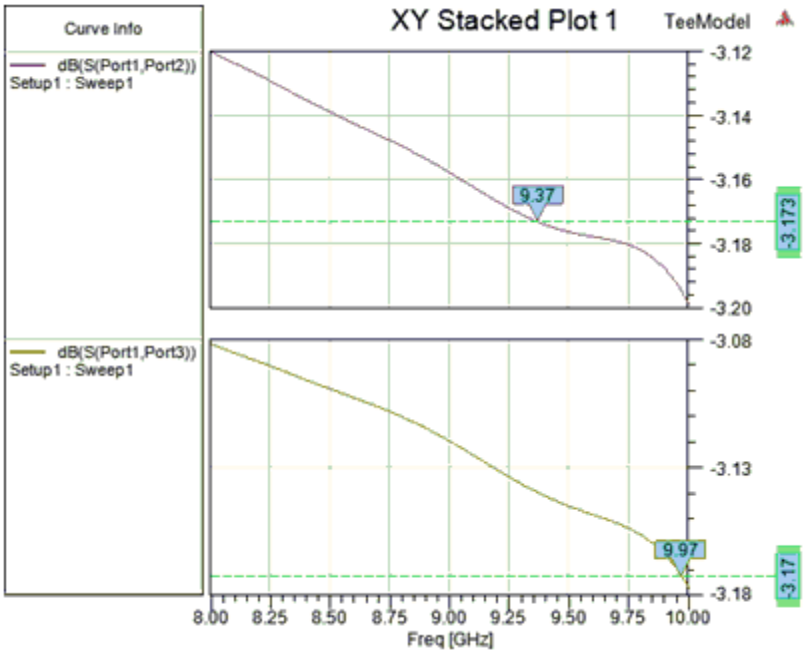
When two more Y Markers are present in a Stacked Eye Diagram, then delta annotations are shown between a pair of adjacent Y Markers in all the stacks, as shown in figure below:

Ansys Electromagnetics Suite 2026 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:

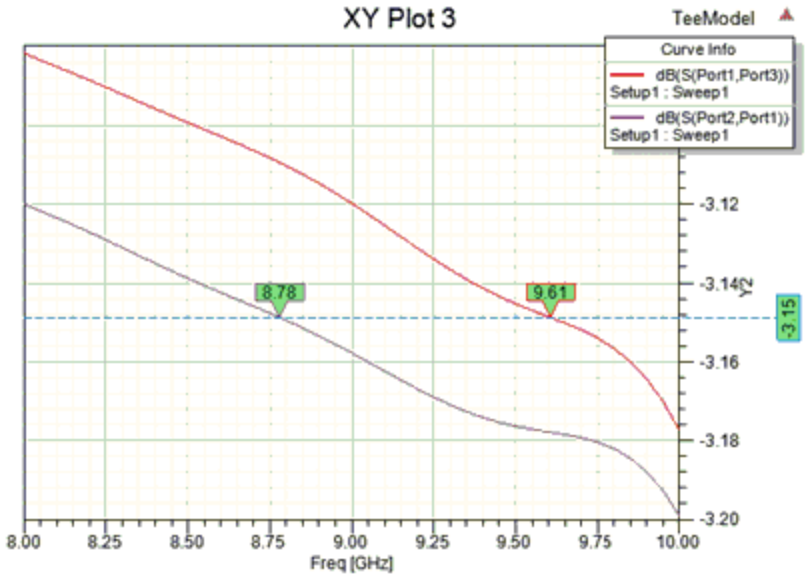


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

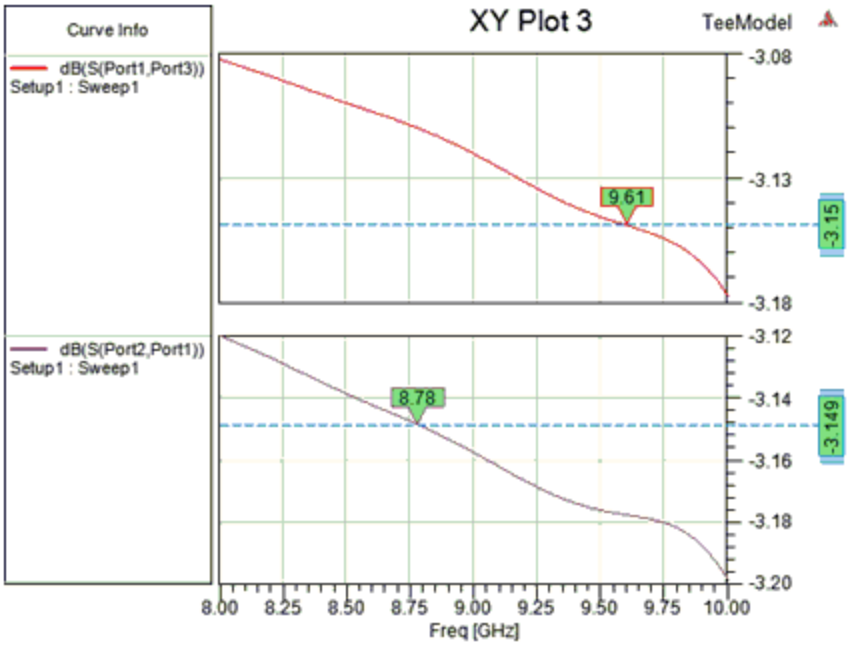
Note that on deleting a stack, Y Markers in other stacks are not affected.

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):



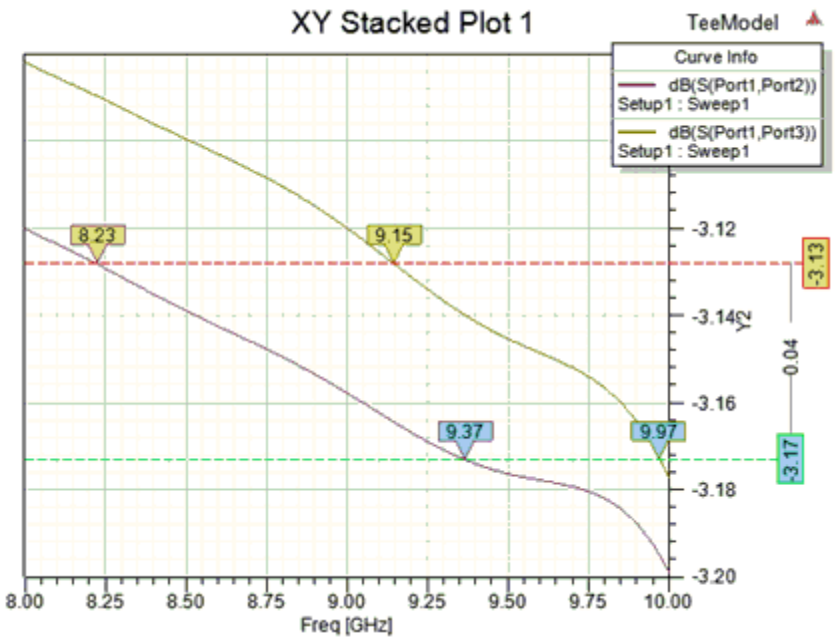
If you change the Display Type property of this plot to Rectangular Stacked Plot then a Rectangular Stacked XY Plot is created with each curve in its own stack and a Y Marker is shown in each stack with value -3.15 (blue Y Marker):



Ansys Electromagnetics Suite 2026 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.

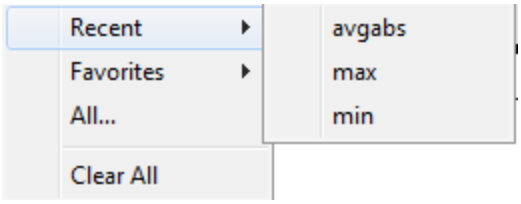
   This opens the **Properties** window for the Axis.

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.

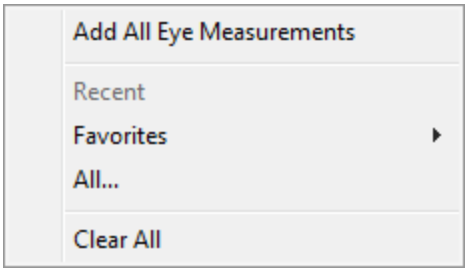
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 shortcut 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 2026 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 shortcut 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

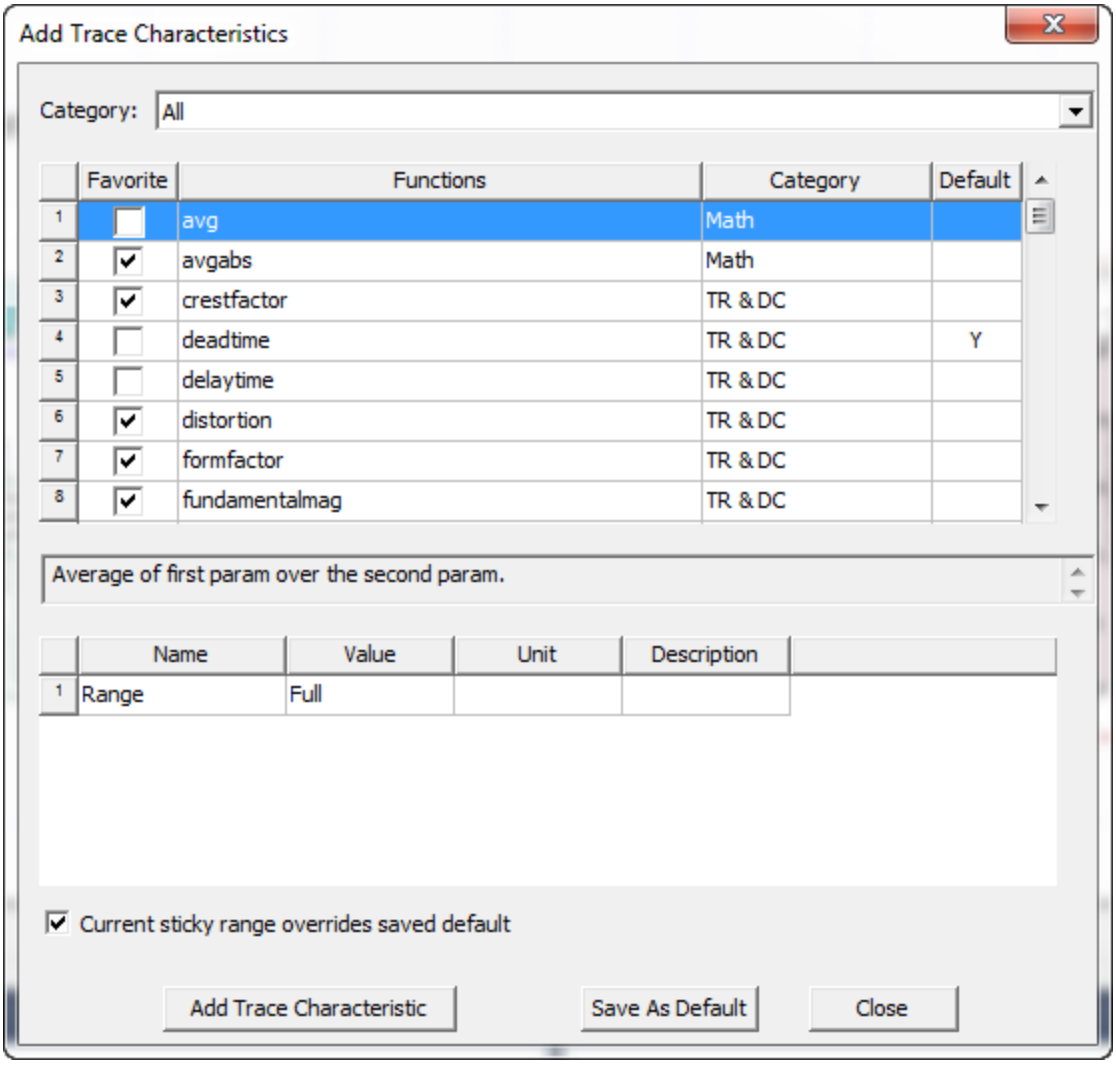
You can add trace characteristics to your list of favorites.

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 shortcut menu.
3. Select **All**.

   The *Add Trace Characteristics* dialog box displays.

Ansys Electromagnetics Suite 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

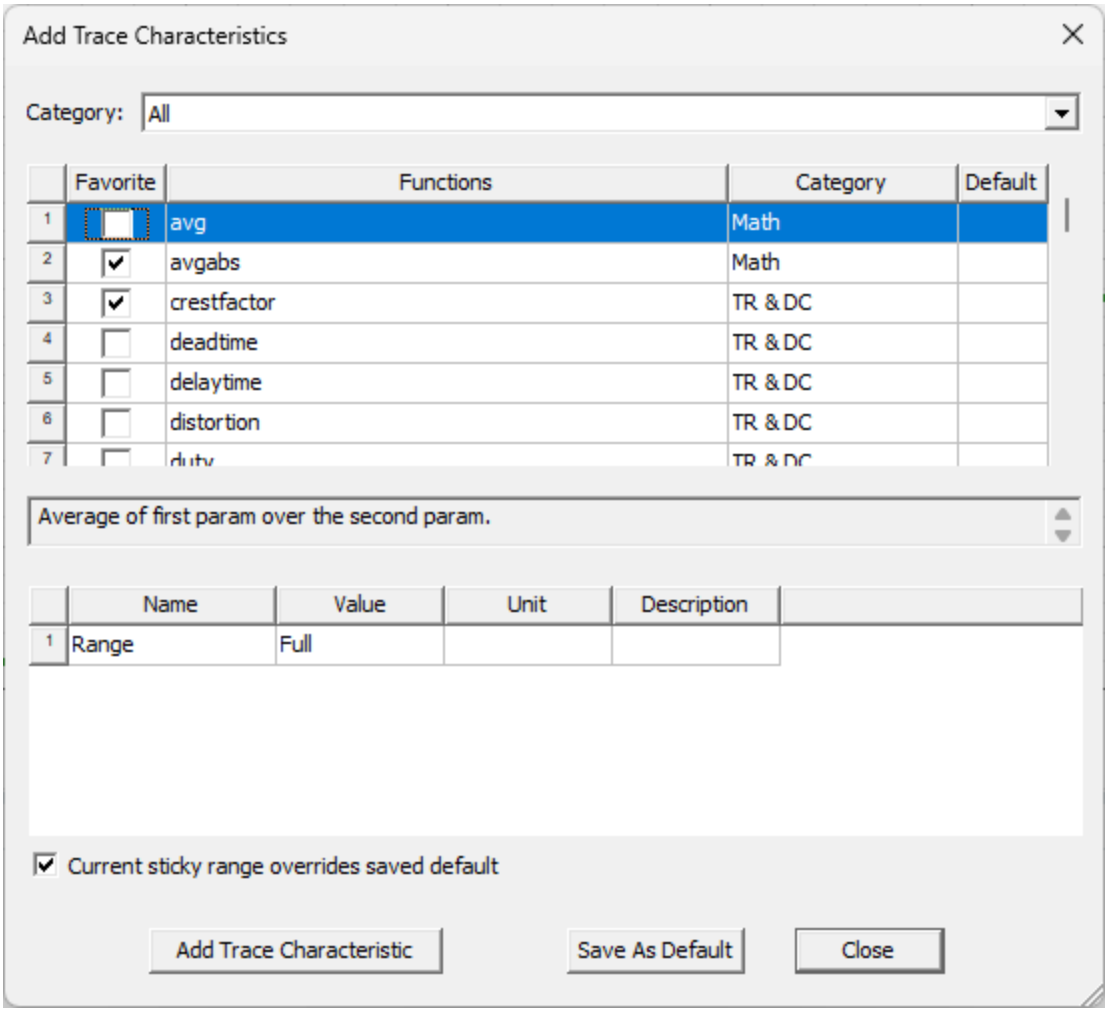
Adding Characteristics Using the Add Trace Characteristics Window

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:



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
7. Click the **AddTrace Characteristic** button to add the specified characteristics to the trace.
8. Click **Close**.

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.

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**.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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).

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

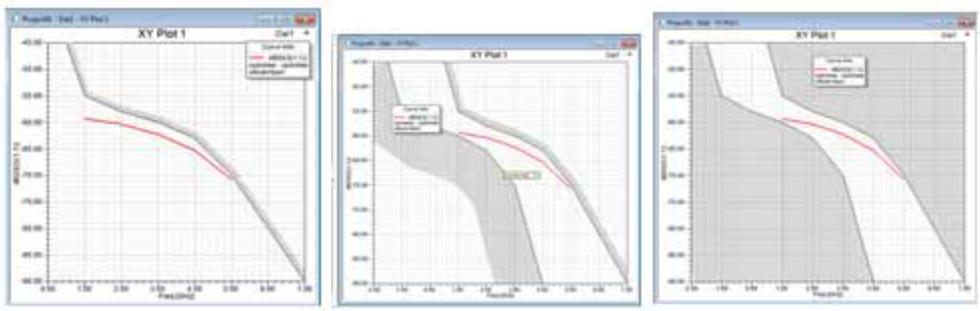
Right-click the report in the target design to which you would like to copy the trace data and select **Paste**.

> **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.

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)



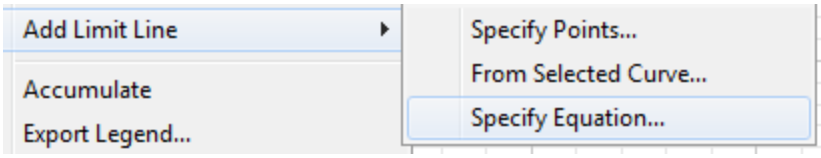
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

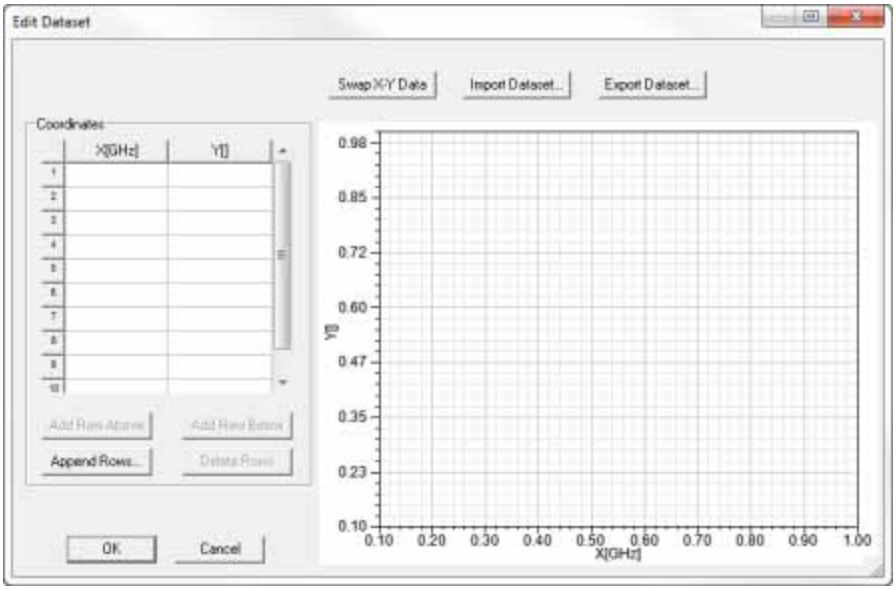
Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked. On 2D Plots, the axes extents are based on the extents of the curves.

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**.

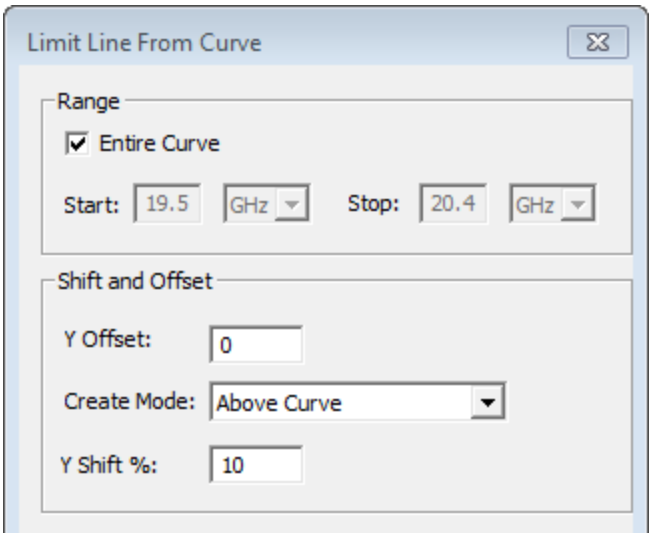


Select **Specify Points** to open an *Edit Dataset* dialog box so that you can specify points.

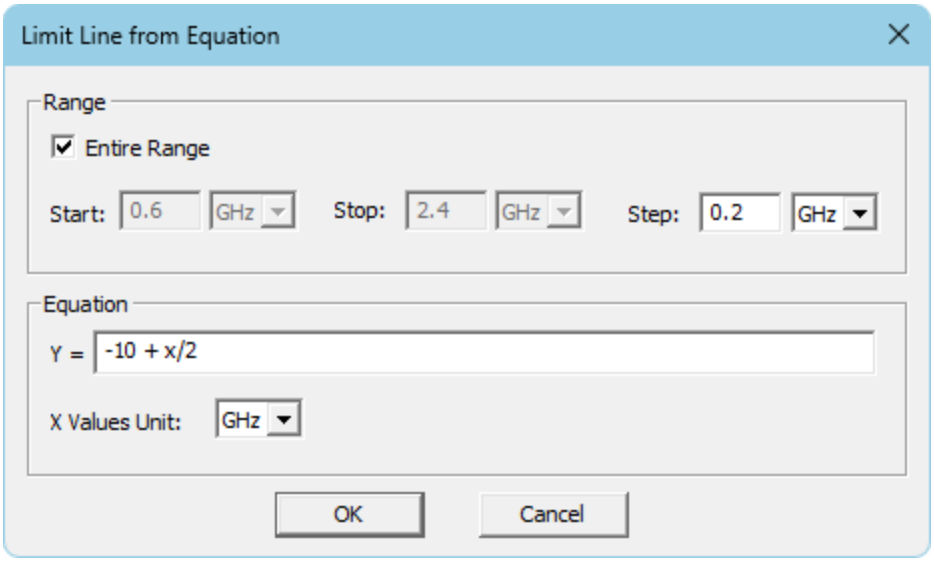


Select **From Selected Curve** to open the *Limit Line From Curve* dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 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 do the following:
    - enter the EY values directly.
    - import XY values from a .tab file.
    - export Dataset to a file.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

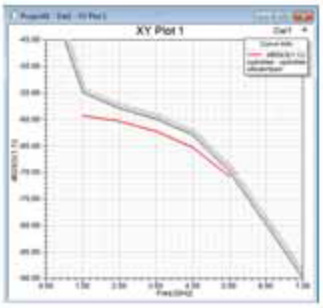
> **Note:**
>
> Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale and so on). This Y axis association defaults to the first available Y axis when the limit line is created. However, 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:
   - Define a range using the Entire Curve, or a specified Start and Stop.

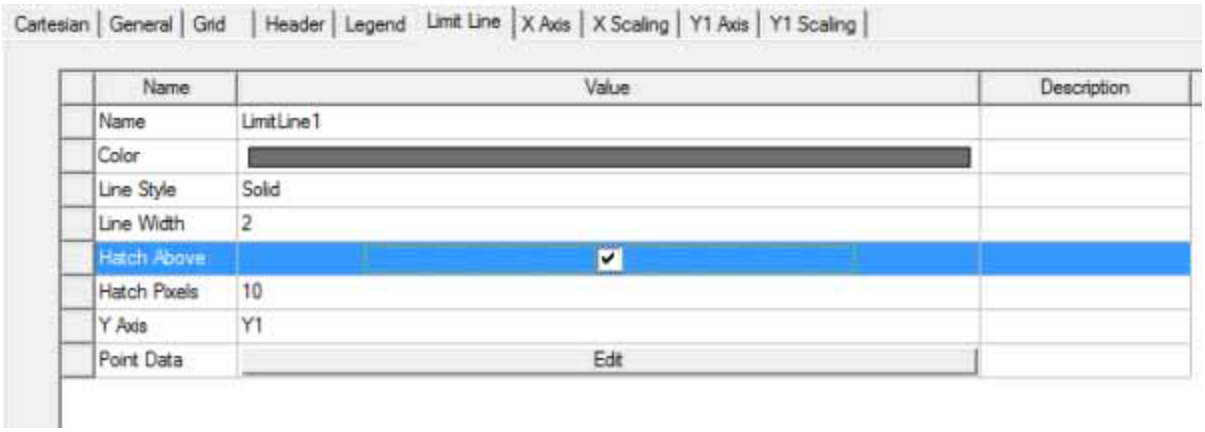
   If you uncheck “Entire Curve,” the Start and Stop fields are enabled and initialized based on the zoom level.

   - 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. You can select the limit line in the plot to edit its properties via the **Limit Line** tab of the plot properties.



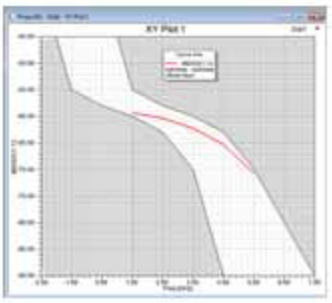
   - Line properties: Color, Style and Width
   - Y axis association
   - Hatch properties
   - Hatch width in pixels
   - Hatch direction (hatch above or below the Limit line)
   - The point definition of the limit line itself.

   For a limit line specified as an Equation, the Properties also include:

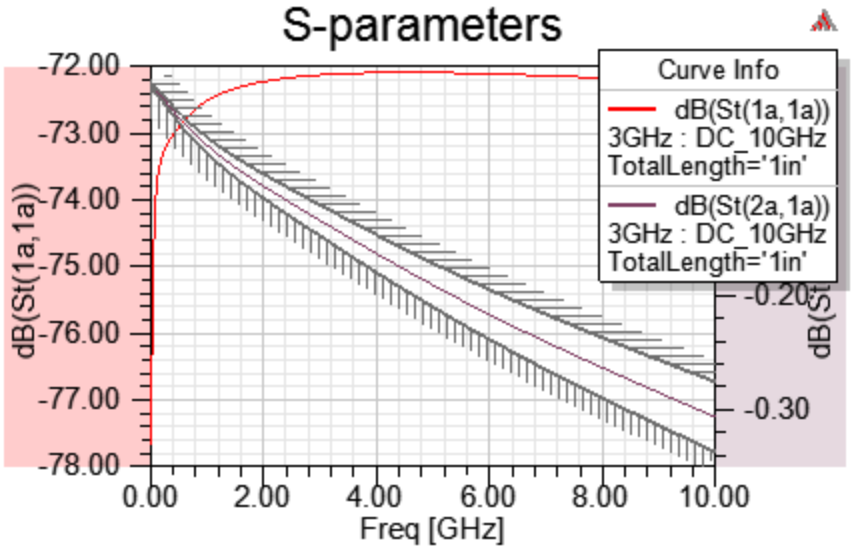
   - Start, Stop, Step, and Equation values.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



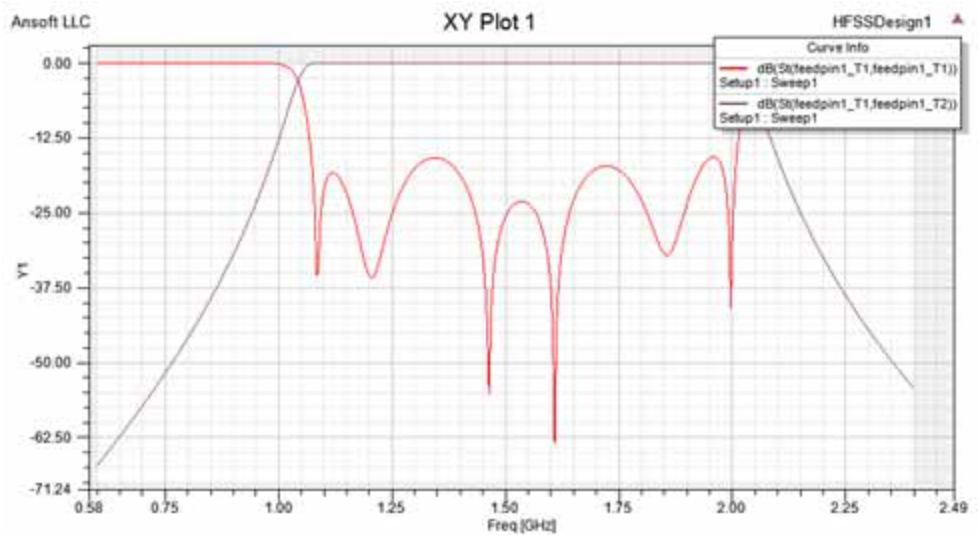
The following example shows the a limit line from curve plot, where the Hatch Above property for the lower limit line has been unchecked.



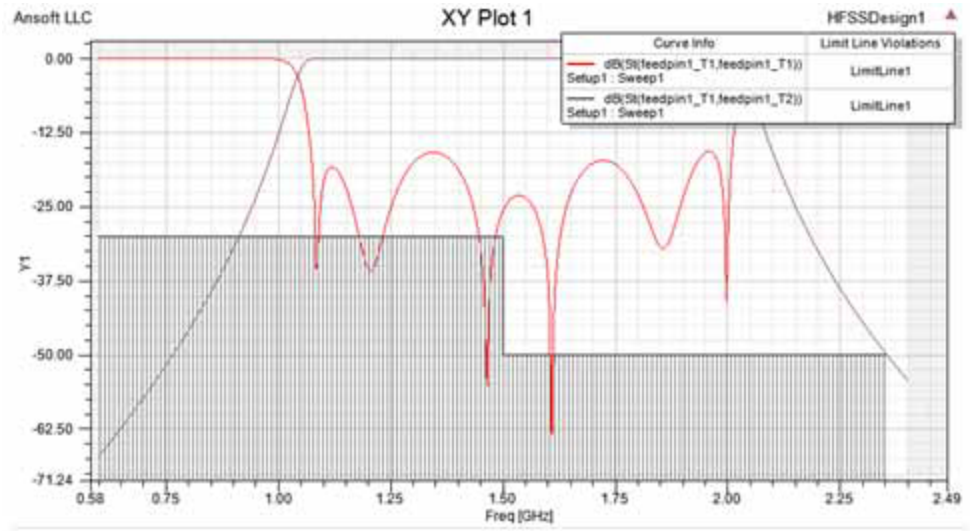
Limit Line Violations

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

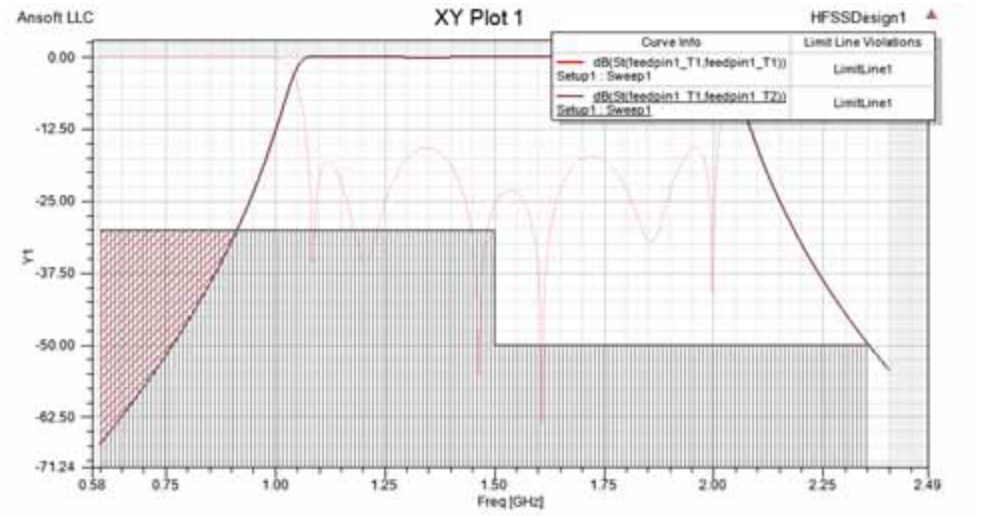


Suppose that the response cannot be below -30 dB until 1.5 GHz, and cannot be below -50 dB at 1.5 GHz or higher. You add a limit line for this requirement in the plot using 'Add a Limit Line' functionality. The plot automatically calculates whether a curve violates this requirement, that is, the limit line and show it in the legends window, as shown in following figure:



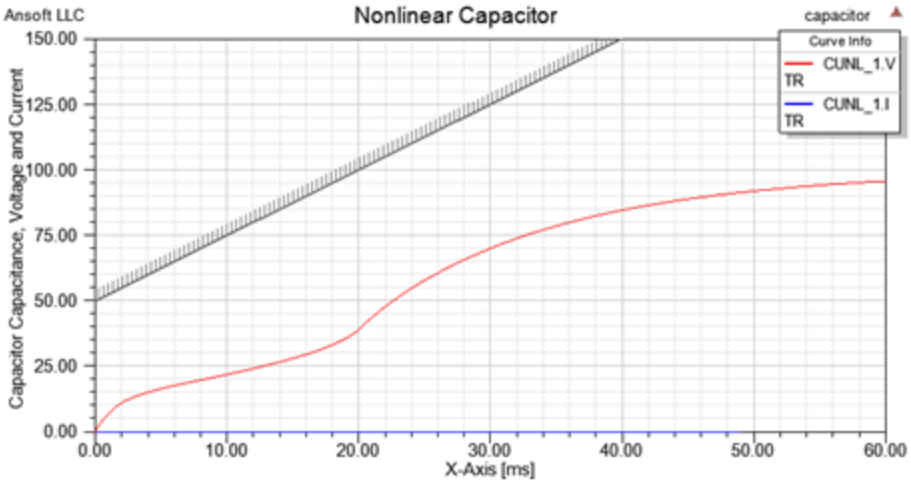
If a curve is selected, then the plot shows the region of the curve that violates the limit line (shaded with slanted red lines), as shown in following figures:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sweeping a Variable in a Report

In Icepak, a swept variable is a variable that typically has more than one value. You can plot any calculated or derived quantity against one or more of the swept variable's values.

For large projects or projects with many variables, you may obtain faster post processing before generating a solution by selecting which variables function as Sweep variables. Only the variables with Sweep enabled are indexed for post processing. See Adding a Design Variable and Adding a Project variable.

To specify the swept variable values to plot a selected quantity against:

1. In the **Report** dialog box, select the variable from the X (Primary Sweep) drop-down menu.
2. To modify the values that will be plotted for a variable:
   a. Click the ellipsis **[...]** button on the **X (Primary Sweep)** line of the **Report** dialog to displays a pop-up list of the possible values.
   b. Select **Use all values** or click the Edited button to display a dialog that lets you specify the sweeps to use.

   All of the selected variable's values will be plotted.

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 Icepak 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 2026 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 2026 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. **Note:** Do not include any variables related to the unit (such as **freq**) into an equation unless **cang_deg_val** is also included.
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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dBW(x)	10*log10(|x|)
dB10	10*log(|x|) to base 10
dB10normalize	10*log [normalize(mag(x))]
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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; otherwise, returns 0.
overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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	Converts the complex number in rectangular coordinates to polar coordinates
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 between the peak-to-peak value and the average value of 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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	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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
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 Icepak, a value from a calculated expression, an intrinsic variable value.

To select solution quantities to plot:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Results > Monitor 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.
Residual	Solution continuity, energy, and velocity (X, Y and Z) quantities.
Monitor	User-created flow and thermal monitor points and faces.
Network	Internal node thermal data from network boundary conditions.

3. Select a **Quantity** from the list.
4. If desired, select a **Function**.
5. Click **Close**.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was automatically calculated by Icepak 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	User defined expressions applied to derive quantities from the original

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variables	field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by Icepak, 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:

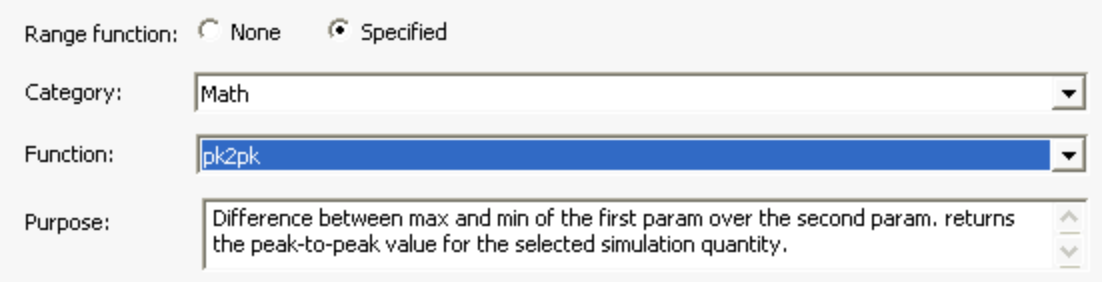
1. Click **Range Function** in the *Report* dialog box.

   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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	rSidelobeY
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime, MinEyeWidth, MinEyeHeight

> **Note:**
>
> Refer to the SI Wave or Nexxim help for more information. The Purpose field offers brief descriptions of each.

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 between the peak-to-peak value and the average value of 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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Math Functions	
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.

Radiation Functions	
IsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
IsidelobeY	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.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 positive 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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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.

Eye Measurement Range Function Parameters

The Eye Measurement category of range functions provide the means to evaluate several characteristics of eye diagrams. Each of the Eye Measurement functions includes the following parameters. Specify the Value by editing the Value text field for the parameter name. Specify the unit for the parameter by selecting from the Unit drop-down menu.

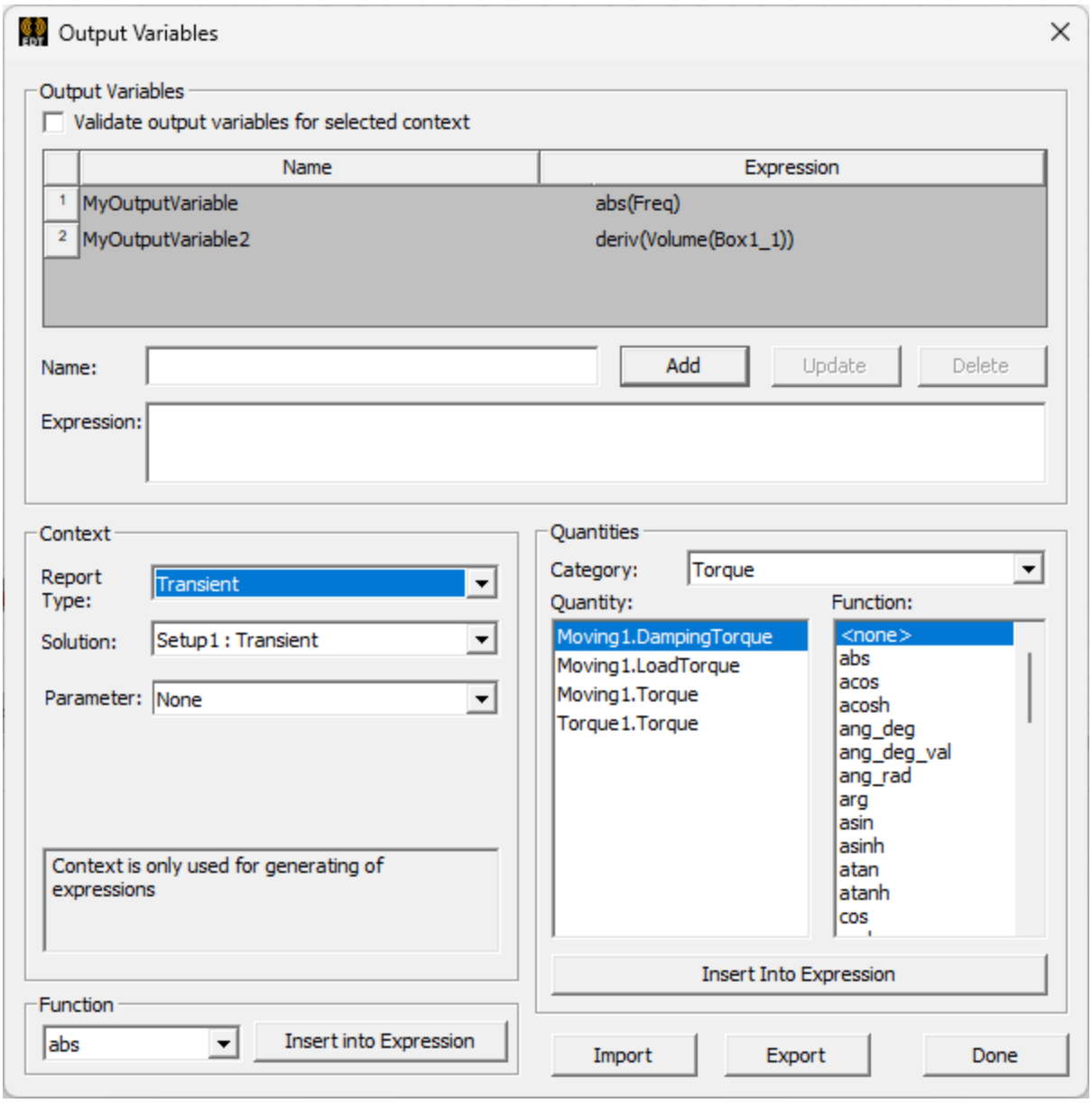
Name	Default Value	Default Unit	Description
Unit Interval	0	ns	Unit interval of signal
Start Offset	0	ns	Offset at beginning of signal
End Offset	0	ns	Offset at end of signal
Autocrossing amplitude	1		Nonzero number means that crossing amplitude is calculated automatically.
Crossing amplitude	0	mV	Specify crossing amplitude used for eye measurement data computation.

Specifying Output Variables

Access the **Output Variables** window one of several ways:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Icepak** > **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.



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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 2026 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.

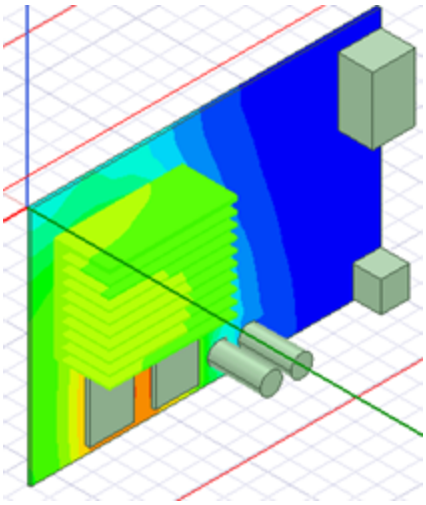
You can select from the functions in the **Output Variables** dialog box's **Function** list or type them directly into the Expression field, if necessary.

Plotting Field Overlays

Field overlays are representations of basic or derived field quantities on surfaces or objects for the current design variation. The Electronics Desktop allows for the display of existing field overlays in the Layout Editor view.

Set the design variation via the **Set Design Variation** dialog box. This dialog box is accessible from the **Solution Data** window via by clicking the ellipsis button on the right of the Design Variation field, and via the **[***solver***]** > **Results** > **Apply Solved Variation** command.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

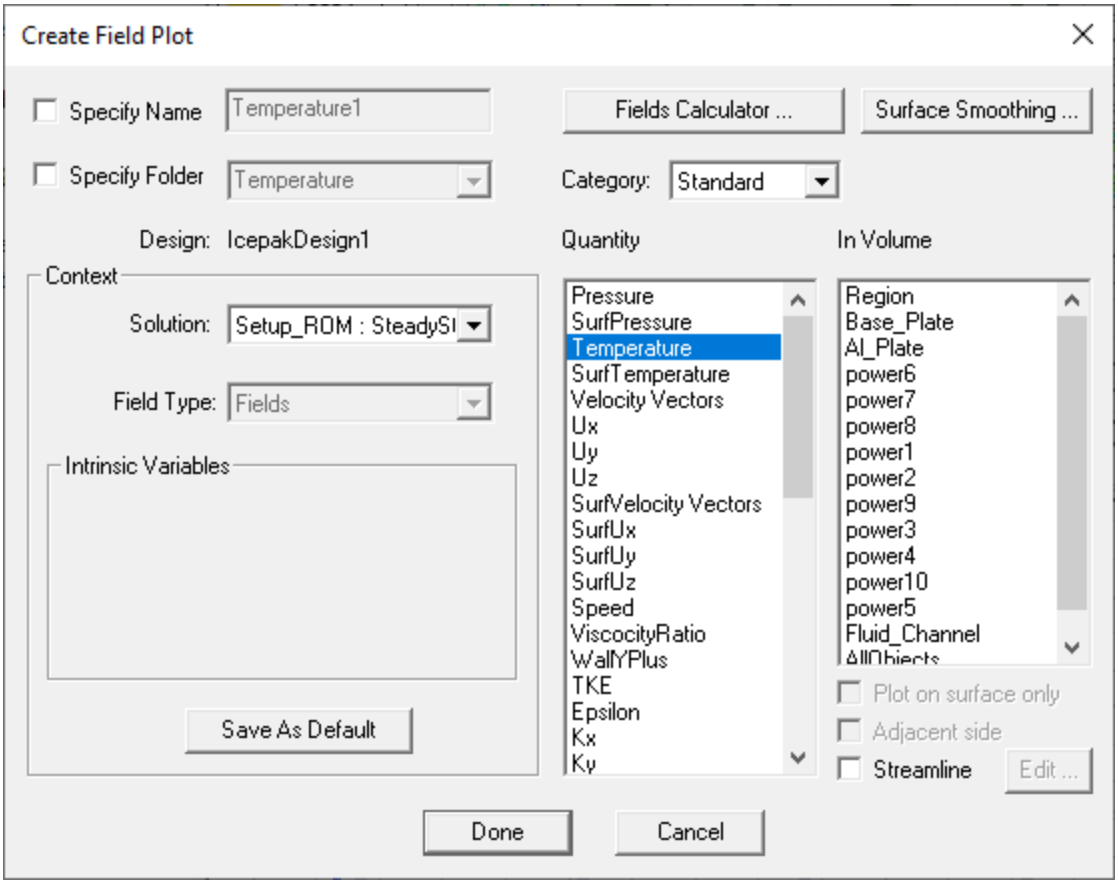


To Plot Fields

To plot a basic field quantity:

1. Select a point, line, surface, cutplane, or object to create the plot on or within.
2. Click **Icepak** > **Fields**, or right-click **Field Overlay** in the **Project Manager** and select **Plot Fields**, or right-click in the modeler window, and select **Plot Fields** from the context menu.
3. On the **Plot Fields** menu, click the field quantity you want to plot. The **Create Field Plot** dialog box appears. The **Specify Name** field shows a name based on the field quantity you selected, and the **Quantity** list shows the field quantity selected. See Icepak Post Processing Quantities for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The available selections depend on the solved solution.

If you select a scalar field quantity, a scalar surface or volume plot will be created. If you select a vector field quantity, a vector surface or volume plot will be created. If you select a vector quantity, you will be able to specify a Streamline plot. If the quantity you want to plot is not listed, see Named Expression Library.

4. To specify a name for the plot other than the default, select **SpecifyName**, and then type a new name in the **Name** text box.
5. Select the solution to plot from the **Solution** drop-down menu.
6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** drop-down menu, or type the name you wish to use. Plot folders are listed under **Field Overlays** in the project tree. Plot folders let you group plots with the same quantity together. All field plots under the same folder share the same color key.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. If desired, you can select a different field quantity to plot from the **Quantity** list.
9. Select the volume (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 with the selected object or objects. You can select and deselect any items in the **In Volume** list. You can mix model objects with non-model boxes. For example you might want to see a plot from part of two model objects by restricting the region to a non-model box overlapping those parts.

   > **Note:**
   >
   > Multiple selection should be used when there is a discontinuous field on a surface. If not, the field on both sides of the surface is plotted and each interferes with the other.

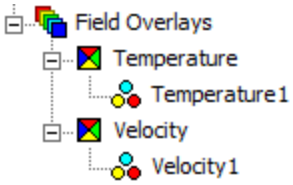
10. If you selected a 3D object in object-selection mode, you can select **Plot on surface only** to display the data on only the surface of the object.
11. If desired, for 2D geometry or the faces of 3D geometry, select the **Adjacent side** check box. See Reporting Sides for information on the data displayed.

    > **Note:**
    >
    > For joule heating cases, if no results for electric potential are displayed, enable **Adjacent sides** to display the data.

12. Click **Done**.

The field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** dialog box.

The new plot appears in the view window. It is listed in the specified plot folder in the project tree. If you have created a field plot on a simulation in progress, the field plot is updated after the last adaptive solution. Each category of plots (such as Temperature) are listed separately in the **Project Manager**.



If you want to update the field overlay, select the Field icon in the Project tree that contains the field plot of interest, right-click to display the short cut menu, and select **Update Plots**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To turn off the display of the plot, right-click on the plot and select **Plot Visibility** from the short-cut menu. Unchecking **Plot Visibility** turns off the plot display.

Plotting Streamlines in Icepak

Streamlines display particle traces for vector field plots. Use streamlines to plot and animate particle traces to visualize flow throughout the computational domain.

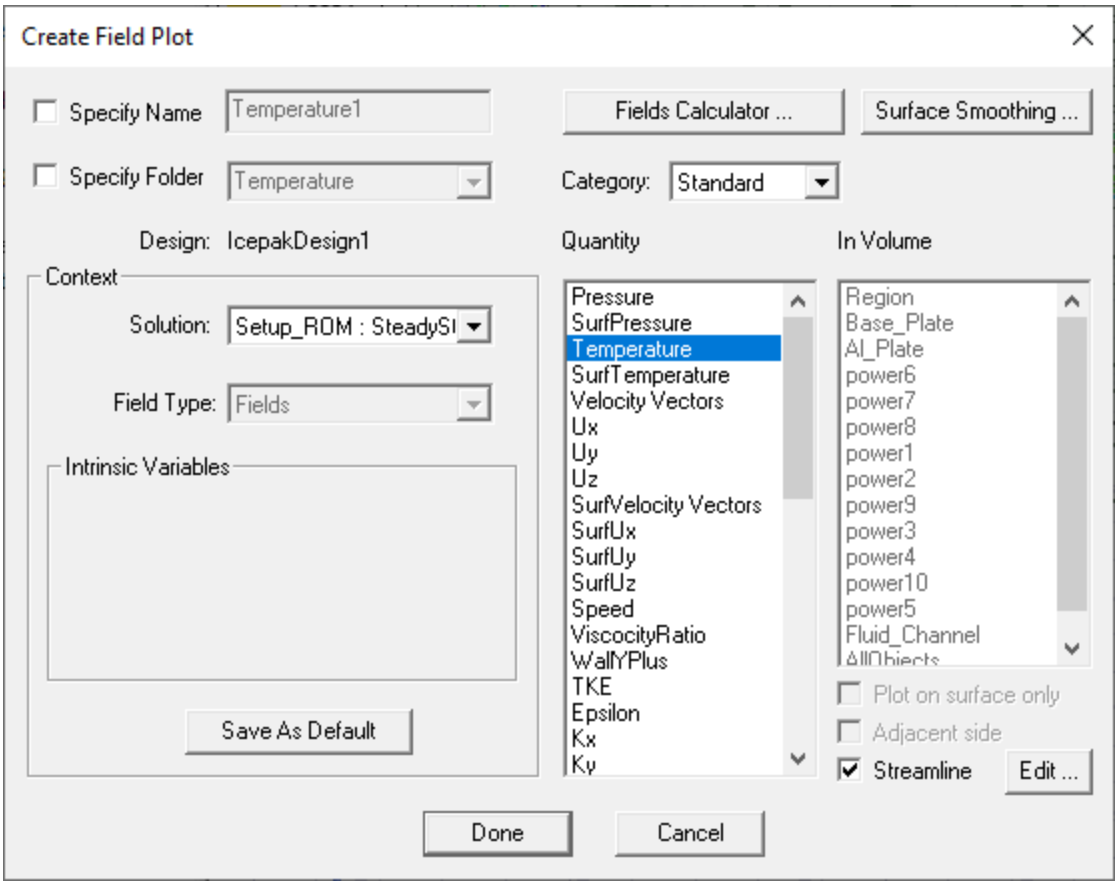
After a simulation is complete:

1. Press **F** to enter face selection mode.
2. In the **3D Modeler** window, select the face of an opening or grille boundary condition.

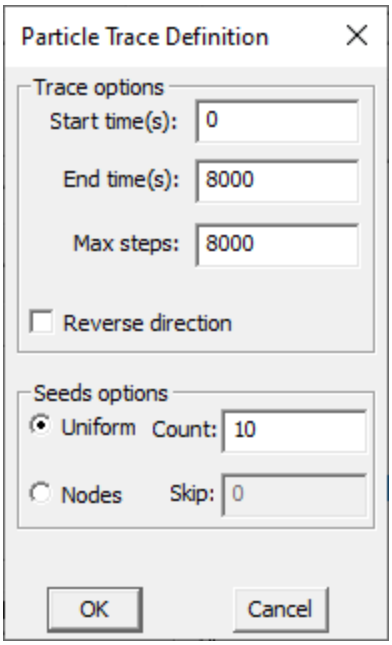
   > **Note:** In addition to these object faces, streamlines can be plotted on planes you have created. Select a plane in the History tree or **3D Modeler** window and proceed.

3. Right-click and select **Plot Fields** and then any of the following post-processing quantities:
   - **Pressure** > **Pressure**
   - **Temperature** > **Temperature**
   - **Velocity** > **Ux**, **Uy**, or **Uz**
   - **Velocity** > **Speed**
   - **Turbulence** > **ViscosityRatio**
   - **Turbulence** > **WallYPlus**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



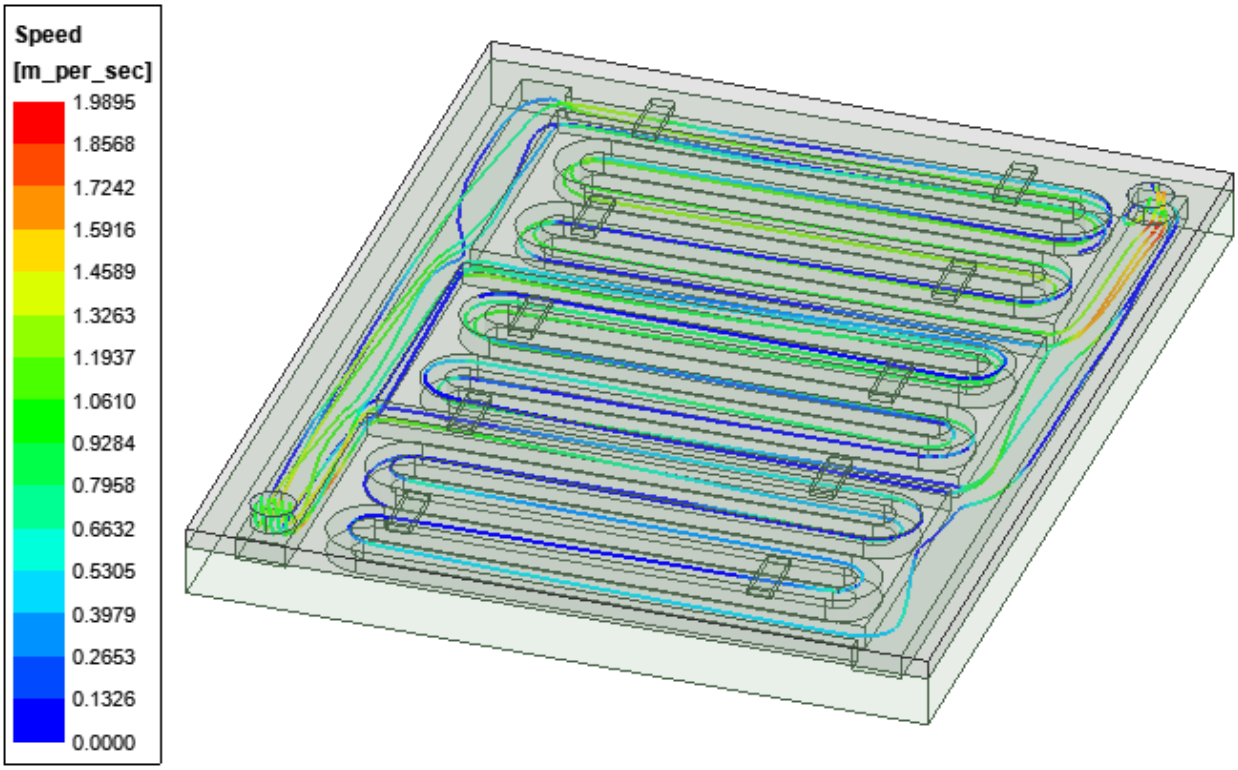
4. In the lower right corner of the **Create Field Plot** dialog box, select **Streamlines** and click **Edit**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

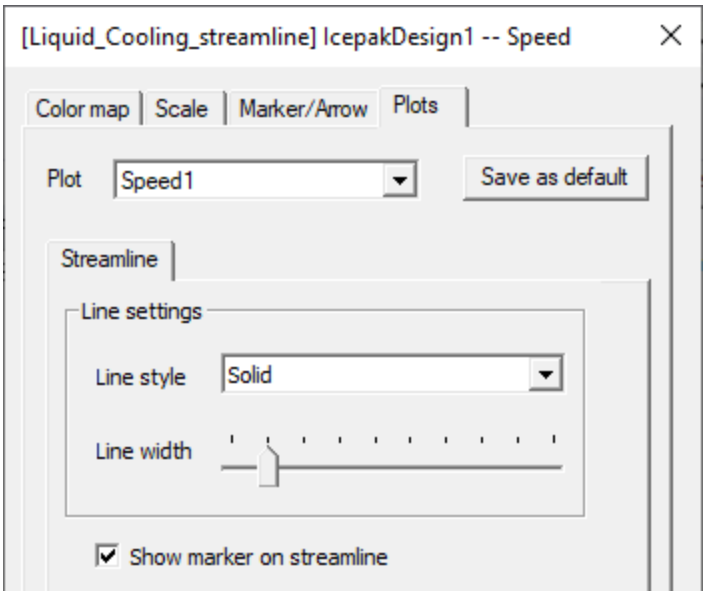
5. In the **Particle Trace Definition** dialog box under **Trace Options**, define the following:
    - **Start time(s)** specifies the time at which the particle traces begin.
    - **End time(s)** specifies when the particle traces should end. The Electronics Desktop estimates an end time from the time scale of the flow such that a particle introduced with the highest velocity occurring in the solution will traverse the length of the enclosure. You can specify a smaller value if you want the particle traces to end before they reach the end of the domain.
    - **Max steps**: specifies the maximum number of time steps at which to plot the particle traces.
    - **Reverse direction** can be used to reverse the direction of the particle traces by tracing their path prior to their introduction at an object face (for example, to display the path of an air stream coming into and exiting the outlet grille).
6. Under **Seeds options**, select **Uniform** or **Nodes**.
    - **Uniform** specifies that particle traces will be uniformly distributed. Specify the total number of particles to be released into the field.
    - **Nodes** specifies that particle traces will be released from each mesh node point on the surface on the selected face. By default, **Skip** is set to 0 indicating that a particle trace is drawn for each face in the domain. If you increase **Skip** to 1, every other particle trace will be displayed. If you increase **Skip** to 2, every third particle trace will be displayed and so on. The order of particle traces will determine which particle traces are skipped or drawn; therefore, adaption and reordering will change the appearance of the particle trace display when a non-zero **Skip** value is used.
7. Click **OK** in the **Particle Trace Definition** dialog box.
8. Click **OK** in the **Create Field Plot** dialog box to display the particle traces.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



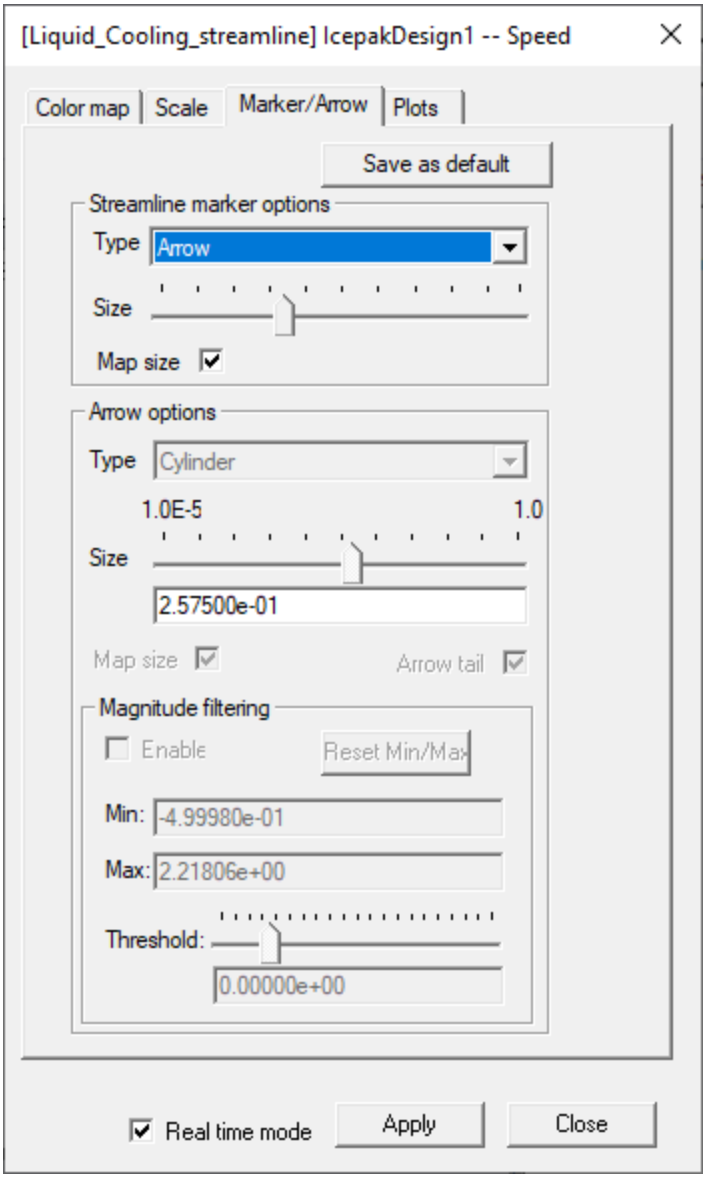
Display Streamline Markers

1. Right-click on the legend and select **Modify**.
2. In the dialog box, select the **Plots** tab and select the **Show marker on streamlines** check box.



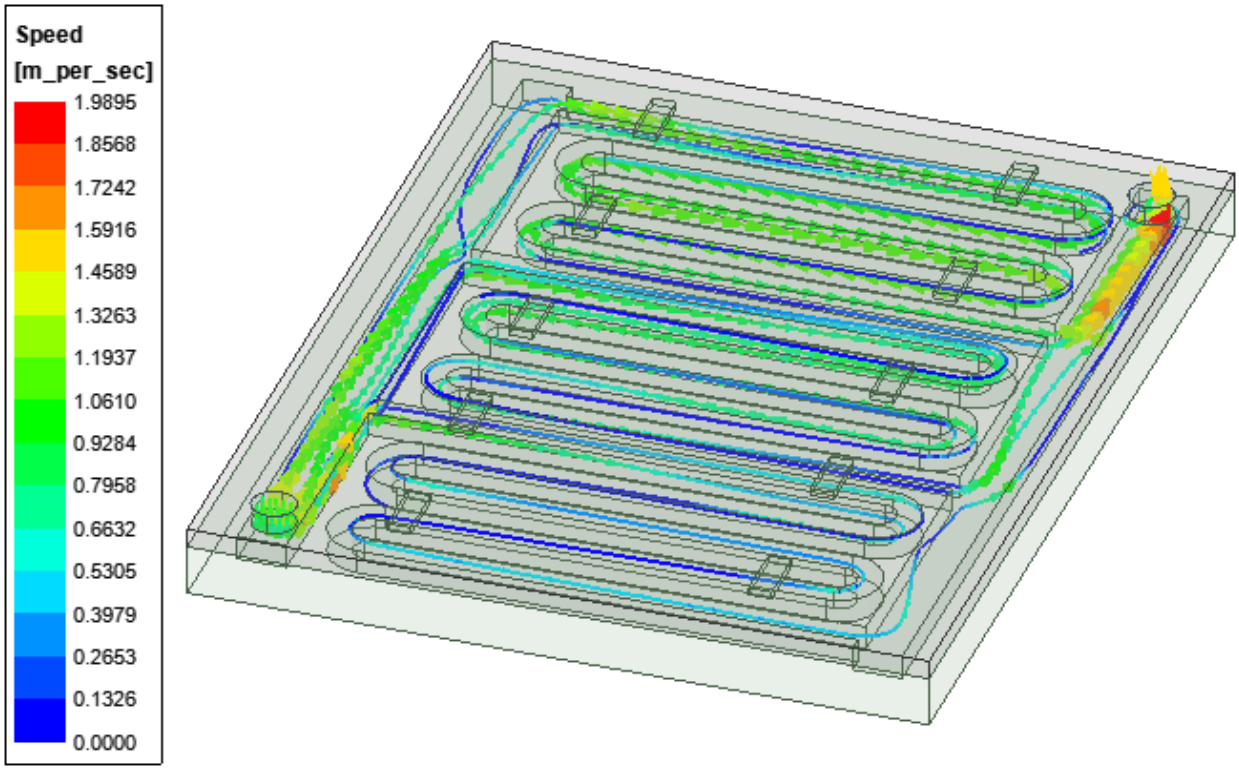
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **Marker/Arrow** tab, under **Streamline marker options**, select a **Type:**
   - **Sphere**
   - **Box**
   - **Tetrahedron**
   - **Octahedron**
   - **Arrow**
4. Adjust the **Size** as needed.



5. Click **Apply** and **Close**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

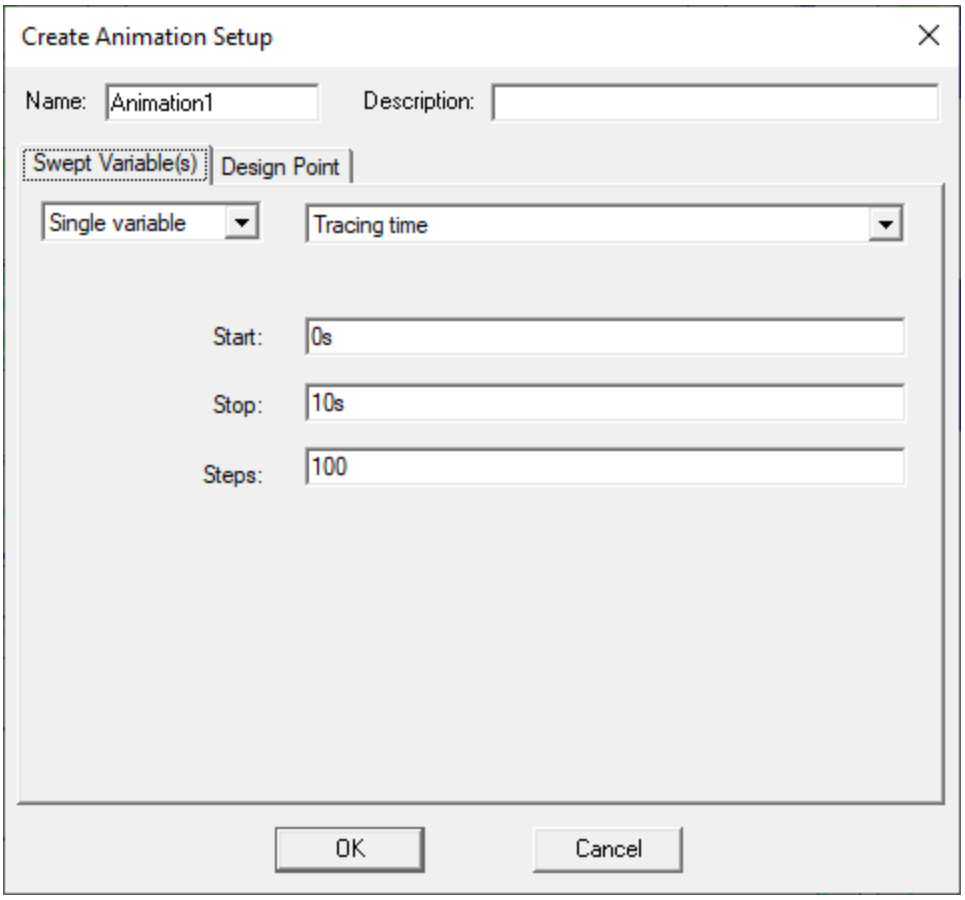


Animating the Streamlines Field Plot

A field plot containing streamlines can be animated to illustrate the particle trace movement throughout the component.

1. In the **Project Manager**, expand **Field Overlays** > **Speed**.
2. Right-click **Speed1** and select **Animate**.
3. In the **Select Animation** dialog box, click **OK**.
4. In the **Animation** dialog box, adjust the **Speed** slider bar to slow the animation.
5. In the **Create Animation Setup** dialog box, enter the **Start** time, **Stop** time, and number of time **Steps** to animate.

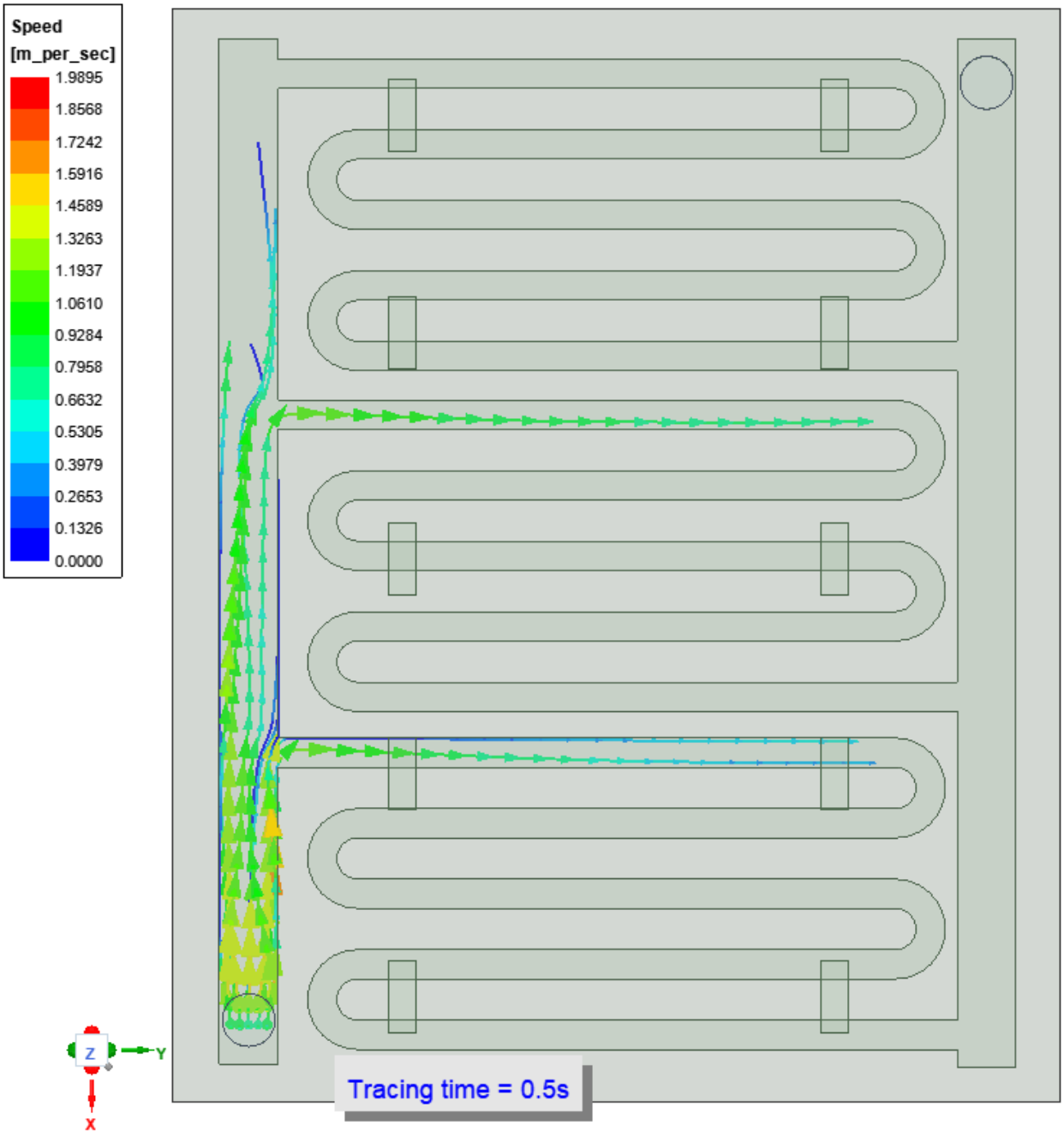
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



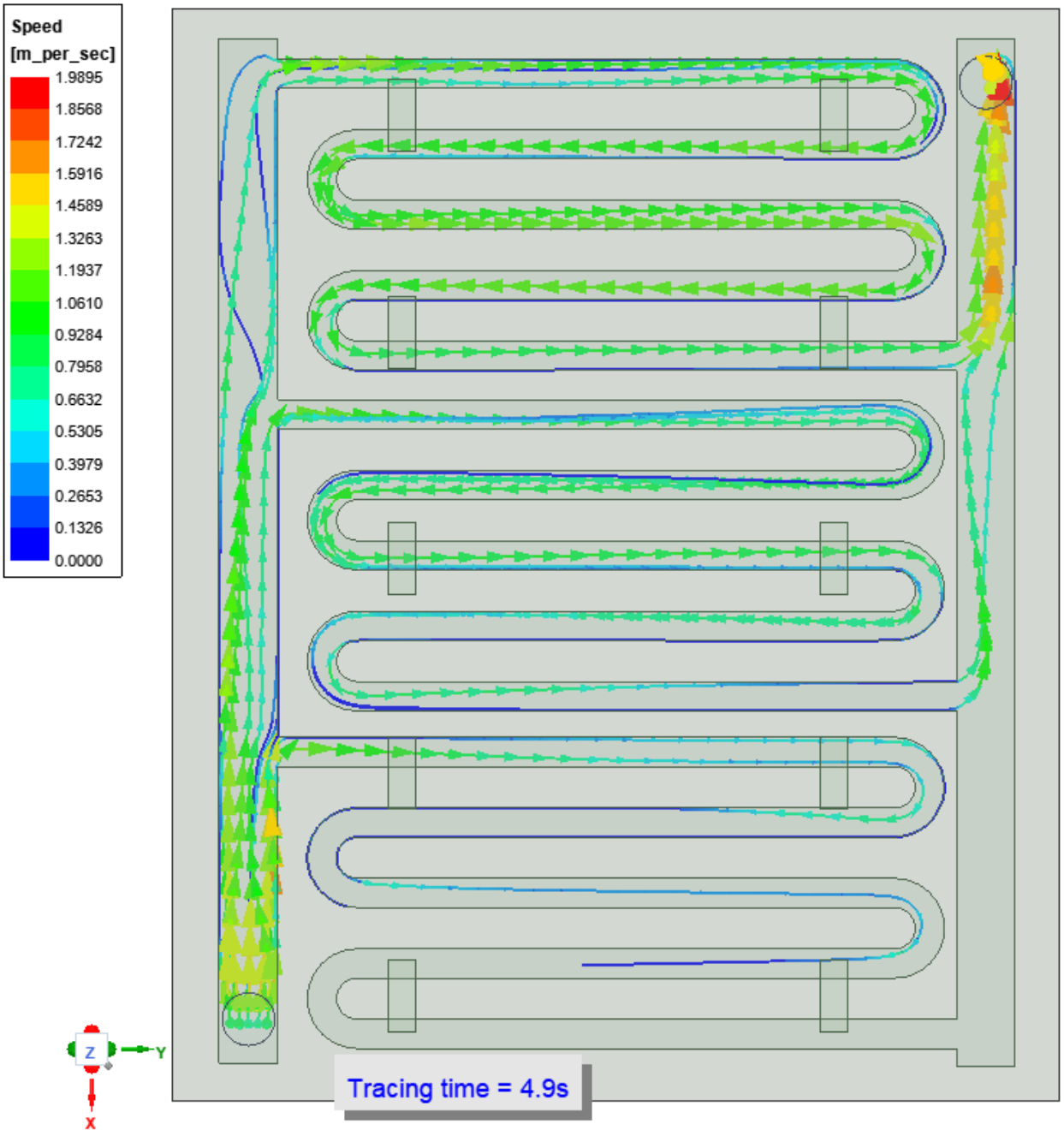
6. Click **OK** to animate the plot.

> **Note:** Use the **Orient** options to view the streamlines plot from the most advantageous angle.

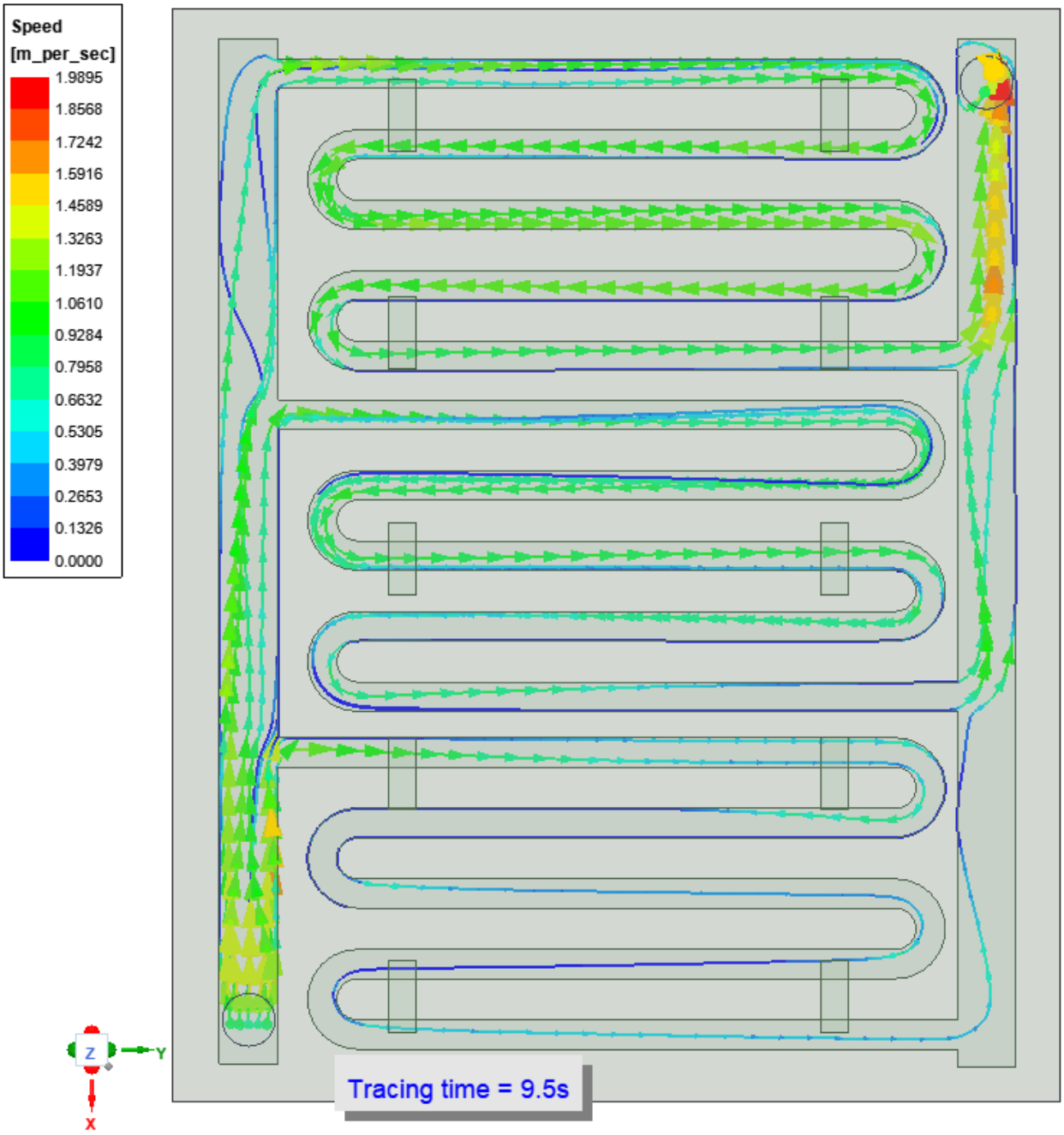
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Icepak Post Processing Quantities

The following sections are categories of post processing quantities available to display Icepak results.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure

- **Pressure** is the relative static pressure of the fluid and is available when flow is computed.
- **SurfPressure** is the relative static surface pressure of the fluid and is available when flow is computed.

Temperature

- **Temperature** is available when temperature is computed.
- **RadTemperature** is the radiation temperature and is available when temperature is computed and you are solving your problem with radiation effects.
- **SurfTemperature** is the surface temperature and is available when temperature is computed. The surface temperature can be computed only on the surface of model objects; it cannot be computed for plane cuts. Surface temperature results are direct solver values and are usually more accurate compared to temperature results which are interpolated nodal values.

Velocity

- **Speed** is the magnitude of the velocity vector and is available when flow is computed.
- **Velocity Vectors** are node-based velocity vectors.
- **Ux**, **Uy**, or **Uz** is the x, y, or z component of the velocity vector and is available when flow is computed.
- **SurfVelocity** is the magnitude of the surface velocity vector and is available when flow is computed.
- **SurfVelocity Vectors** are face center-based velocity vectors.
- **SurfUx**, **SurfUy**, **SurfUz** is the x, y, or z component of the surface velocity vector and is available when flow is computed.

Turbulence

- **ViscosityRatio** is the ratio of the turbulent viscosity to the laminar viscosity. It can be reported if any of the available turbulence models are used.
- **WallYPlus**is a nondimensional parameter defined by the equation:

$$y^+ = \frac{\rho u_\tau y_P}{\mu}$$

where $u_\tau = \sqrt{\tau_w/\rho_w}$ is the friction velocity, y_P is the distance from point P to the wall, ρ is the fluid density, and μ is the fluid viscosity at point P.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **TKE** is the turbulent kinetic energy. It can be reported if the two-equation (standard $k-\varepsilon$) turbulence model, the enhanced two-equation model, or the RNG $k-\varepsilon$ turbulence model is used.
- **Epsilon** is the turbulent dissipation rate. It can be reported if the two-equation (standard $k-\varepsilon$) turbulence model, the enhanced two-equation model, or the RNG $k-\varepsilon$ turbulence model is used.

Thermal Conductivity

- **Kx**, **Ky**, or **Kz** is the x, y, or z component of the orthotropic thermal conductivity vector and is available when traces are imported.
- **Kx_Centroid**, **Ky_Centroid**, or **Kz_Centroid** is the x, y, or z component of the orthotropic thermal conductivity at the centroid of a cell and is available when traces are imported.

Joule Heating

- **ElectricPotential** is available when joule heating is applied to a block boundary condition's bulk conductivity property using a thermal modifier and a contact resistance is specified for a conducting plate boundary condition.
- **CurrentDensity** is current density per unit area.
- **CurrentDensityX**, **Y**, and **Z** are the vector components of the current density per unit area.
- **JouleHeatingDensity** is the joule heating power per unit volume.
- **SurfCurrentDensity** is the surface current density per unit area.
- **SurfCurrentDensityX**, **Y**, and **Z** are the vector components of the surface current density per unit area.
- **SurfJouleHeatingDensity** is the surface joule heating power per unit volume.

EM Mapping

Volume Heat Loss and Surface Heat Loss are imported scalar quantities that are available when EM Loss thermal boundary conditions are defined in the Icepak design.

- **VolumeHeatLoss** is the imported volumetric EM power per unit volume. It is a nodal quantity.
- **SurfaceHeatLoss** is the imported surface EM power per unit area. It is a nodal quantity.

Others

- **HeatFlowRate** is a derived scalar quantity computed from the temperature field. The heat flow can be computed only on the surface of model objects; it cannot be computed for plane cuts or isosurfaces. Two types of heat flow can be computed on an object face:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The heat flux, $\dot{q}_d$, at a point on a surface is computed from

$$\dot{q}_d = -Ak\nabla T \cdot \vec{n}$$

where A is the surface area of the face, k is the conductivity of the material, T is the temperature, and $\vec{n}$ is the normal to the surface.

The heat flow, $\dot{q}_c$, at flow boundaries is computed from

$$\dot{q}_c = A\rho c_p \left(T - T_{ref}\right) \vec{u} \cdot \vec{n}$$

where A is the surface area of the face, ρ is the density of the fluid, c_p is the specific heat, T_{ref} is the default ambient temperature, and $\vec{u}$ is the velocity.

The Heat flow (into Solid) for a solid surface denotes the total heat flowing into the solid object through the surface. It usually represents the conductive component of the heat transfer at the surface. The Heat flow (into Fluid) for a solid surface denotes the total heat flowing into the fluid domain through the surface. When there is no radiative heat transfer in the domain this value represents the convective component of the heat transfer at the surface. When radiation is included the value represents both the convective and radiative components of heat transfer. In such cases the convective component can be determined by subtracting the radiative heat flow value for the surface from the Heat flow (into Fluid) value.

- **ConductionHeatFlow** for a solid surface denotes the total heat flowing into the solid object through the surface. It usually represents the conductive component of the heat transfer at the surface.
- **ConvectiveHeatFlow** for a solid surface denotes the convective heat flowing into the fluid domain through the surface.
- **HeatTransferCoeff** is a derived scalar quantity computed from the temperature field.

  The heat transfer coefficient is computed as

  $$h = \frac{q}{T - T_{ref}}$$

  where q is the heat flux for the surface and T_{ref} is a reference temperature.
- **HeatFlux** is a derived scalar quantity computed from the temperature field. The heat flux can be computed only on the surface of model objects; it cannot be computed for plane cuts. Two types of heat flux can be computed on an object face:

  The heat flux, $\dot{q}_d$, at a point on a surface is computed from

  $$\dot{q}_d = k\nabla T \cdot \vec{n}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where k is the conductivity of the material, T is the temperature, and $\vec{n}$ is the normal to the surface.

The heat flux, $\dot{q}_c$, at flow boundaries is computed from

$$\dot{q}_c = \rho c_p \left(T - T_{ref}\right) \vec{u} \cdot \vec{n}$$

where ρ is the density of the fluid, c_p is the specific heat, T_{ref} is the default ambient temperature, and $\vec{u}$ is the velocity.

- **HeatFlux Vectors** are vectors displaying the direction of heat flow. They are derived from nodal values, so a finer mesh generates more accurate results. The vector magnitude and direction are computed from

$$\dot{q} = \ -k\nabla T$$

where k is the conductivity of the material, T is the temperature.

- **MassFlowRate** is a derived scalar quantity computed from the velocity field. The flow rate can be computed only on the surface of model objects; it cannot be computed for plane cuts. The sign convention for mass flow values at all flow boundaries (internal and external) is shown such that flow going in the direction of the coordinate axes is positive.
- **MassFlux** is the mass flow rate per unit area and can only be computed on the surface of model objects.
- **RadiationFlow** is a derived scalar quantity. It is available if you are solving your problem with radiation effects. It can be computed only on surfaces of objects.
- **RadiationFlux** is the radiation heat flow per unit area and can only be computed on surfaces of objects. It is available if you are solving your problem with radiation effects.
- **VolumeFlowRate** is a derived scalar quantity computed from the velocity field. The flow rate can be computed only on the surface of model objects (typically, a fan, vent, or opening); it cannot be computed for plane cuts or isosurfaces. The sign convention for volume flow values at all flow boundaries (internal and external) is shown such that flow going in the direction of the coordinate axes is positive.
- **FluidDensity** is the density of fluid per unit volume and can only be computed in fluid regions. Fluid density is only available when the density is varying (i.e., when ideal gas model is used or thermal modifier is specified for the density).

Reporting Sides

When reporting on 2D geometry or boundary conditions or the faces of 3D geometry or boundary conditions, which side (outward normal direction) of the geometry or boundary condition reported depends on the data quantity selected and whether or not the **Adjacent side** check box in the **Setup Calculation** dialog box is enabled.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table lists which side of a face is reported for 3D object faces, 2D sheet geometry, and a face selection with one planar face.

Quantities	Adjacent Side	Reported Side
SurfTemperature, SurfVelocity, Surface Ux, Uy, Uz	Disabled	Opposite direction of the face normal.
SurfTemperature, SurfVelocity, Surface Ux, Uy, Uz	Enabled	Direction of the face normal.
SurfPressure, HeatFlowRate, HeatFlux, RadiationFlux, Heat TransferCoeff	Disabled	Direction of the face normal.
SurfPressure, HeatFlowRate, HeatFlux, RadiationFlux, Heat TransferCoeff	Enabled	Opposite direction of the face normal.

> **Note:**
>
> If a face selection contains more than one face or only one non-planar face, normal is not applicable and displayed as **N/A**.

Plotting Derived Field Quantities

Derived field quantities are field quantity representations that have been deduced from the original field solution using the Fields Calculator.

1. Select a point, line, surface, or object to create the plot on or within.
2. On the **Icepak** menu, or right-click on the **Field Overlays** icon in the Project tree, and point to **Fields>NamedExpression**.
3. Select the derived quantity you want to plot, and then click **OK**.

The **Create Field Plot** dialog box appears.

4. To specify a name for the plot other than the default, select **SpecifyName**, and then type a new name in the **Name** text box.
5. Select the solution to plot from the **Solution** drop-down menu.
6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** drop-down menu, or type the name you wish to use. Plot folders are listed under **Field Overlays** in the project tree.
7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. Select the derived field quantity to plot from the **Quantity** list.
9. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This selection enables you to limit plots to the intersection of a volume and the selected object.

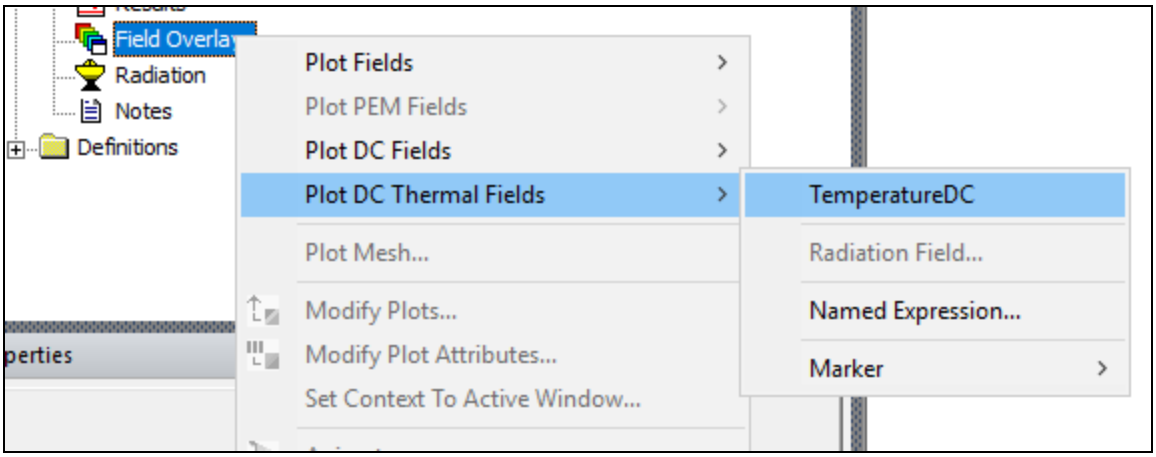
10. Click **Done**.

    The derived field quantity you created in the Fields Calculator is plotted on the surfaces or objects you selected. The new plot is listed in the project tree under **Field Overlays**.

Overlaying DC Temperature Fields (General Mode Only)

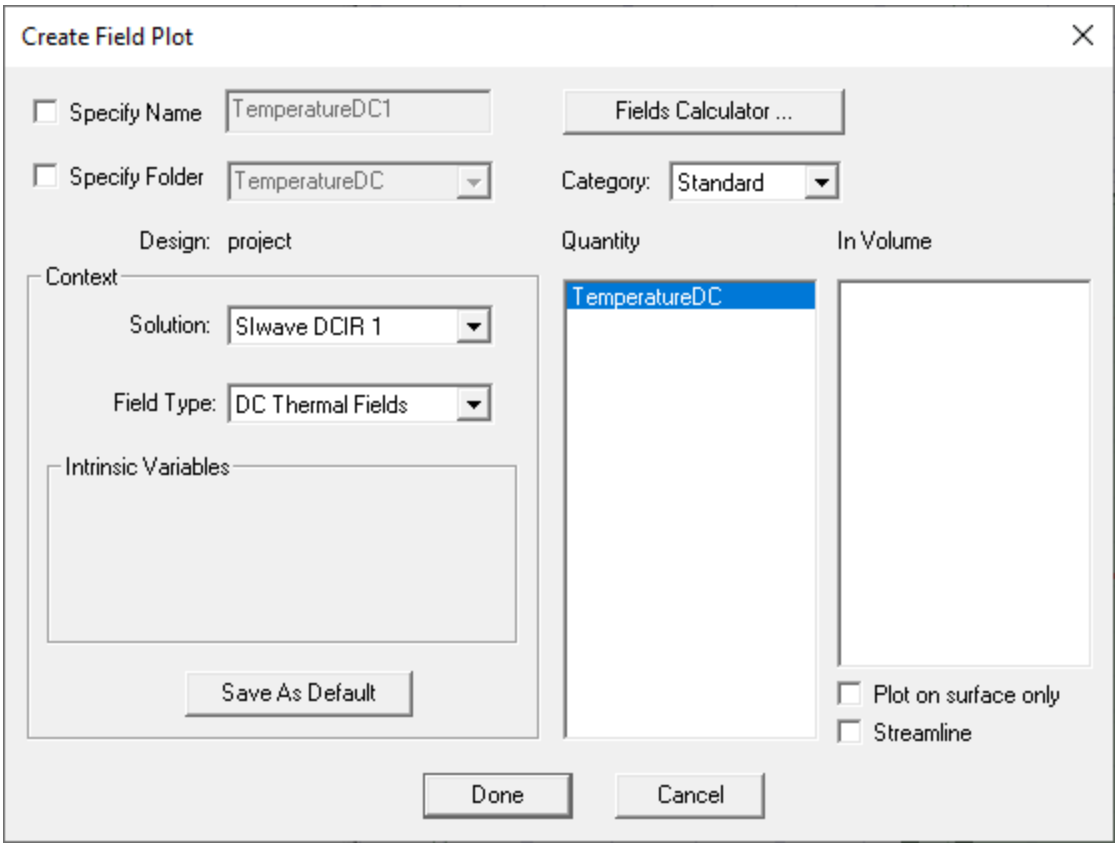
Projects with two-way coupling between Icepak and HFSS 3D Layout can have a field overlay that displays the DC temperatures. The 3D Layout project must have an Icepak DC IR analysis. To create such an overlay:

1. In the Project Manager, open the 3D Layout design and right click **Field Overlays** > **Plot DC Thermal Fields** > **TemperatureDC**.



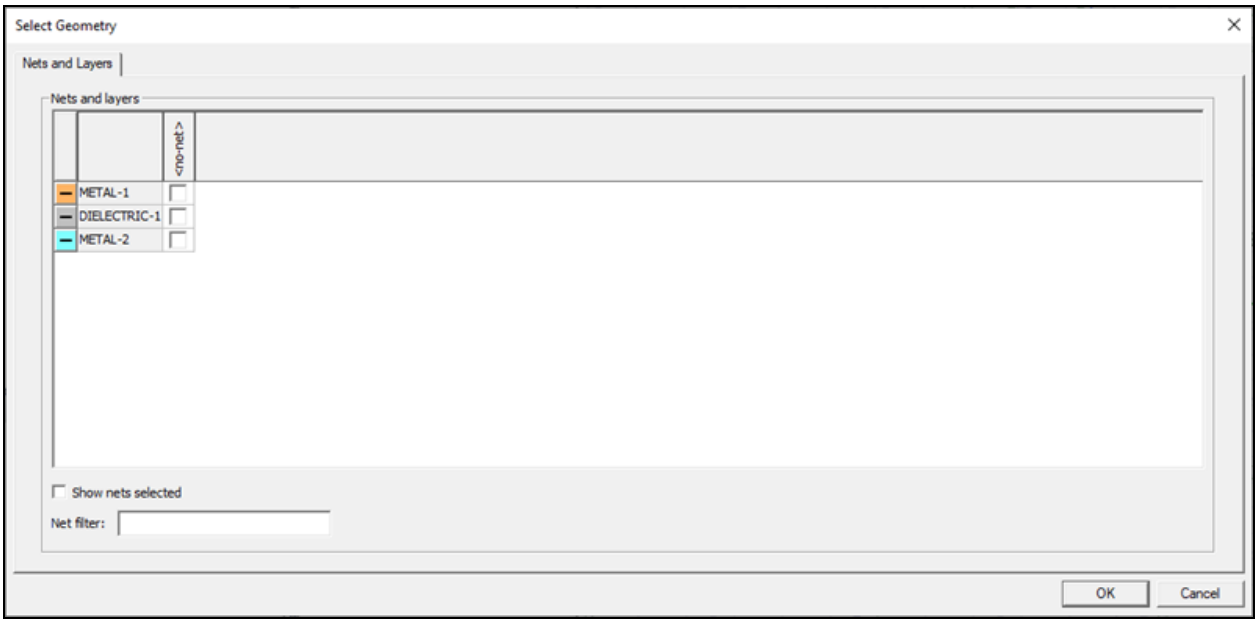
    The **Create Field Plot** window opens.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



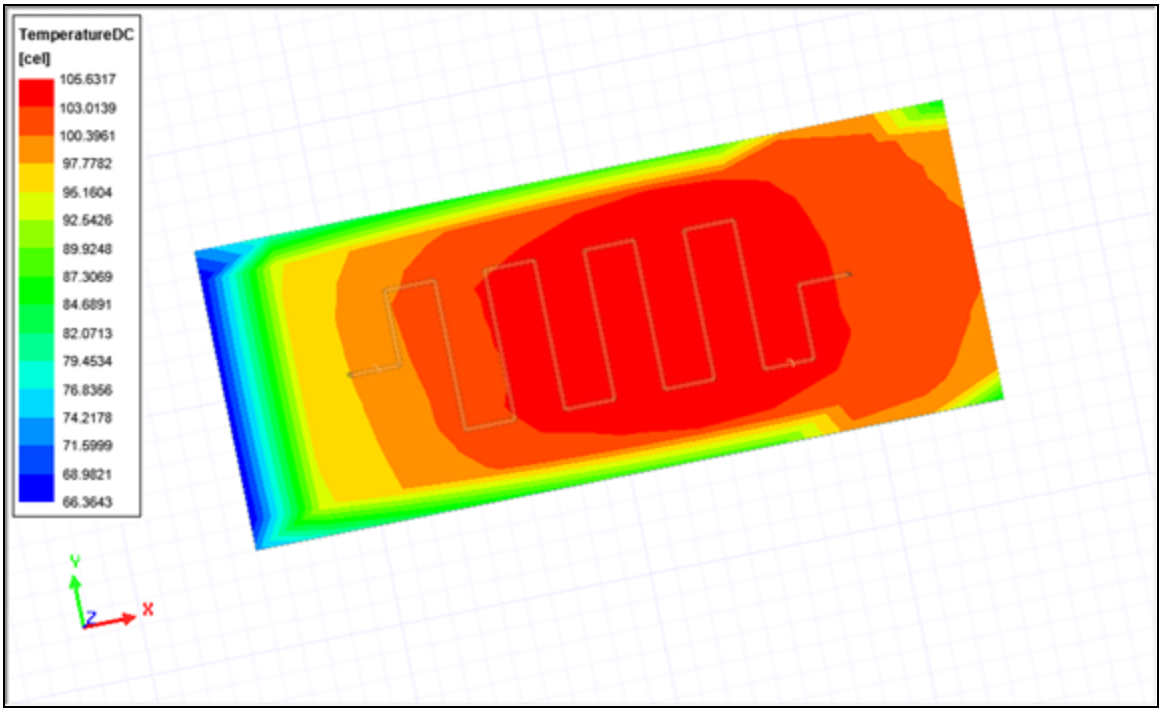
TemperatureDC is automatically selected from the Quantity list.

2. Click **Done**. The **Create Field Plot** window closes, while the **Select Geometry** window opens to its **Nets and Layers** tab.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the layer whose temperature field should display.
4. Click **OK** and the overlay appears.



Creating 2D Reports from Named Expressions

You can create a 2D report from named expressions 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 **Icepak**, 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 **Icepak** 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.
8. Add one or more traces to include in the report.
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.

Creating Scalar Field Plots

A scalar plot uses shaded colors or contoured lines to illustrate the magnitude of field quantities on surfaces or volumes.

1. Do one of the following:
   a. To create a scalar surface plot, select the faces on which you want to plot the fields.
   b. To create a scalar volume plot, select the objects within which you want to plot the fields.
2. Click **Icepak** > **Fields** > **Plot Fields**.
3. On the **Plot Fields** menu, click the scalar field quantity you want to plot.

   The **Create Field Plot** dialog box appears.

4. Follow the procedure for plotting field overlays.

The plot uses the attributes specified in the **Plot Attributes** dialog box.

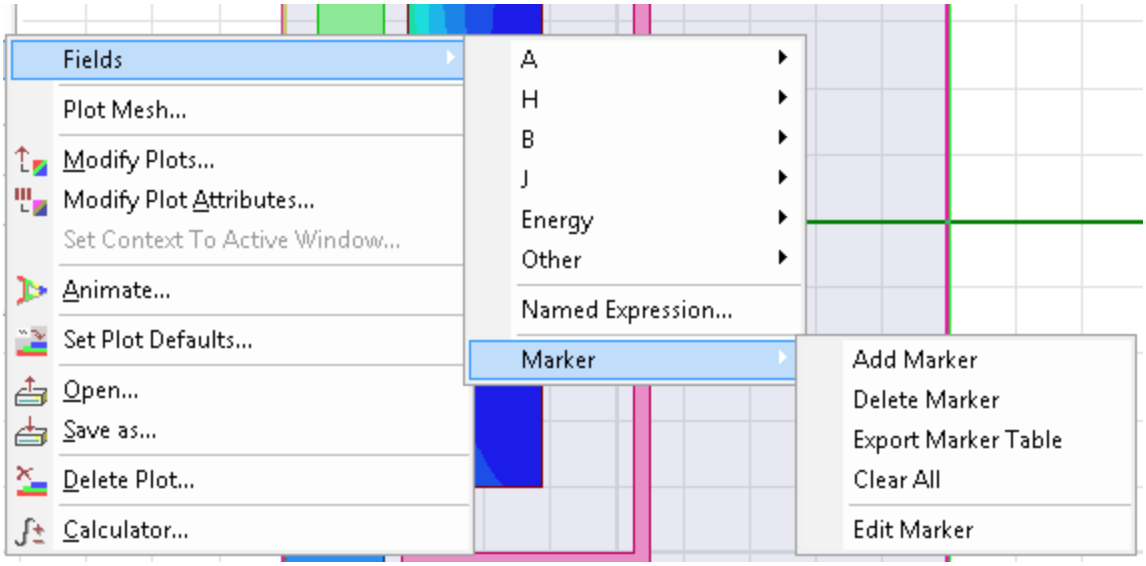
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new plot will be listed in the specified plot folder in the project tree.

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

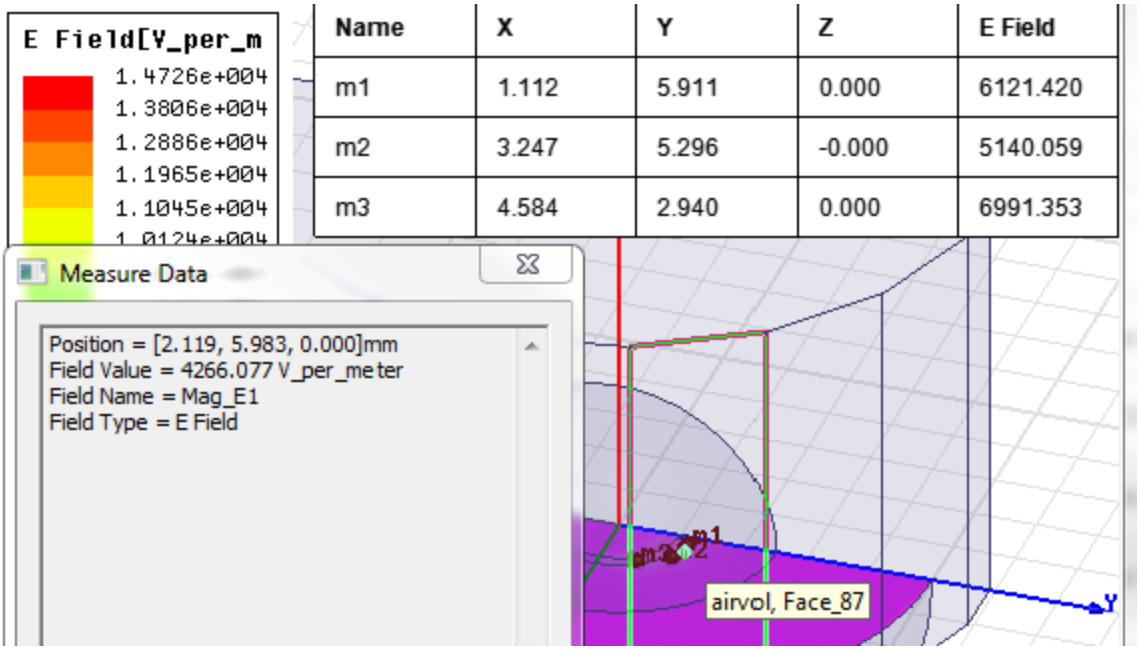
To add one or more field plot markers to a scalar field:

1. On the main menu click **Icepak** 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**.

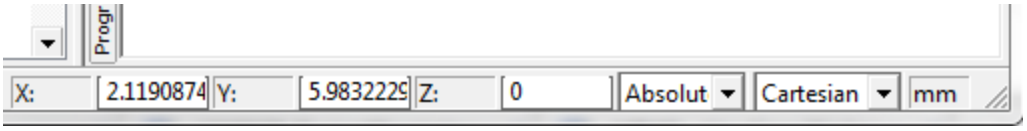
   The *Measure Data* dialog box opens, and a round dot appears at the tip of the cursor.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

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 **Icepak** and then select **Fields** > **Plot Fields** > **Marker** > **Delete Marker** to delete the selected marker(s).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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 **Icepak** 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:

- On the main menu, click **Icepak** 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.

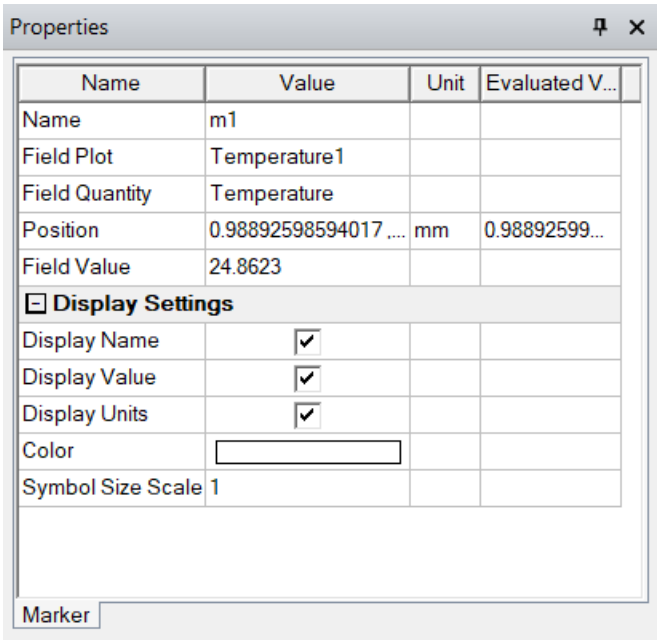
Editing Field Plot Markers

To edit field plot markers:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

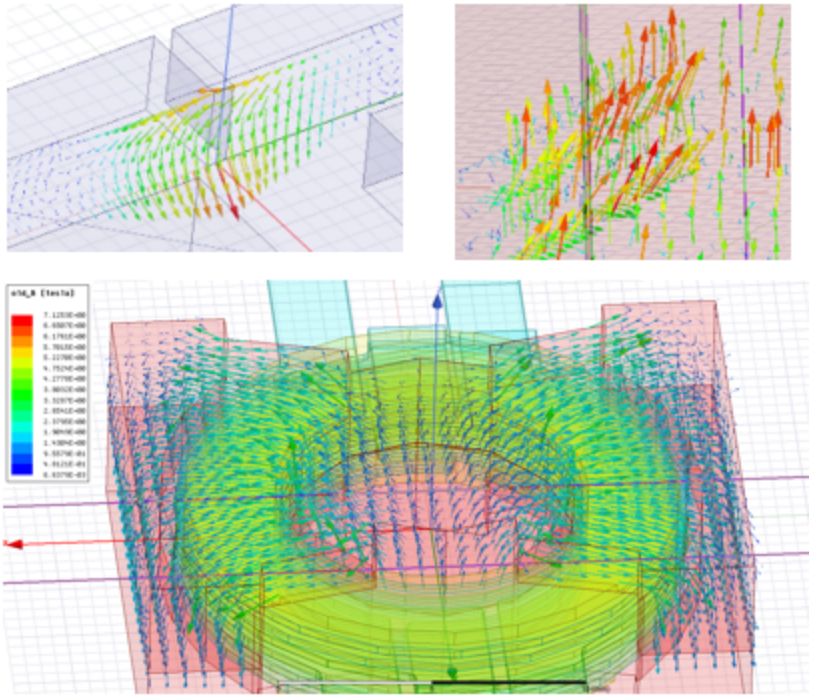
Showing Field Plot Markers

To display field plot markers in the 3D Modeler window, on the main menu click **Icepak** and then select **Fields** > **Plot Fields** > **Marker** > **Show Marker**. The table showing the marker coordinates and associated field value is also displayed in the modeler window.

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** > **Show Marker**.

Creating Vector Field Plots

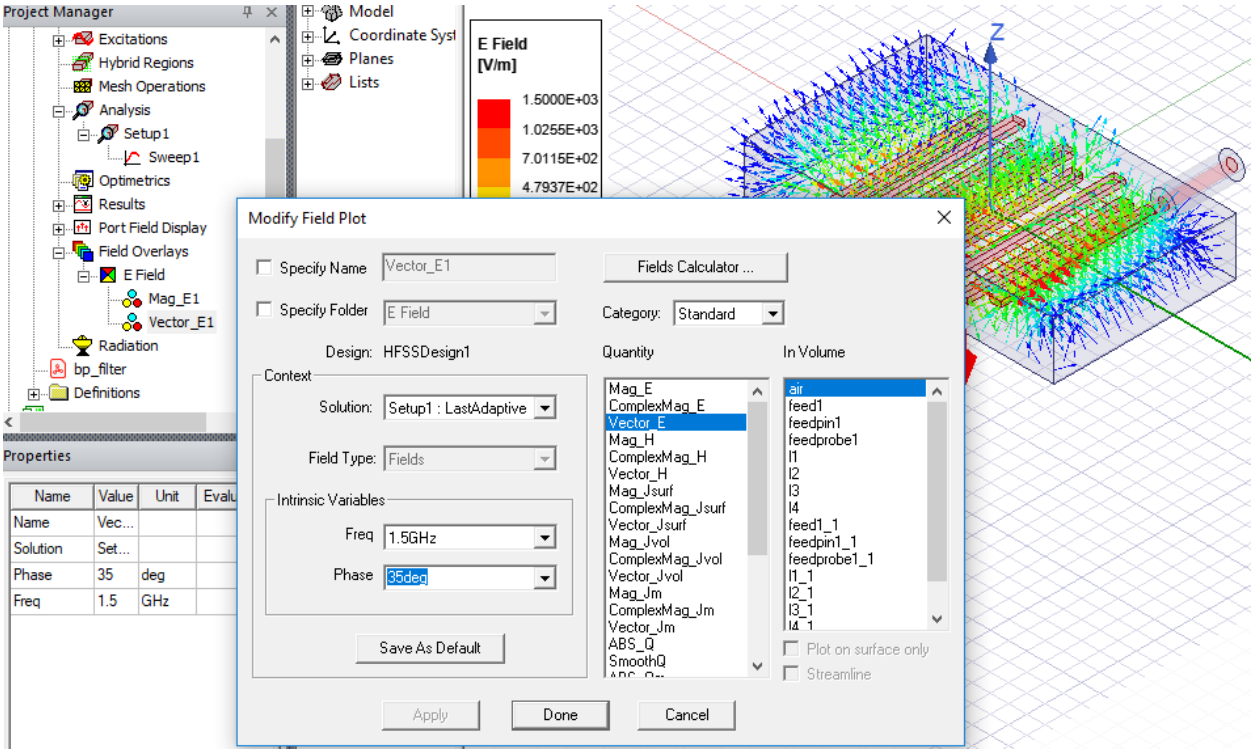
A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities. Vector plots can be created on surfaces or volumes. The vector plots calculates a good initial spacing (min, max, and spacing range). The initial min, max spacing is based on a percentage of the plot geometry size and they are in model units. You can edit the vector min and max threshold spacing and the arrow representation.



1. Do one of the following:
    a. To create a vector surface plot, select the faces on which you want to plot the fields.
    b. To create a vector volume plot, select the objects within which you want to plot the fields.
2. Click **Icepak** > **Fields** > **Fields**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

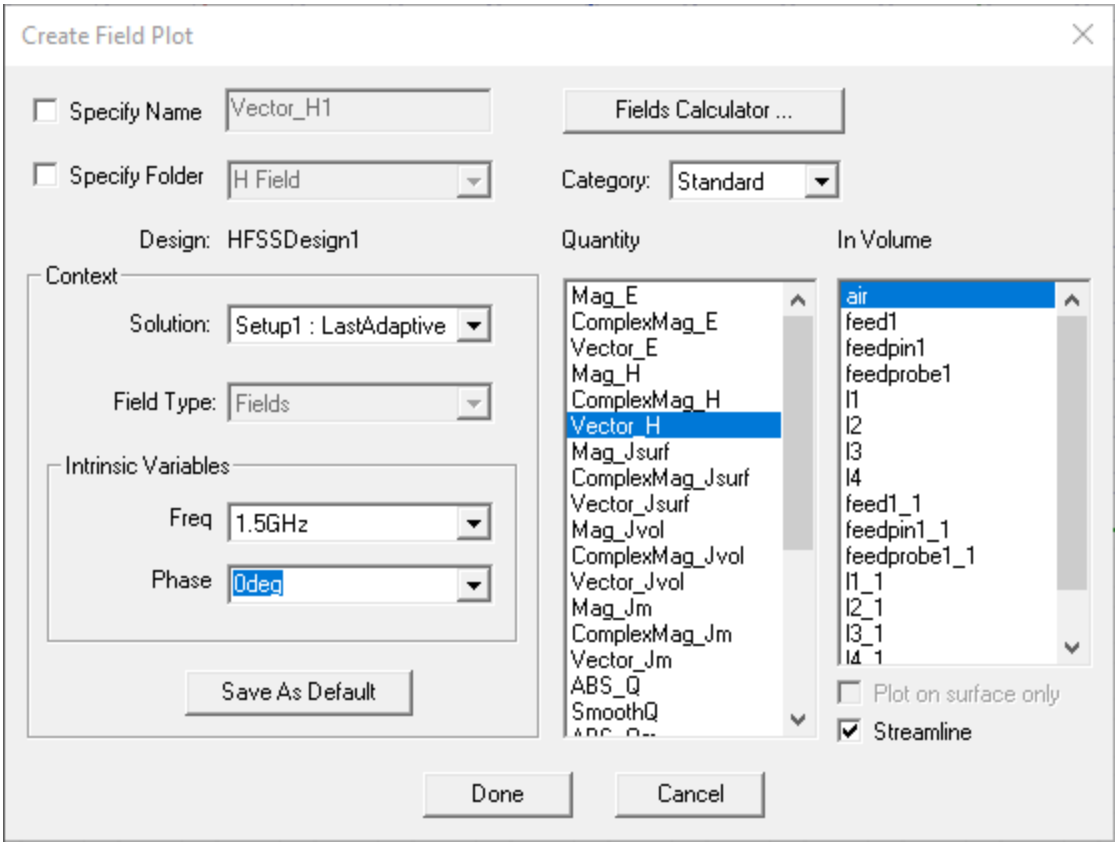
3. From the **Fields** submenu, click the vector field quantity you want to plot.



4. Follow the procedure for plotting field overlays.

If you select a vector quantity when creating a plot, you can also check Streamline for the plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

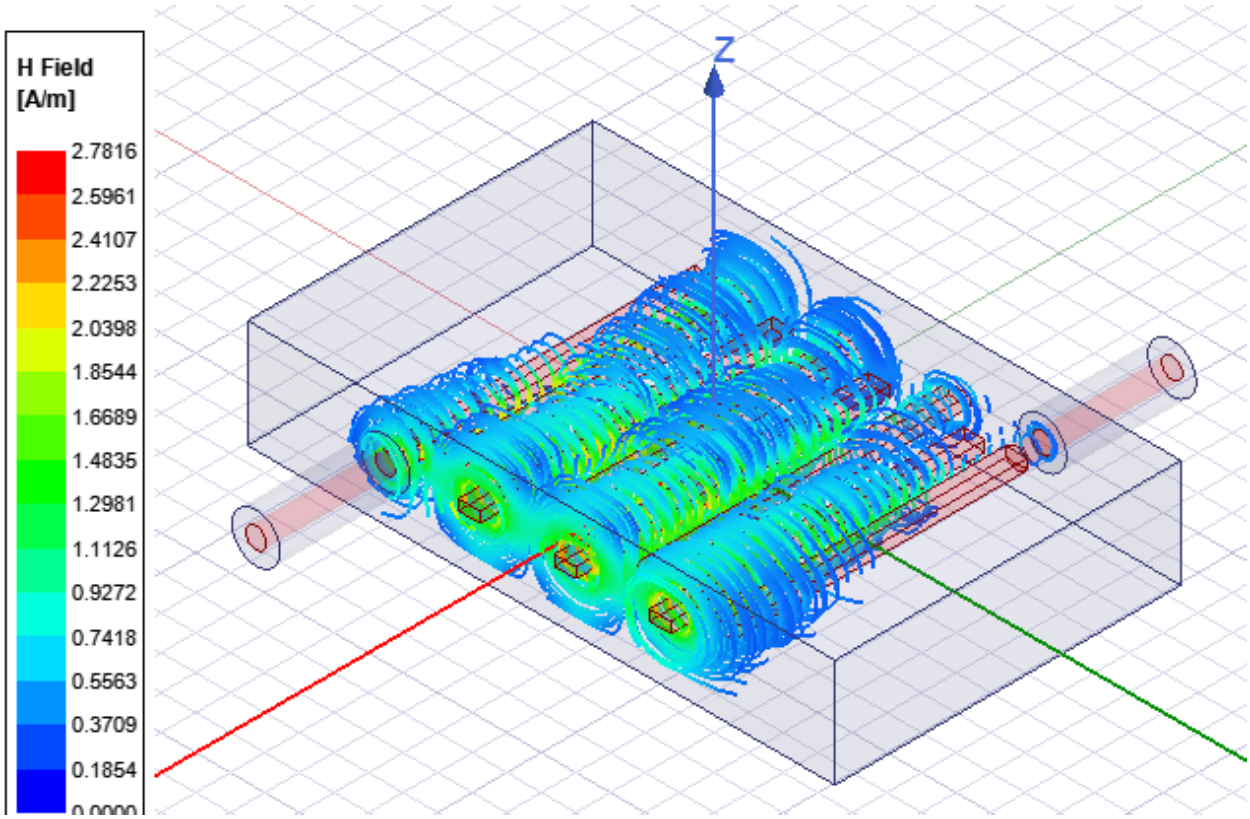


Notes for creating Streamline plots:

1. Before creating the plot, select both starting and ending edges (in 2D) or surfaces (in 3D).
2. In the Creating Field Plot menu, select "In Volume: Region" which is the volume in which the streamlines will appear and is outside of the sources.
3. After the plot is created, on the Attributes/Plot tab reduce "Seeds density" to show more streamlines. If no streamlines appear, reduce this by a factor of 10 (or 100) because the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default seeding was too large.



Modifying Field Plots

1. Click **Icepak**> **Fields**> **Modify Plots** , or in the Project Manager, select the **Field Overlays** icon, right-click, and select **Modify Plots**, or press the **M** hotkey.

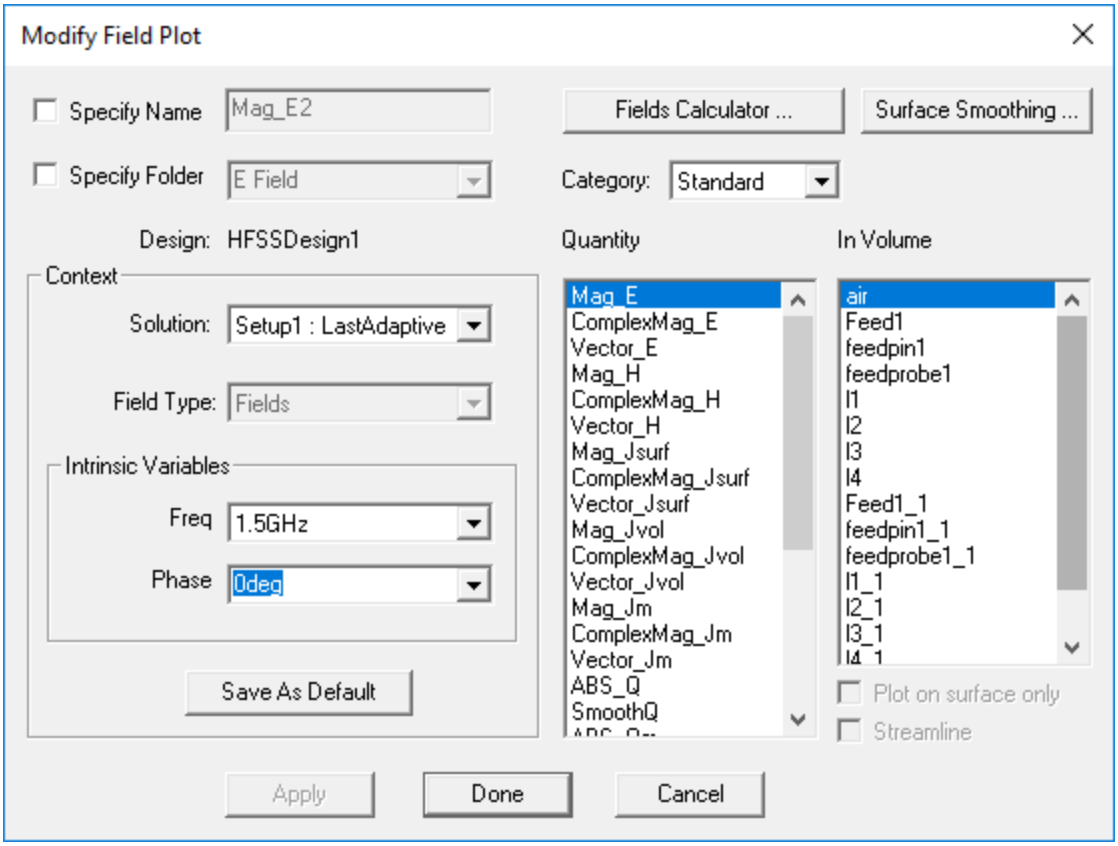
   The **Select Field Plot(s)** dialog box appears.

   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.

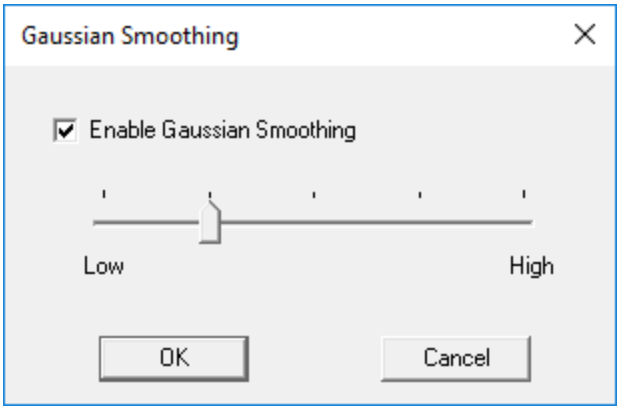
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 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.

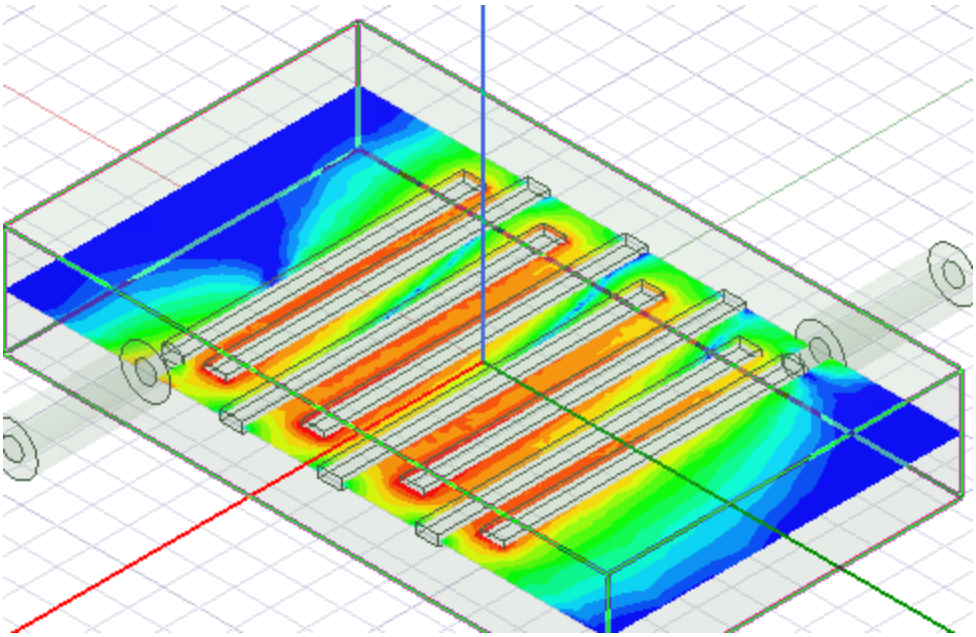
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the check box to enable the feature and use the slider to set the degree of smoothing.

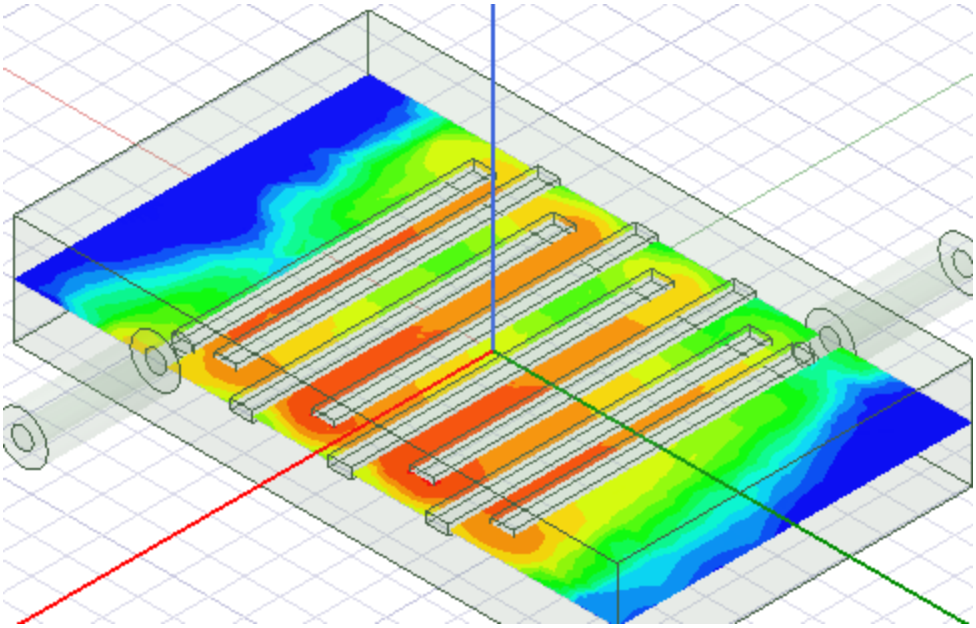
Important: You *must* click **Apply** in the *Modify Field Plots* dialog box to see the changes.

For example, this is a Mag_E fringe plot before applying the smoothing:



This is a Mag_E fringe plot after applying High smoothing:

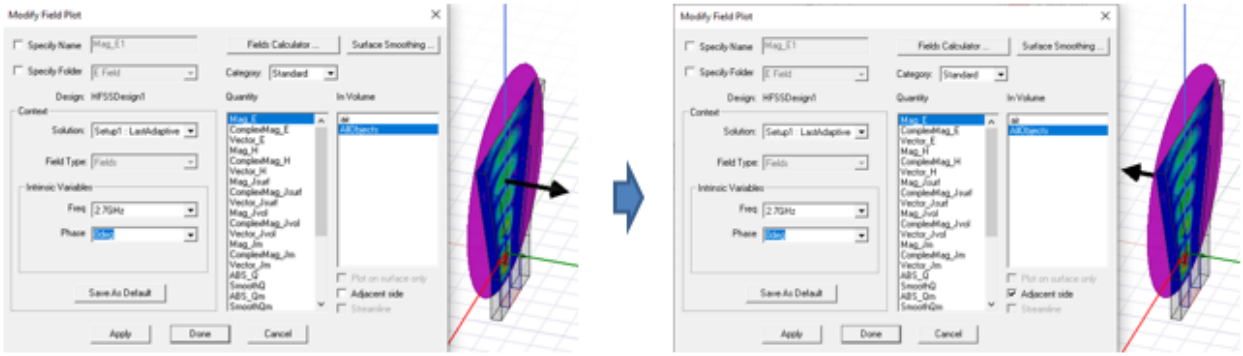
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

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.

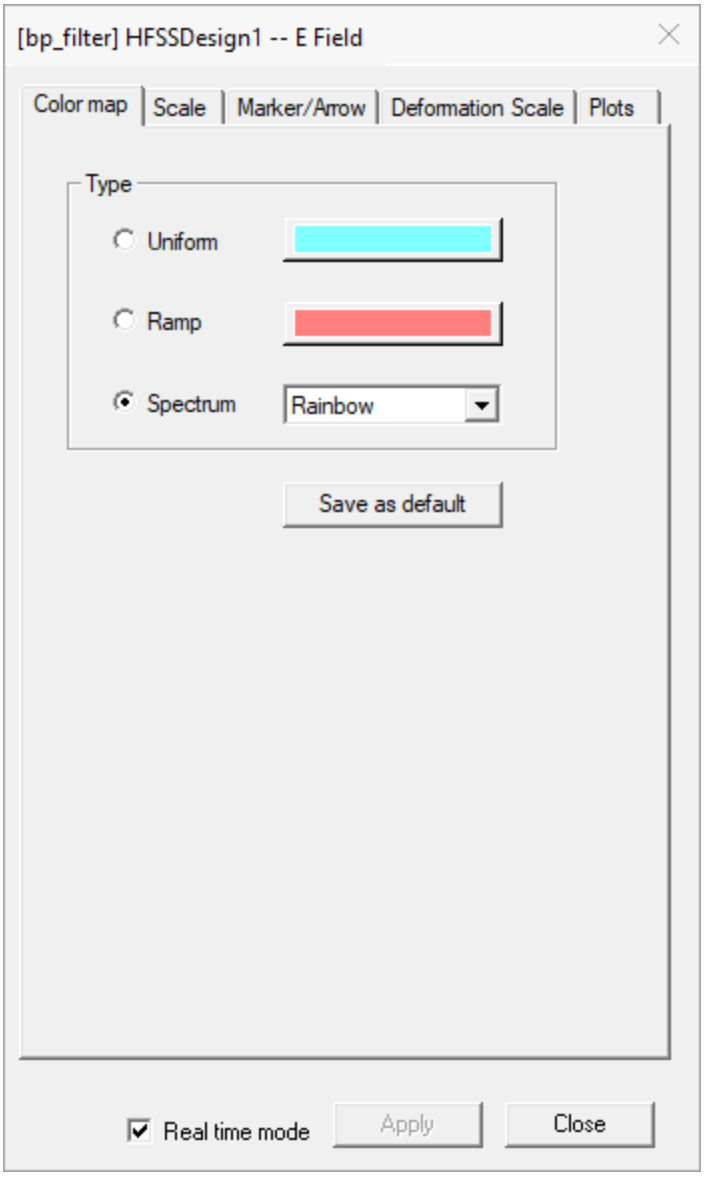
1. Click **Icepak** > **Fields** > **Modify Plot Attributes**. Alternatively, right-click **Field Overlays** in the Project Manager and select **Modify Attributes** from the shortcut menu.

   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 2026 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:**

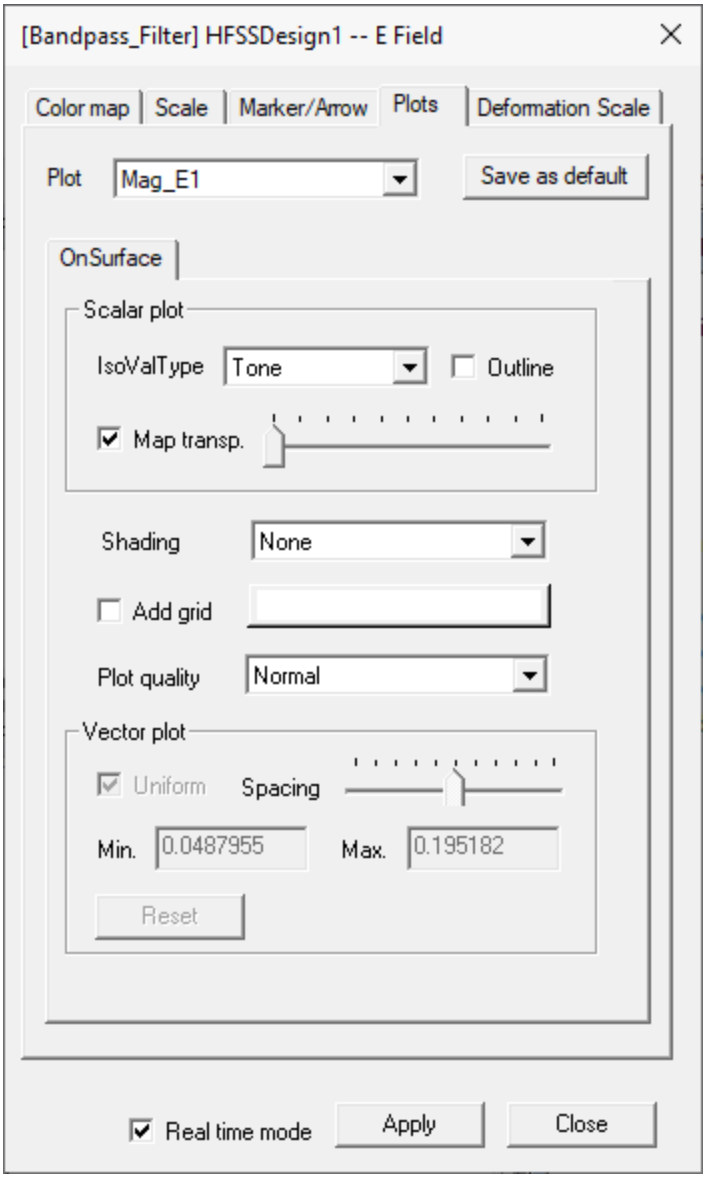
     The scale of field quantities, including the number of divisions in the scale, whether to use dB as the units, whether to use a linear or log scale, auto scale options, and plot number format.

Ansys Electromagnetics Suite 2026 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:**

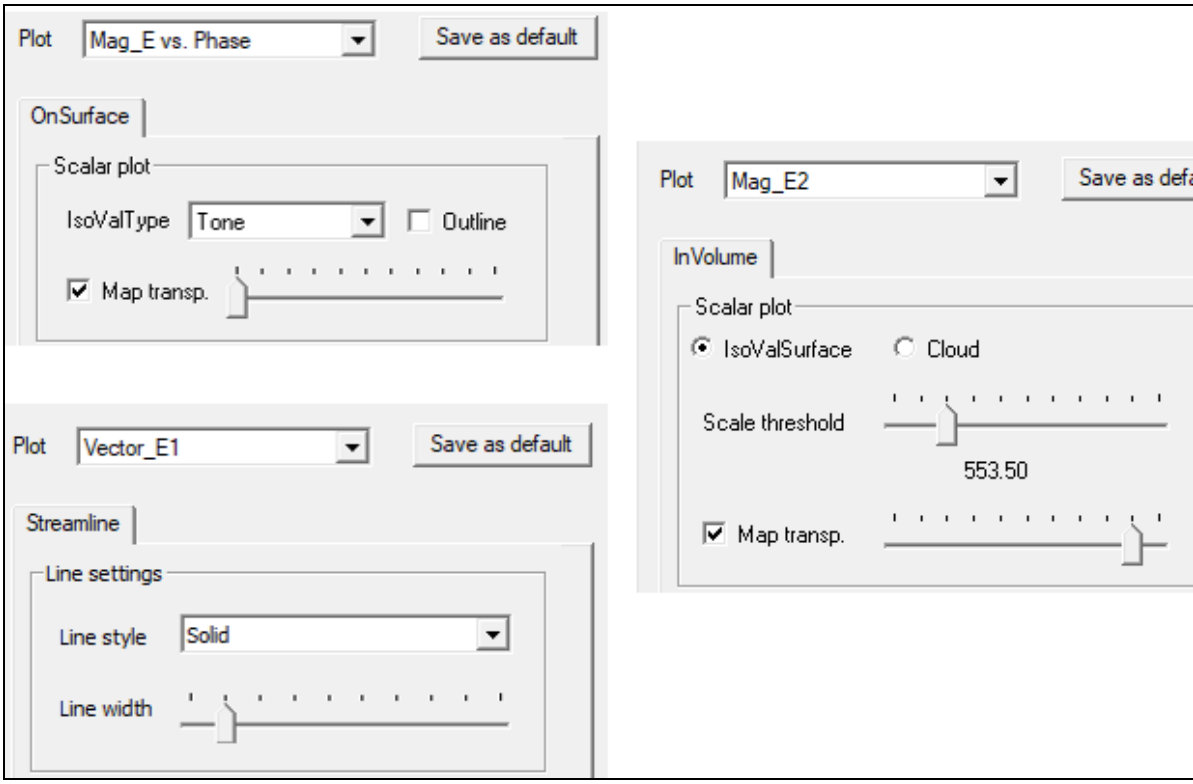
  This is for use with plots that include Stress feedback from Ansys Workbench Integration.
- **Plots (if not vector or streamline):**
  - The **Plot** selected. If multiple plots are available, you can select from a drop down menu.

Ansys Electromagnetics Suite 2026 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** or **Streamline**. The type of plot you select determines the control options shown.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



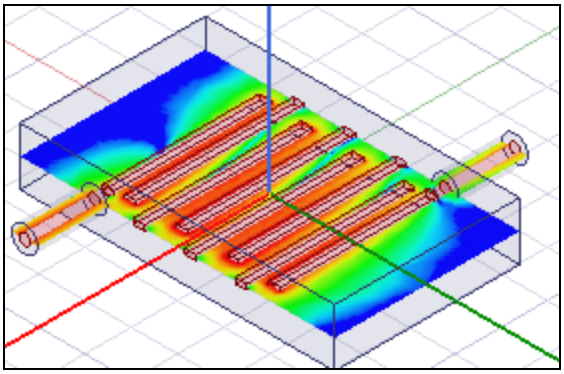
- For scalar surface plots you can select the type of isovalue display. For *Line, Fringe* and *Tone* **IsoValType**, *Outline* is enabled. For *Gourad*, it is disabled.
- Whether to use Shading (for scalar plots), if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting." Shading can be Flat or Smooth.
- **None** Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.

  Flat Provides a flat rendering of the field plot.

  Smooth Provides a smooth rendering of the field plot.
- 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**.

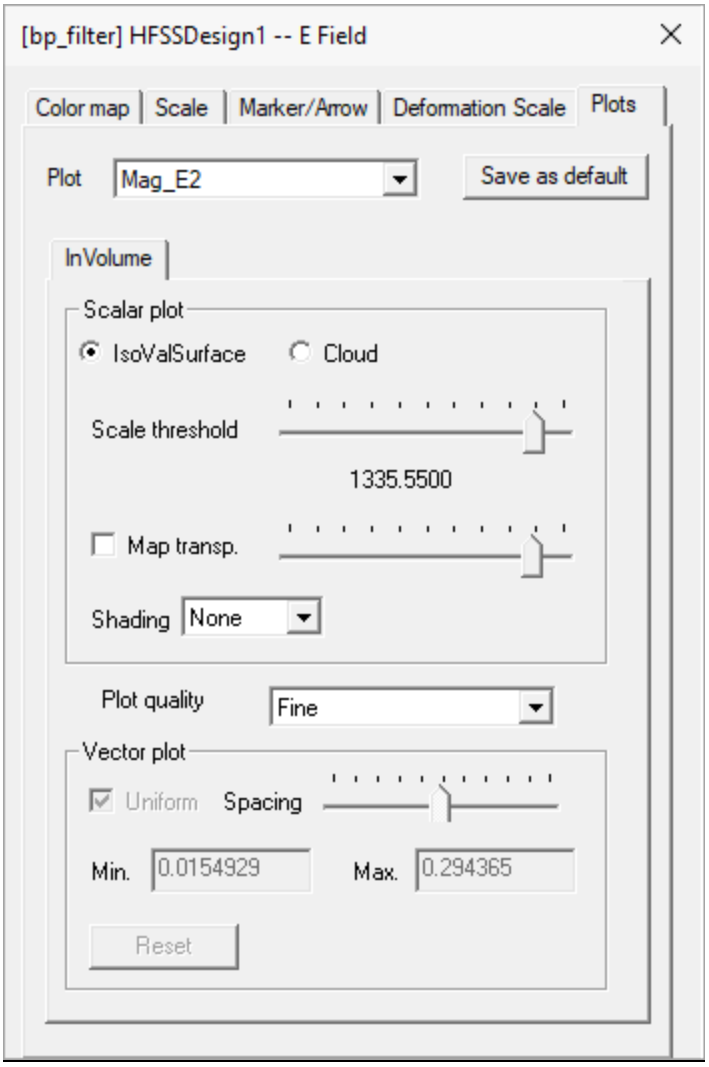
  This affects the use of memory for animating plots. For large plots with more frames to animate, use **Coarse** or **Normal** to reduce memory requirements and improve performance. For smaller plots with few frames, if higher resolution is required, use **Fine** or **Very Fine**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



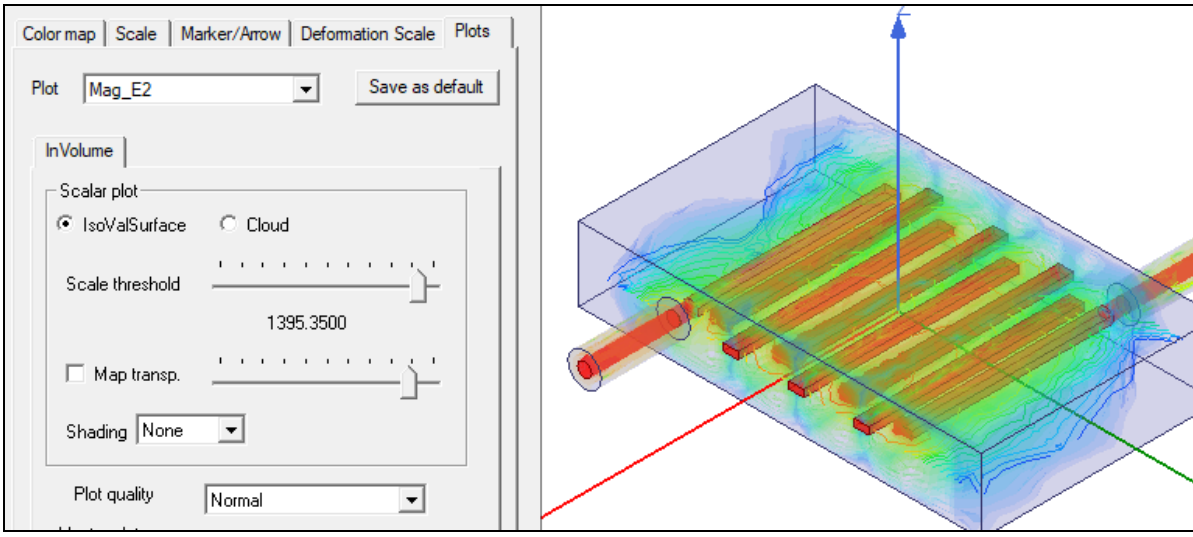
If you select an InVolume plot, display options are available to allow visibility of internal layers and objects, particularly those with high field values. For *IsoValSurface* for an *InVolume* Scalar plot, the choices are for **Scale threshold** and **Map transp**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

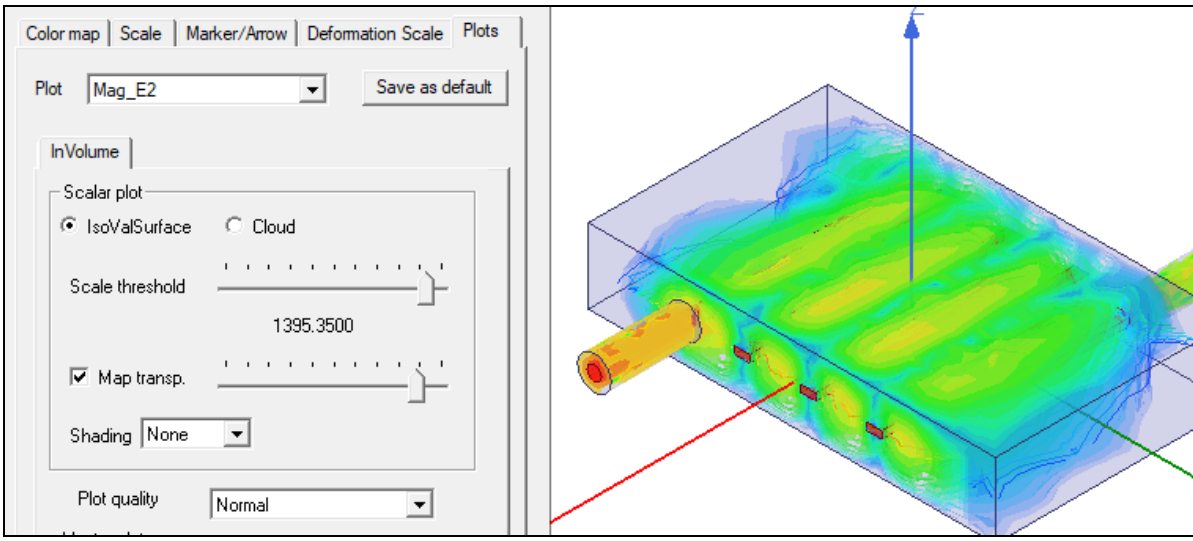


- The “**Scale threshold**” slider bar allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



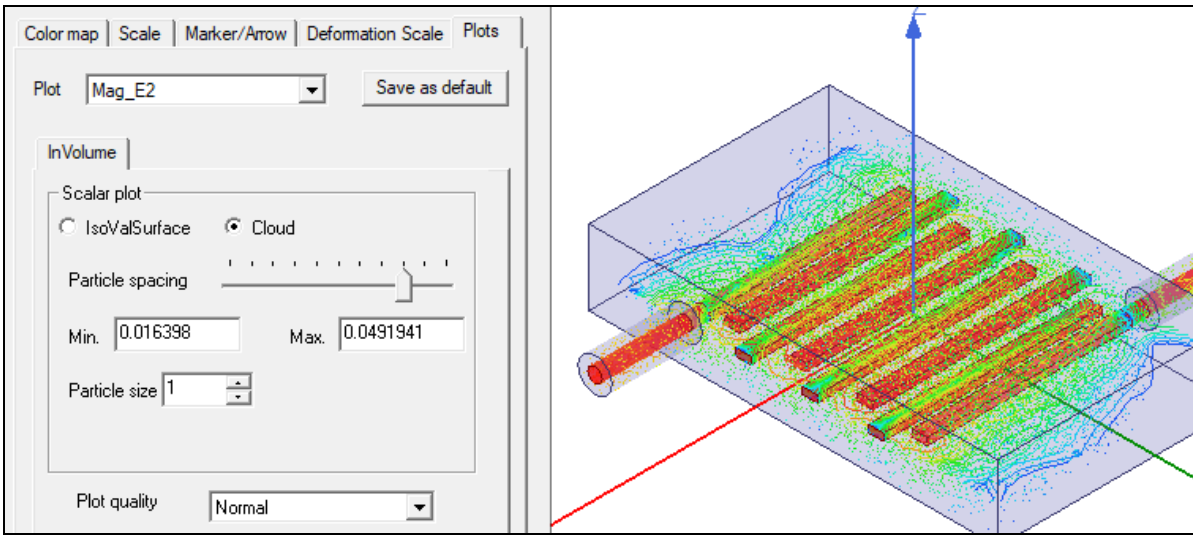
- Iso surfaces with field value higher than the threshold will show as opaque. The range of the "**Scale threshold**" is the same as the color key range from *Min* to *Max*.
- When "**Map transp.**" is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map 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 2026 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.

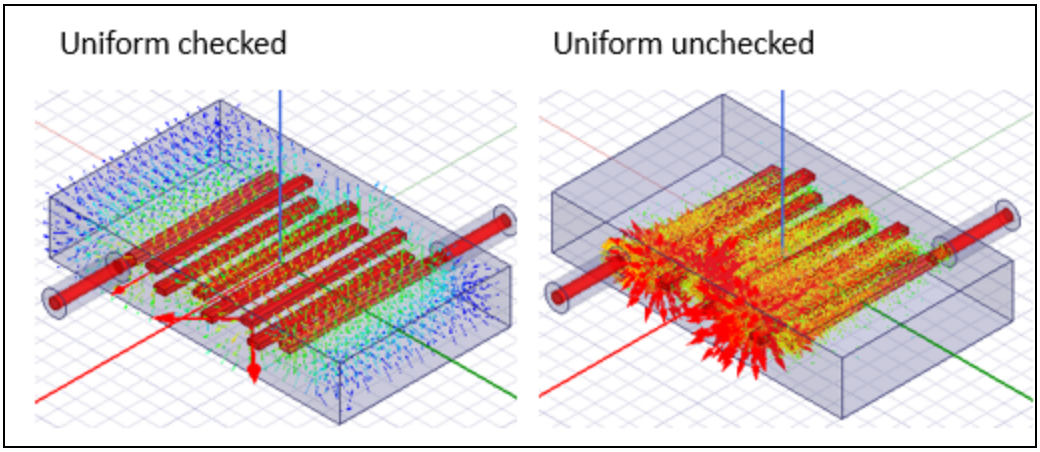
- For *Cloud* plots, field values are represented by points that illustrate the spatial distribution of the solution. You can use the **Particle spacing** slider bar to adjust the plot. The higher the solution value, the greater the cloud density. You can also specify **Min** and **Max** values (numeric 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

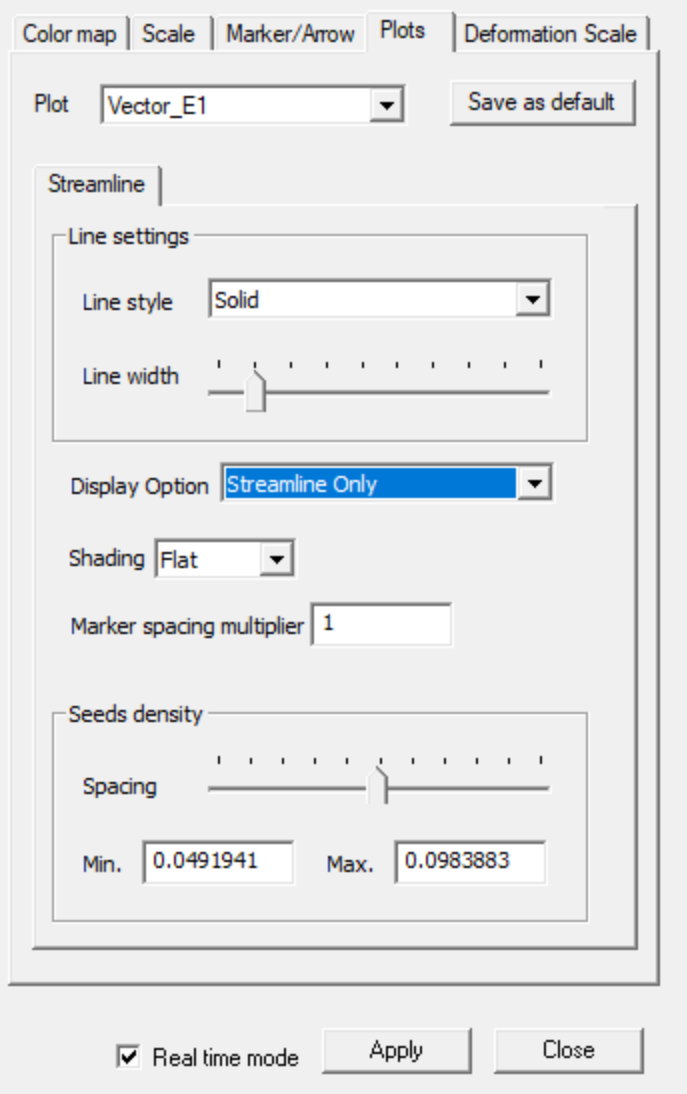


The **Reset** button resets a good initial spacing (and thus Min, Max spacing range) without the need to recreate the field plot.

- **Streamline Plots (if streamline is checked when you create the plot):**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Plot** selected.



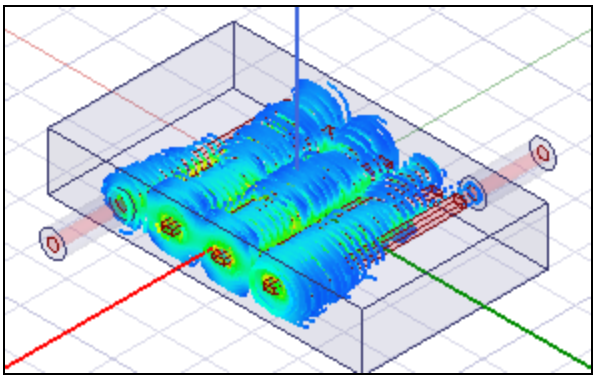
- The **Line style** as *solid* or *cylinder* from drop-down menu.
- **Line width**, specified using a slider.
- Display Option for whether to show Streamline Only, *Marker Only* or *Streamline and Marker*.
- Whether to use Shading, if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting." Shading can be Flat or Smooth.
- **None** Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.

  Flat Provides a flat rendering of the field plot.

  Smooth Provides a smooth rendering of the field plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **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).



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 Field Plot Colors

1. Click **Icepak** > **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).

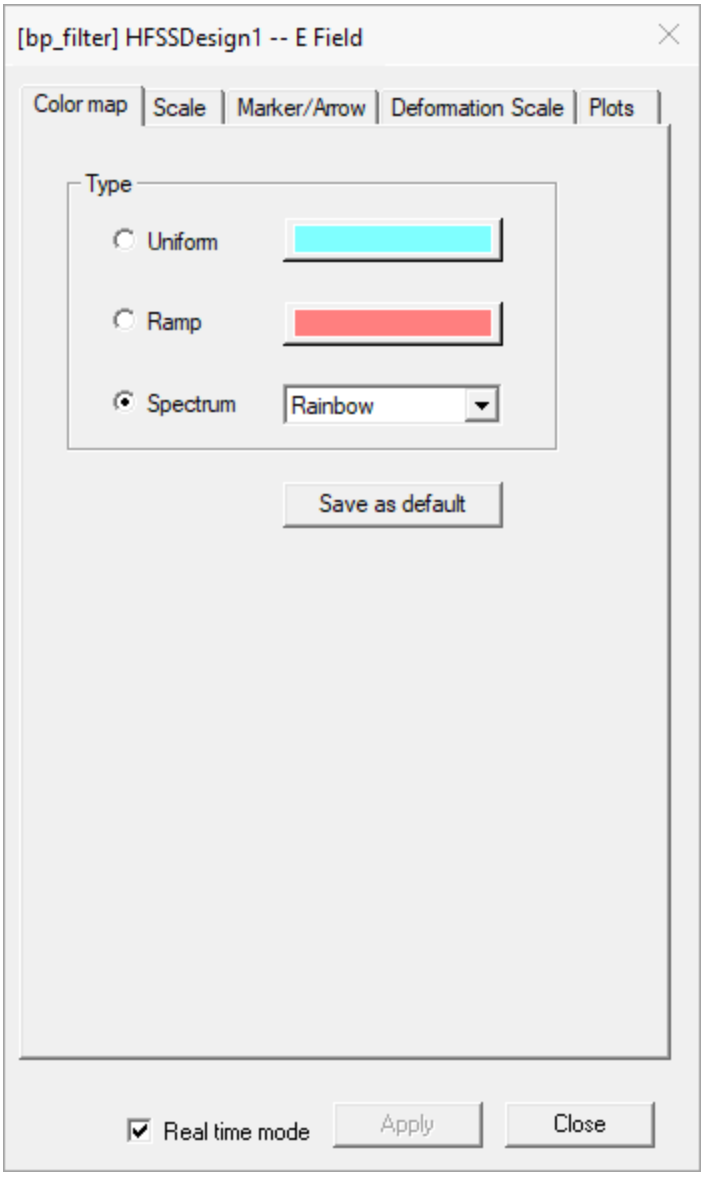
   The **Select Plot Folder** dialog box appears.

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.

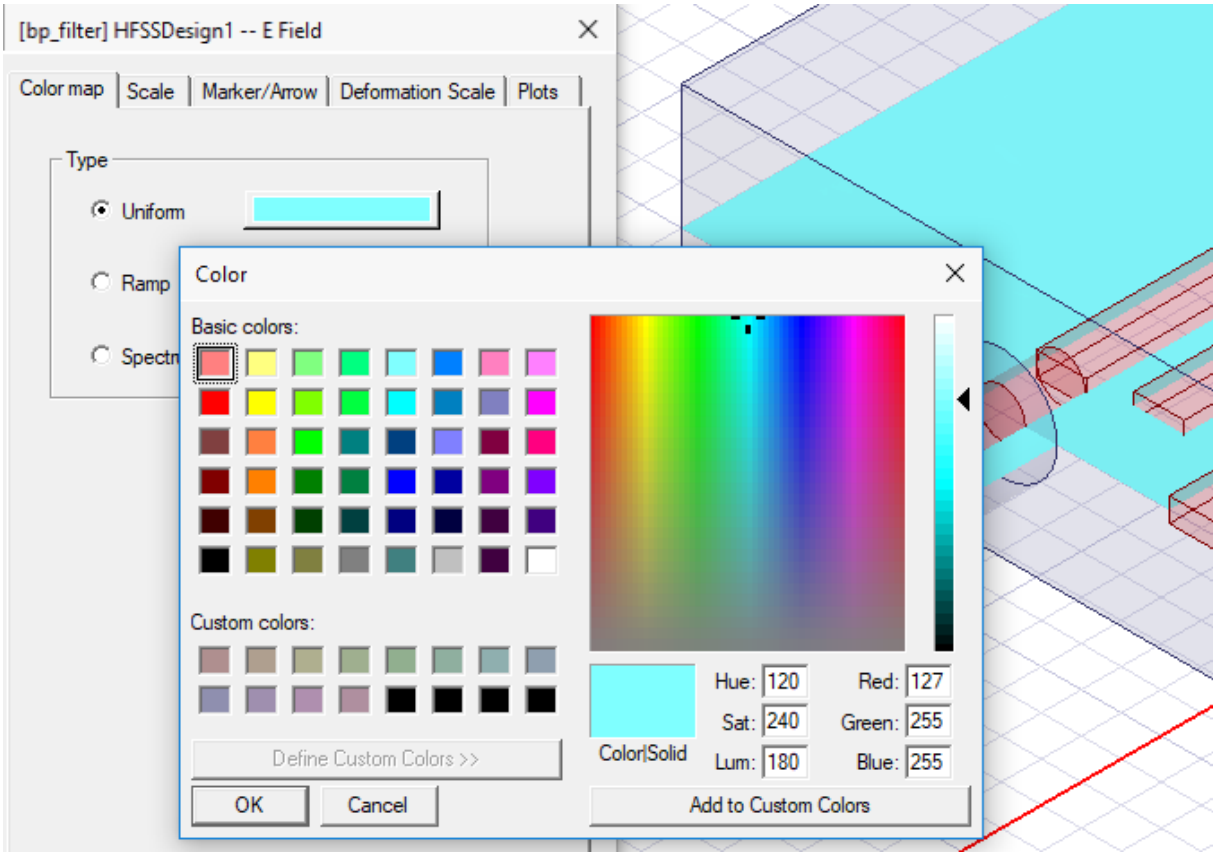
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the **Color Map** tab.
4. Select one of the following color types:

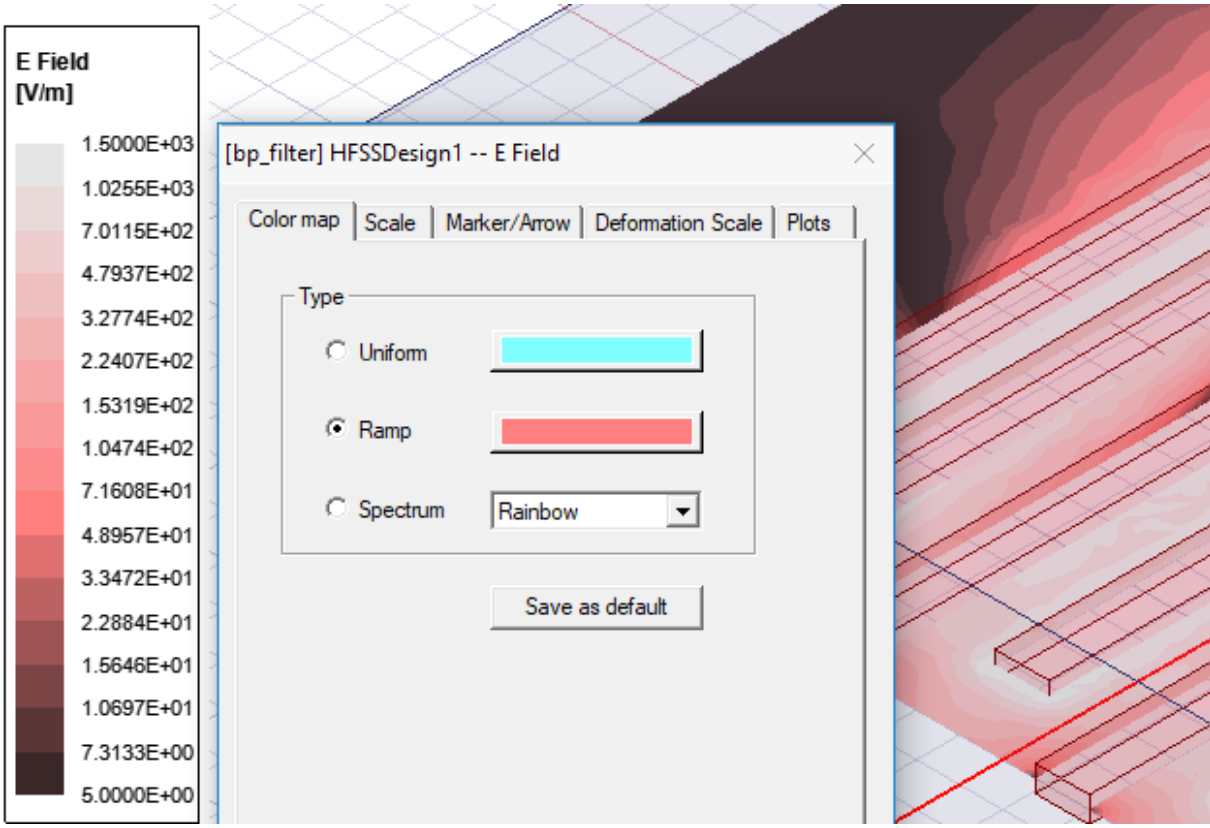
Uniform Field quantities are plotted in a single color. Click the button to choose the plot color from the **Color** button palette.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



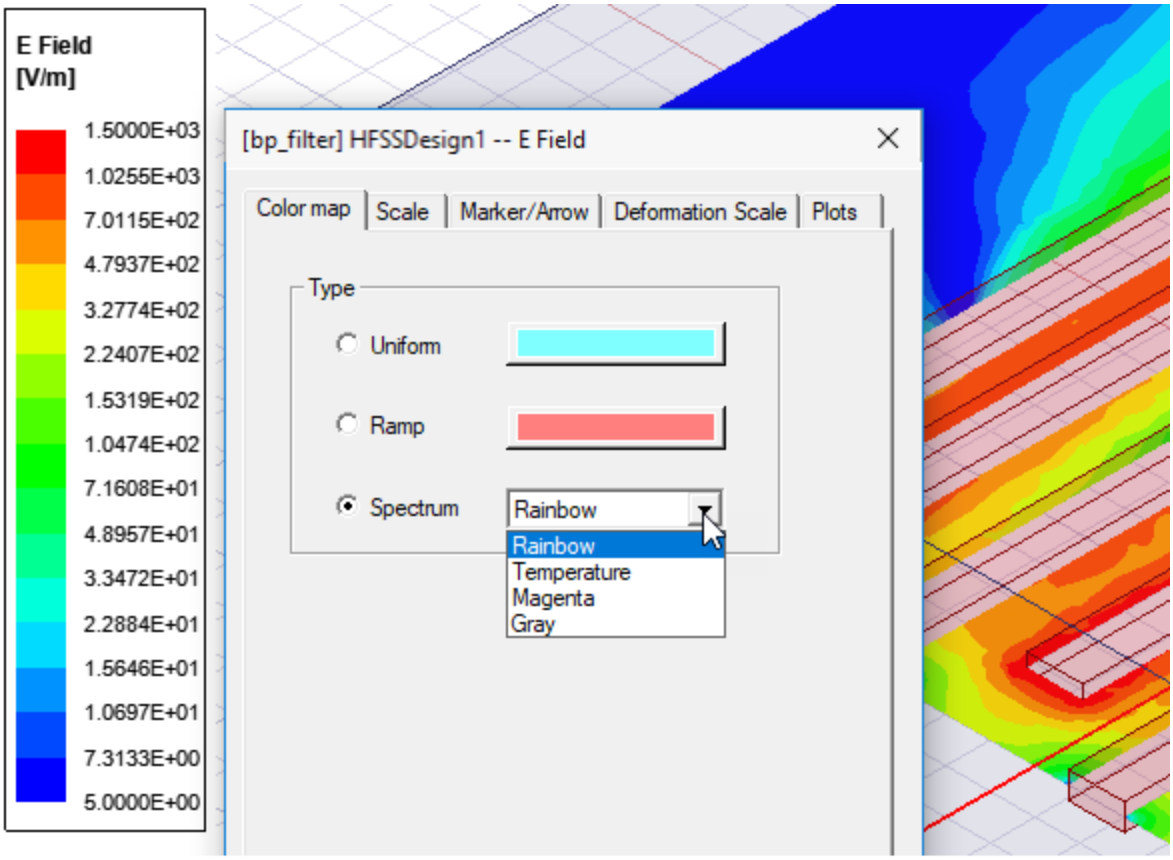
Ramp Field quantities are plotted in shades of a single color. Choose the plot color from the **Color** palette. The shade of the color corresponds to its field value.

Ansys Electromagnetics Suite 2026 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Key Settings

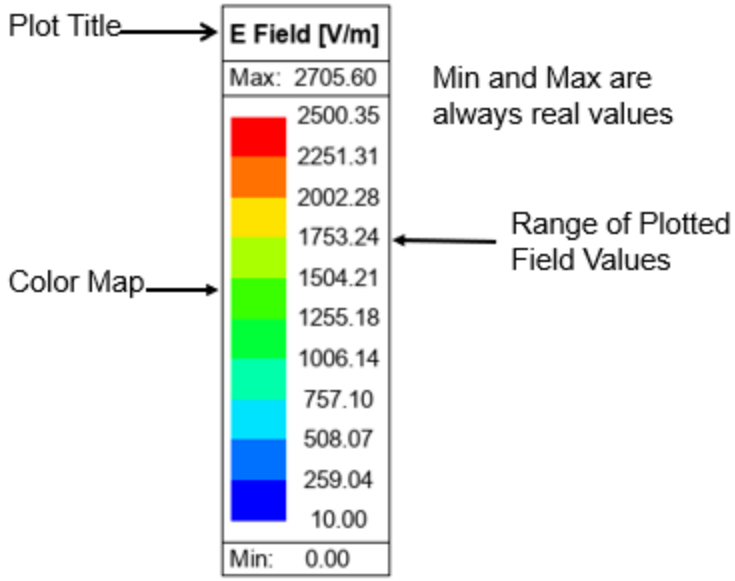
The color key (shown below) displays the range of plotted field values for a field overlay plot. It displays the colors that correspond to the range of field values on the plot. You can click and drag the color key to move it. In the case of no field data, the color key does not appear.

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.

Optionally, when dB is checked on Modifying the Field Plot Scale, dB scale is used for the plot. It disables the Units field and the **Linear** is selected. By design, the min and max on a color key are

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the real range of the field plot which is fixed. It is mainly used as a reference for how you might change how to use the range (it could be a subset, for example, by "Use Limit" or a different lower limit than the real min in "Auto" mode with "Limit Max/Min precision to" is checked, etc.). This also applies to when you use "dB" since the real min could be 0 while the actual lower limit is > 0 so that "dB" applies to generating the levels on the color key. However, dB can not apply to the real min of "0". That is also one of the reasons the Field Plot scale always shows the real range of the field plot as min/max.

Modifying the Field Plot Scale

To change how field quantities are scaled on the field overlay plot:

1. Click **Icepak** > **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.

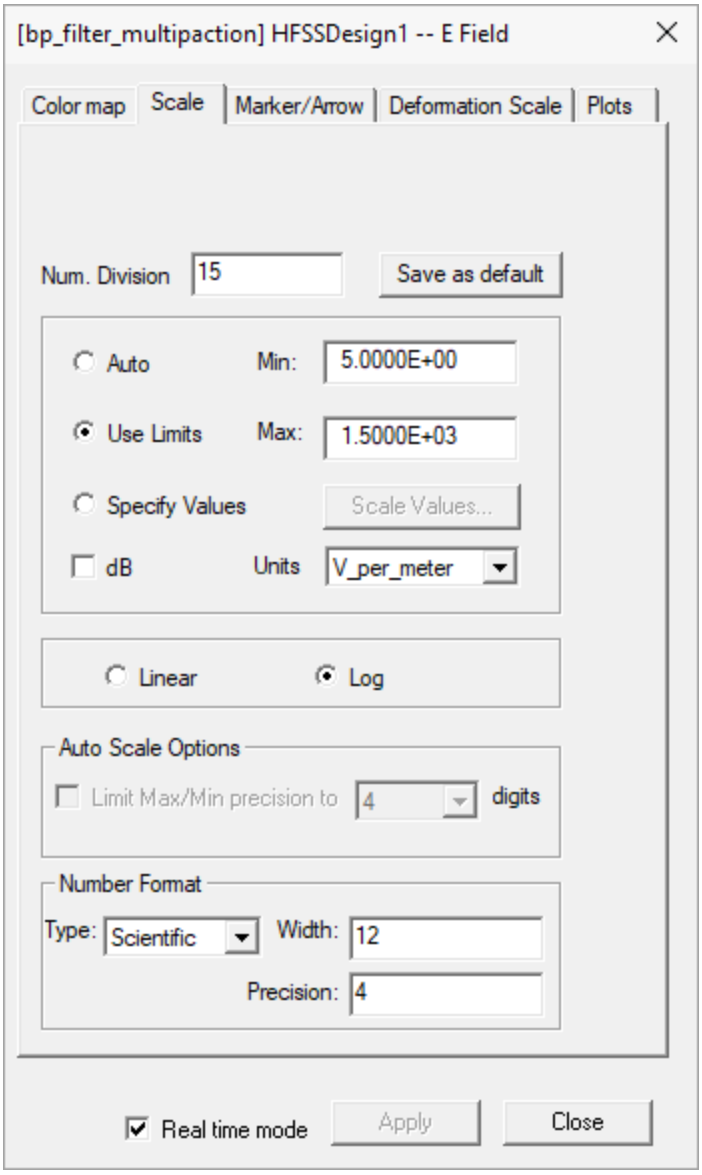
2. Select the plot folder you want to modify and click **OK**.

   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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



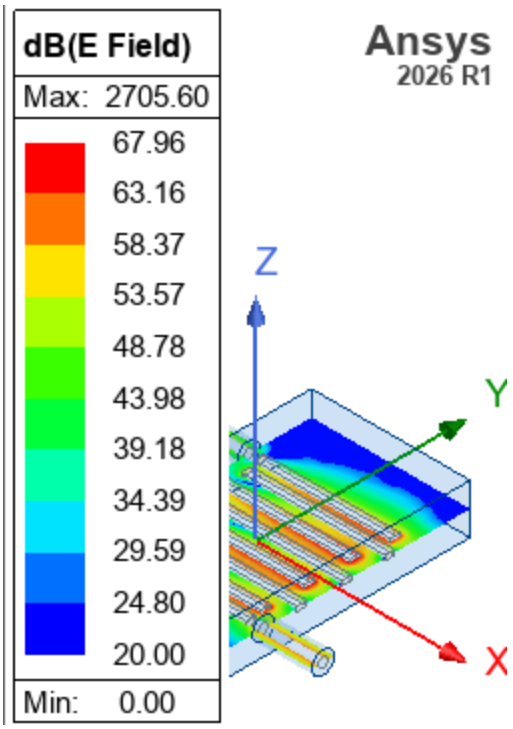
4. Optionally, to change the number of divisions in the field plot scale, set the **Num. Division** field to a new value. You can click **Save as Default**, if desired.
5. Select one of the following scale options:

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 2026 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 and the **Linear** is selected. By design, the min and max on a color key are the real range of the field plot which is fixed. It is mainly used as a reference for how you might change how to use the range (it could be a subset, for example, by "Use Limit" or a different lower limit than the real min in "Auto" mode with "Limit Max/Min precision to" is checked, etc.). This also applies to when you use "dB" since the real min could be 0 while the actual lower limit is > 0 so that "dB" applies to generating the levels on the color key. However, dB can not apply to the real min of "0". That is also one of the reasons the Field Plot scale always shows the real range of the field plot as min/max.



6. Optionally, use the **Units** drop-down menu to select the default unit of measure for the plot.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.)

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

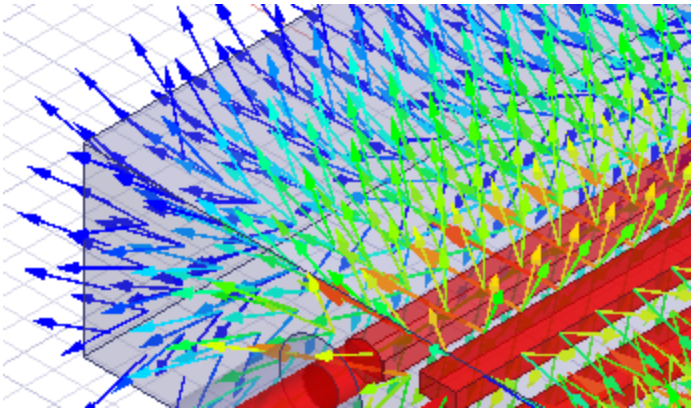
    Auto scale is the default for new plots. For scalar-in-volume plots, iso-surface (rather than cloud) is the default display.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click **Close** to dismiss the dialog box.

Modifying Vector Field Plot Arrows

To change the appearance of a vector field plot's arrows:



1. Click **Icepak**> **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**.

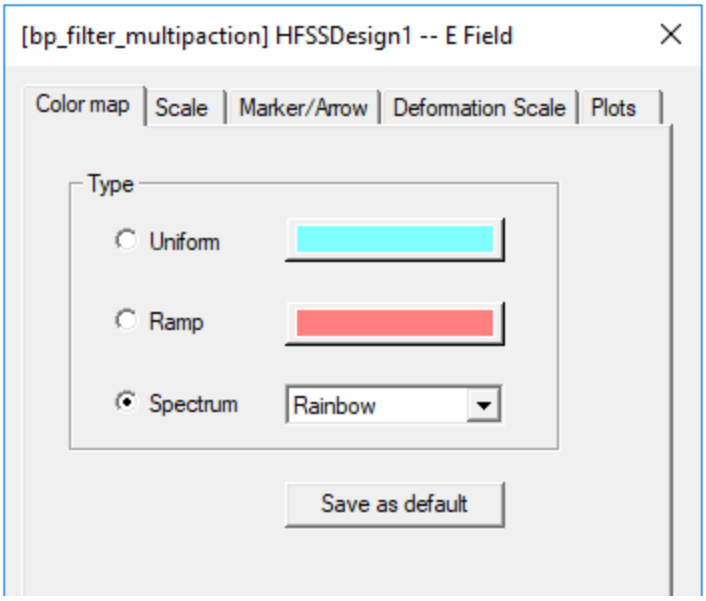
   If you select Modify Plot Attributes, the **Select Plot Folder** dialog box appears. If you select Modify Attributes, the appropriate attributes dialog box appears, bypassing the **Select Plot Folder** dialog box. In this case, skip to step 3.

2. Select the plot folder you want to modify, and then click **OK**.

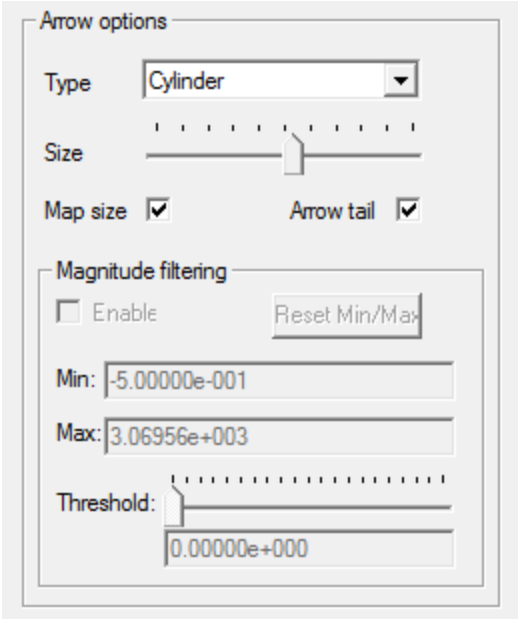
   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

Ansys Electromagnetics Suite 2026 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:

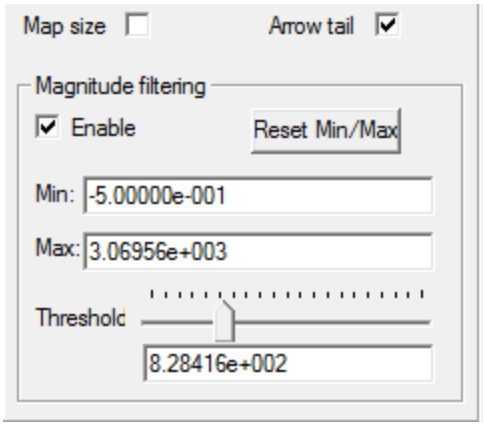
Line	The arrows are displayed as 2D/flat.
Cylinder	The arrow tails are displayed as cylinders. The arrowheads are displayed as 3D/round.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

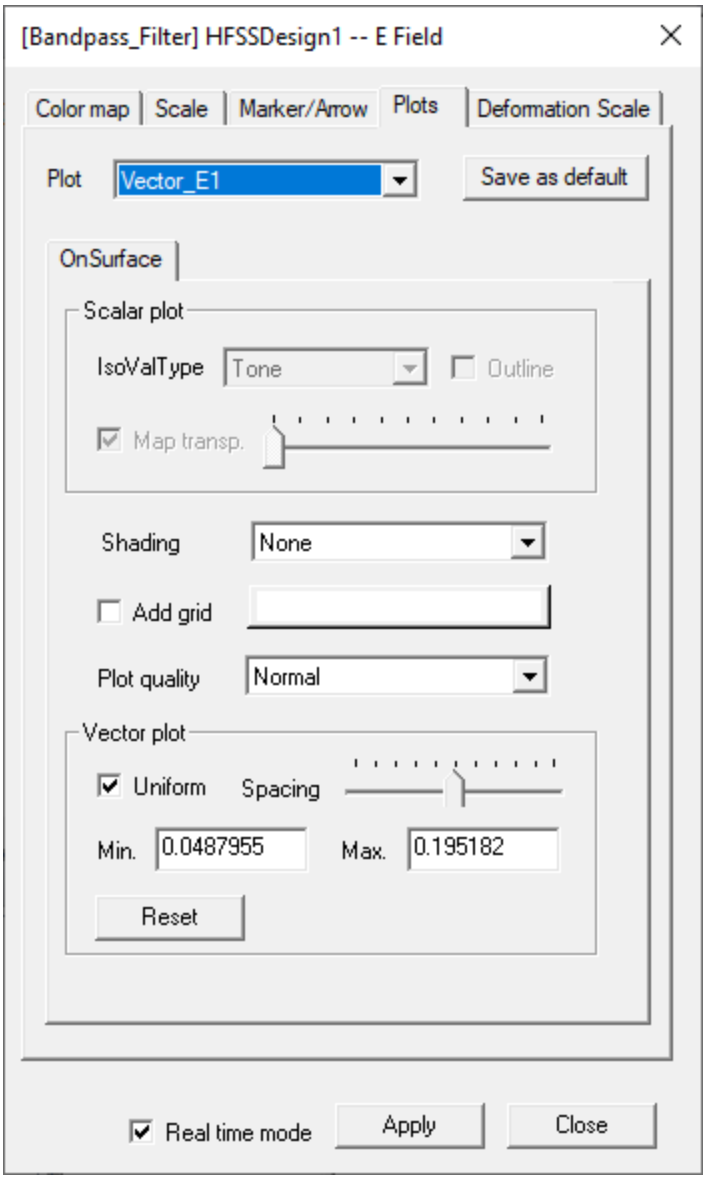
   If you uncheck **Map size**, the Magnitude filtering is enabled. You can specify the Min and Max magnitude filtering, or use a Threshold slider to set a threshold for plotting vectors. Vectors under the threshold or below the Min are not plotted.



The **Reset Min/Max** button lets you reset min/max to the current intrinsic values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click the **Plots** tab. To modify Vector or Streamline plot appearances, first select the plot.



The Shading options affect the appearance of arrows and streamlines. Selecting a plot enables the Vector plot settings for spacing and min and max.

9. Icepak 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Click **Close** to dismiss the dialog box.

Setting the Mesh Visibility on Field Plots

To display or hide the mesh on field plots, or change the mesh's color:

1. Click **Icepak**> **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.

Modifying Scalar Field Plot Isovalues

1. Click **Icepak** > **Fields** > **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

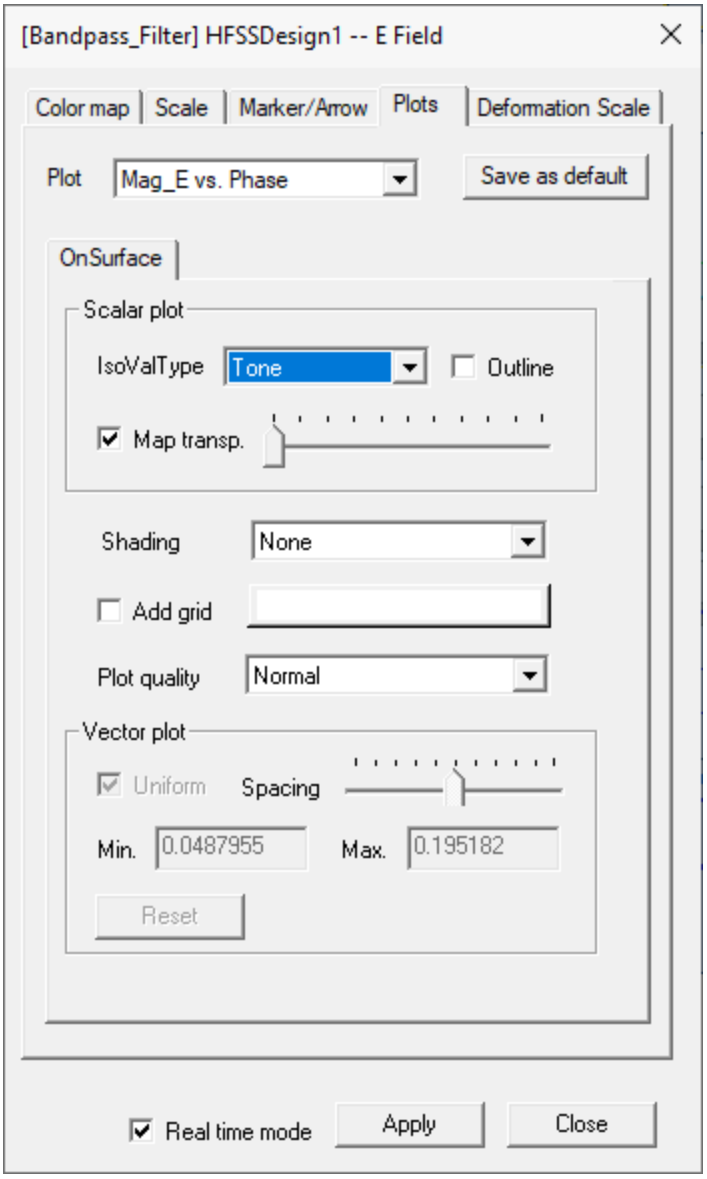
2. Select the plot folder you want to modify, and then click **OK**.

   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 menu lets you select the desired plot. The **OnSurface** tab appears for surface plots and **InVolume** for volume plots.

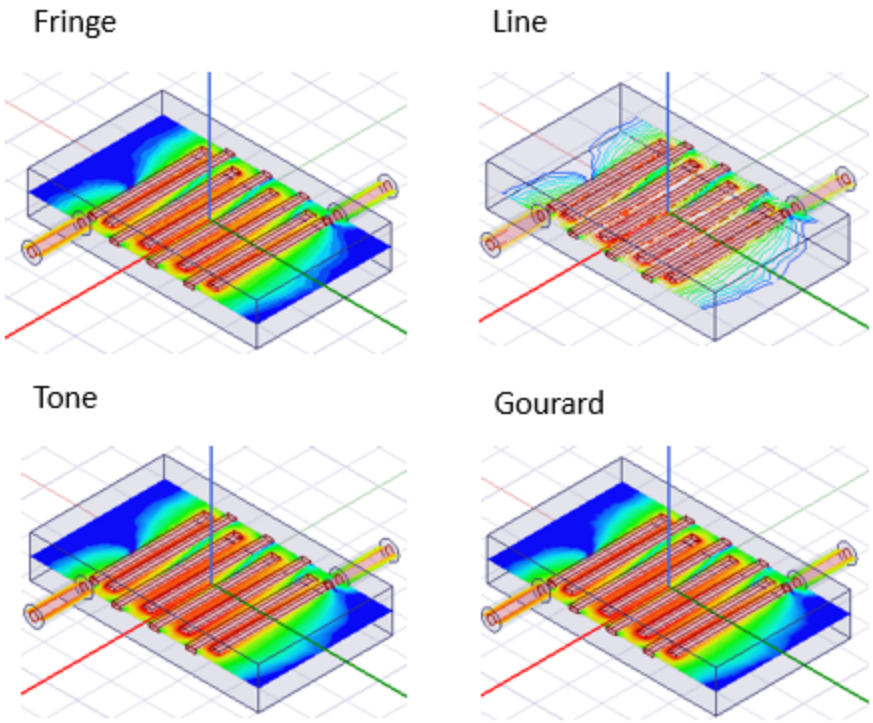
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 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.

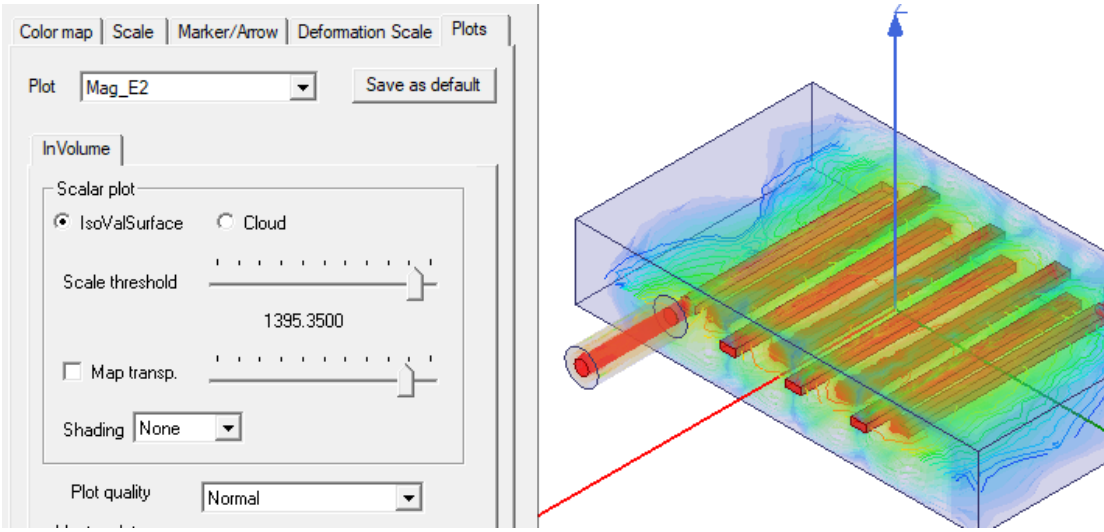
c. For shading on OnSurface plots, if you are using Lighting, you can adjust the Shading as None, Flat, or Smooth.

5. If the plot is a scalar volume plot, do the following:

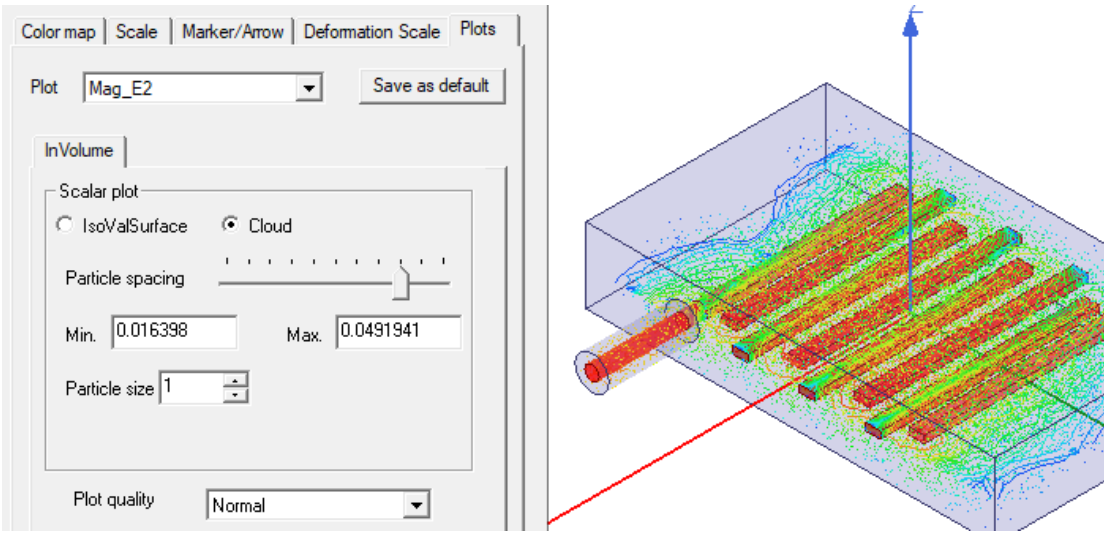
a. Select one of the following display types:

IsoValSurface	Color is drawn on the isovalues. This choice includes “Scale threshold” slider bar that 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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mapping Scalar Field Plot Transparency to Field Values

1. Click **Icepak** > **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.

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 **Icepak**> **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**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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**.

Modifying Line Plots

Field quantities can be plotted directly on a line object. Scalar quantities are plotted as 3D color-shaded lines. Vector quantities are plotted as arrows that are based on the line.

To modify the appearance of line plots:

1. Click **Icepak** > **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.
4. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

5. Select one of the following styles for the line object in the **Line style** drop-down menu:

Cylinder	The line object is shaped like a cylinder.
Solid	The line object is a 3D solid.
Dash-Dash	The line object is represented by dashed black line segments.
Dot-Dot	The line object is represented by a series of dots.
Dash-Dot	The line object is represented by a series of alternating dashed black line segments and dots.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Use the **Line width** slider to increase (move to the right) or decrease (move to the left) the thickness of the line.
7. By default, a polyline object is divided into 100 equally spaced points for post processing. To modify the number of points on the line, type a new value in the **Number of points** text box.
8. 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.

9. Click **Close** to dismiss the window.

ad109dc8.png

Function Command

The **Function** command enters a predefined scalar or vector function into the top register of the calculator:

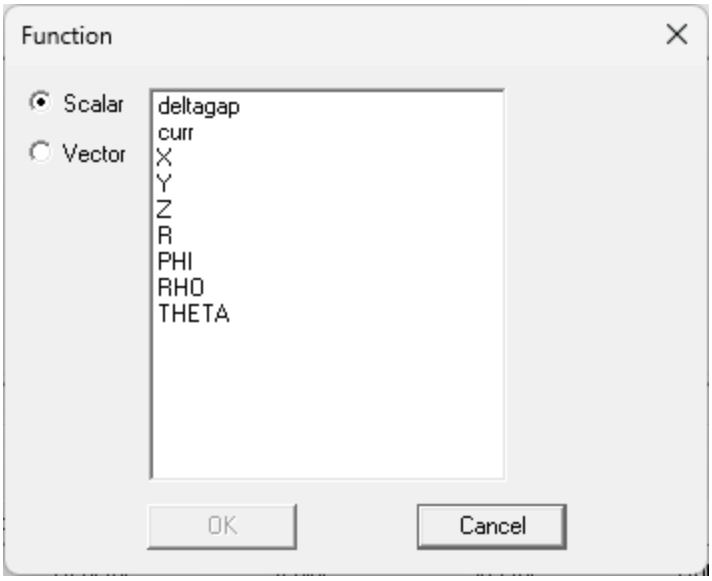
Scalar

A scalar function.

To enter a function:

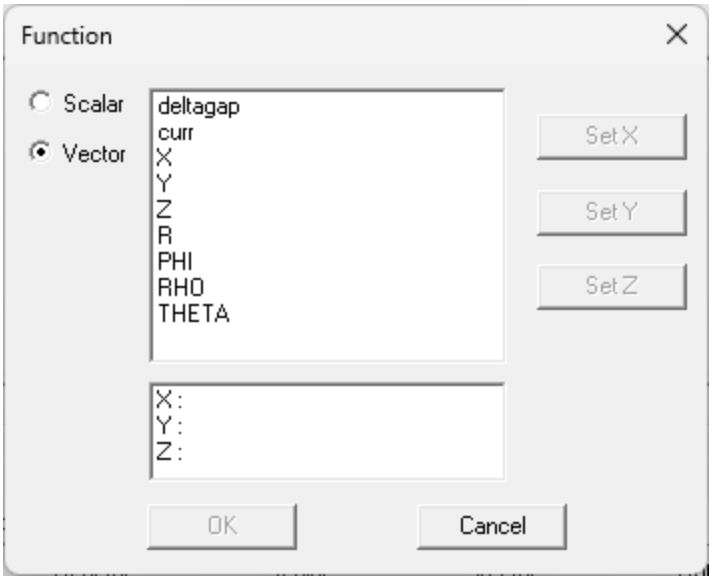
1. Click **Function**.

   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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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. 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:

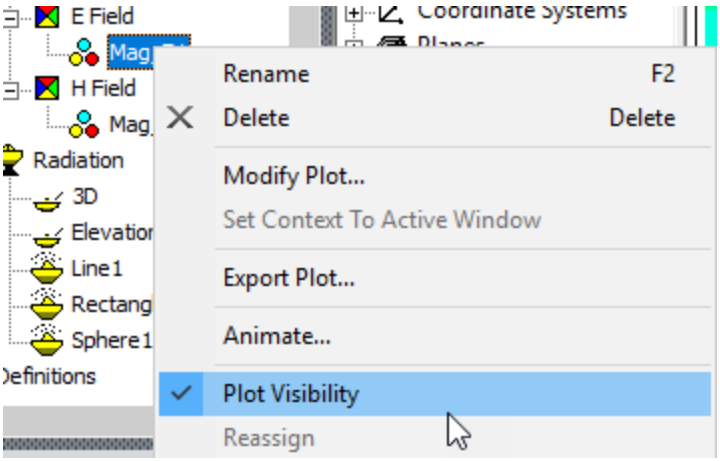
The predefined variables **X, Y,** and **Z**; **RHO, THETA, R,** and **PHI**; and any functions that you created can be used to define functional scalar and vector quantities.

Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

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 Project Manager and checking or unchecking **Plot Visibility**.

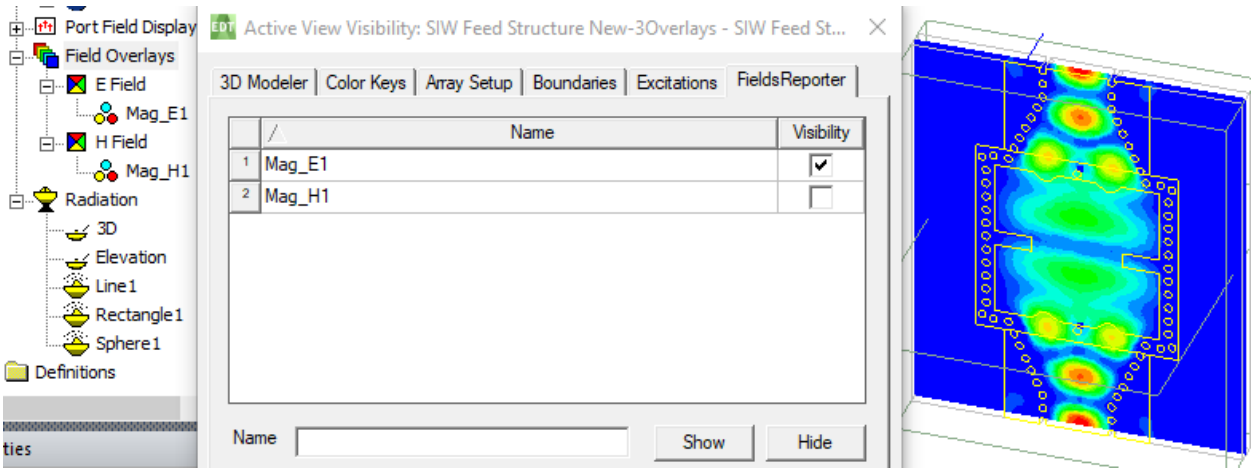
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

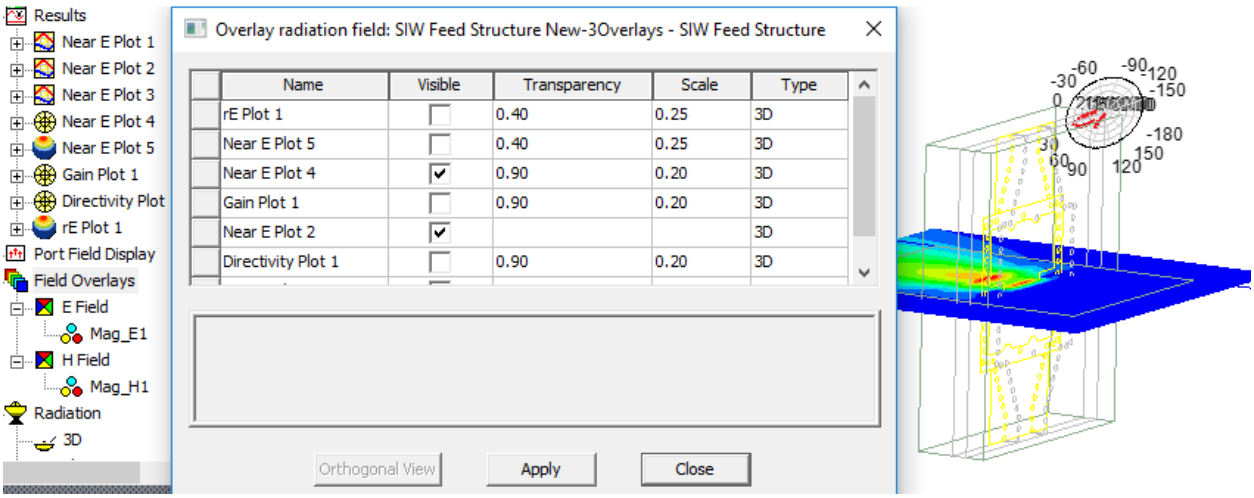


2. Click the **FieldsReporter** tab.
3. In the **Visibility** column, select the field overlay or mesh plots you want to display. Clear the plots you want to hide from view.

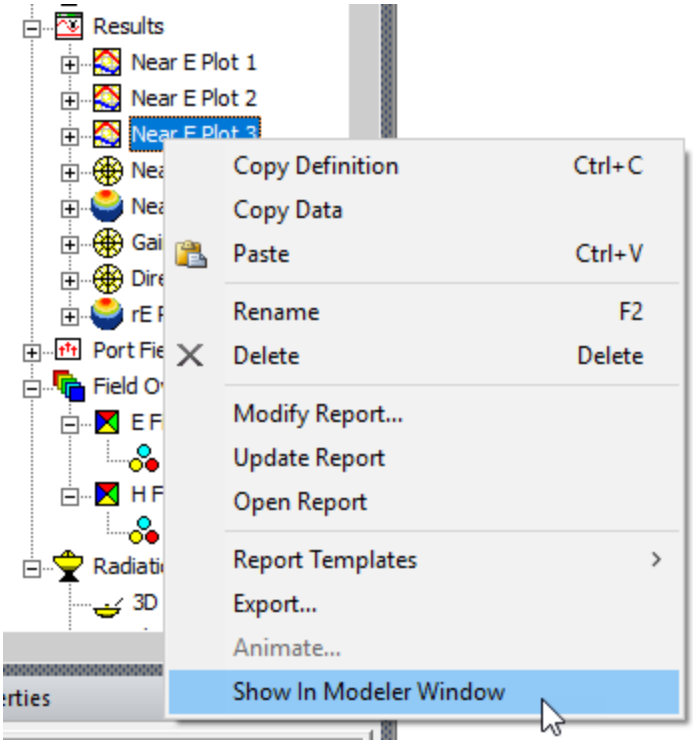
   Only the selected plots will be visible. Multiple plots can be shown at once.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**.



Saving a Field Overlay Plot

Field overlay and mesh plots are saved in the project file; however, you can save a plot to Field Plot File format (.dsp) and then open it in Icepak.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To save field overlay or mesh plot data to a .dsp file:

1. In the project tree, click the plot you want to export.
2. Click **Icepak** > **Fields** > **Save as**.

   The **Select Field Plot(s)** dialog box appears.
3. Select the plots you want to export by checking the **Select** box, and then click **OK**.

   The file browser appears. **Field Plot Files (.dsp)** is the selected file type.
4. Specify the name of the .dsp file and the location in which to save it.
5. Click **Save**.

The plot is exported to the specified .dsp file.

To open the file you created, click **Icepak** > **Fields** > **Open**.

Opening a Field Overlay Plot

To open a field overlay or mesh plot that you have saved to Field Plot File format (.dsp):

1. Click Icepak > **Fields** > **Open**.
2. The file browser appears. **Field Plot Files (.dsp)** is the selected file type.
3. Browse to the location of the .dsp file you want to open, and then click the file name.
4. Click **Open**.

The plot appears in the view window. It is listed under **Field Overlays** in the project tree.

Deleting a Field Overlay Plot

1. Click **Icepak** > **Fields** > **Delete Plot**.

   The **Delete Plots** dialog box appears.
2. Select the plots you want to delete by checking the **Delete** check box.
3. Click **OK**. The selected plot are deleted.

Alternatively, right-click the plot in the Project Manager tree that you want to delete, and then select **Delete**.

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If a plot folder has not been created, click **Field Overlays** in the Project Manager.
2. Click **Icepak** > **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 Icepak 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 folder	Select the existing folder in which you want to store new plots.

> **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**.

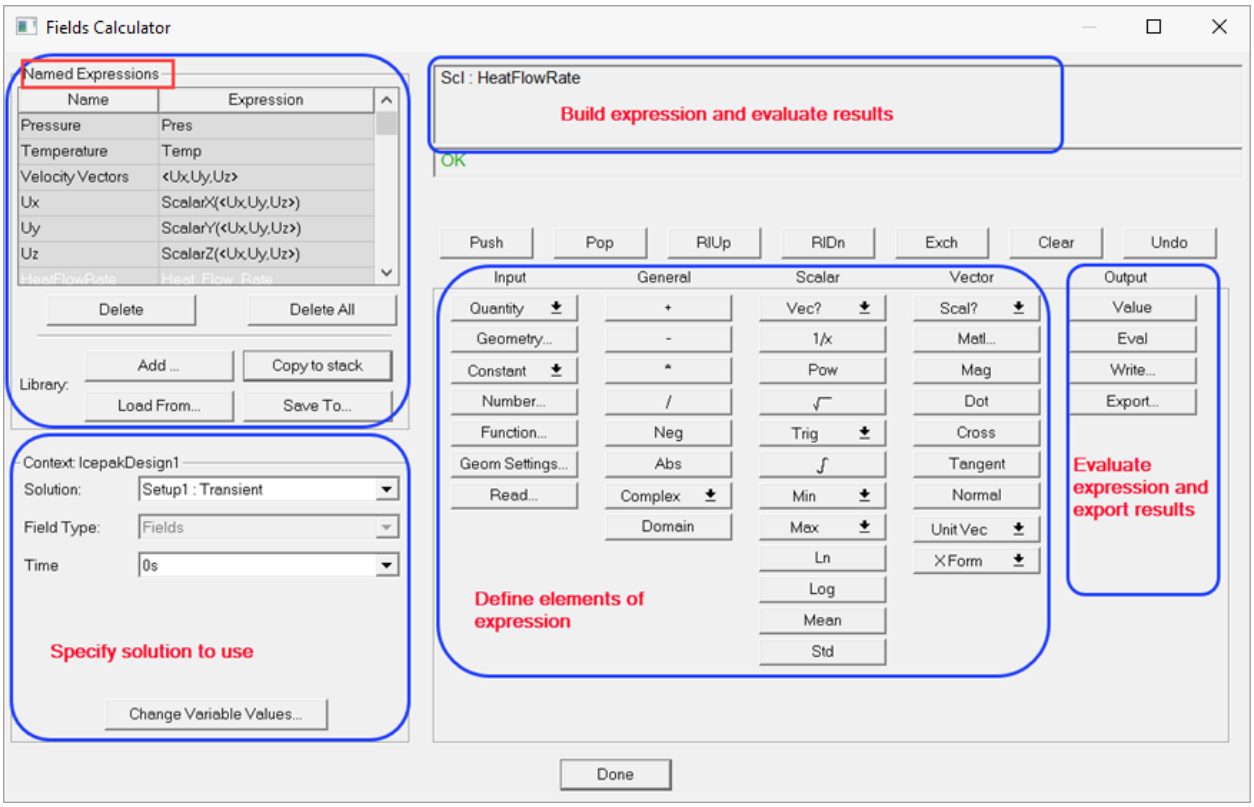
Using the Fields Calculator

The Fields Calculator is a very powerful tool for postprocessing. It is available for HFSS, Maxwell, Q3D, Icepak, and IcepakFEA solutions. While standard postprocessed 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.

The Fields Calculator includes predefined expressions appropriate for each solver and lets you create and save additional named expressions.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To view information on a command or screen area, click over the button or screen area on the illustration below.



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, Vector, and Output.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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.

Opening the Fields Calculator

To open the Fields Calculator, from the Desktop Menu bar do one of the following:

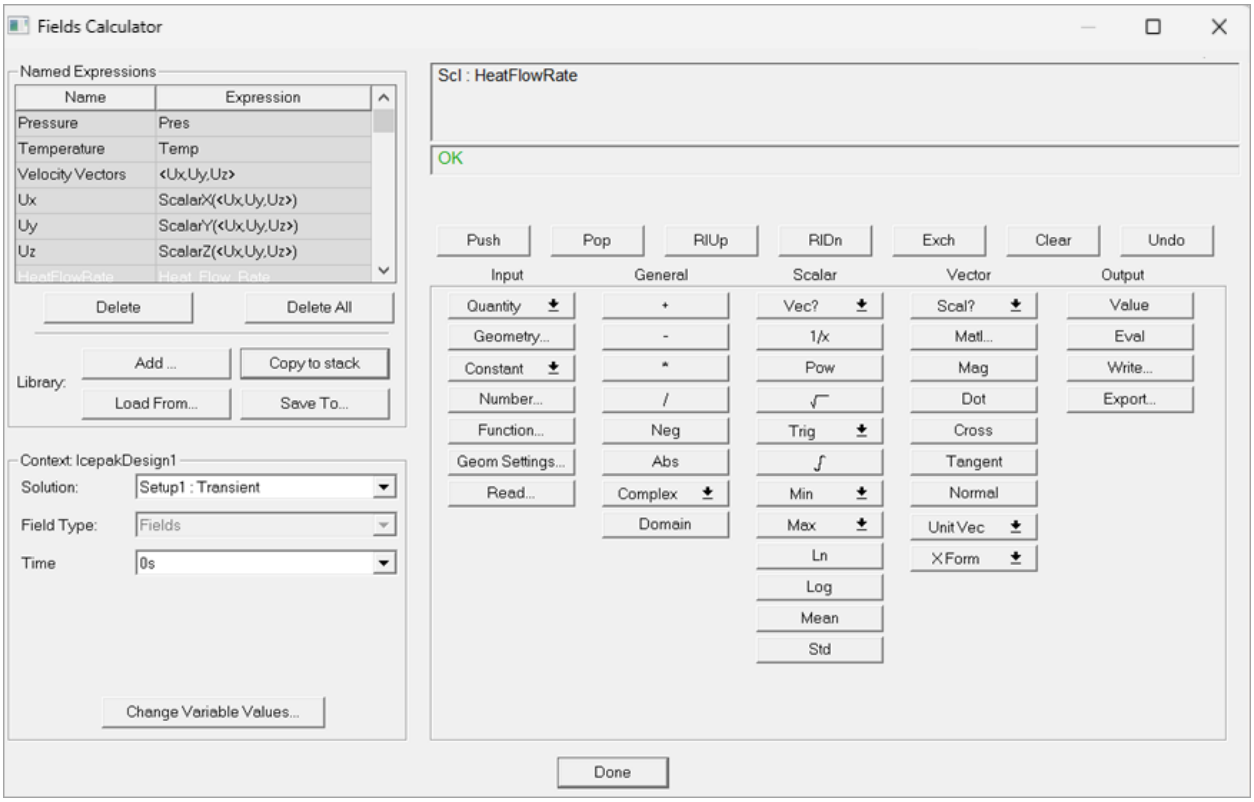
- Click **Icepak** > **Fields**> **Calculator**

  or

- Right-click **Field Overlays** in the Project Manager and choose **Calculator** from the shortcut menu.

The *Fields Calculator* window appears.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

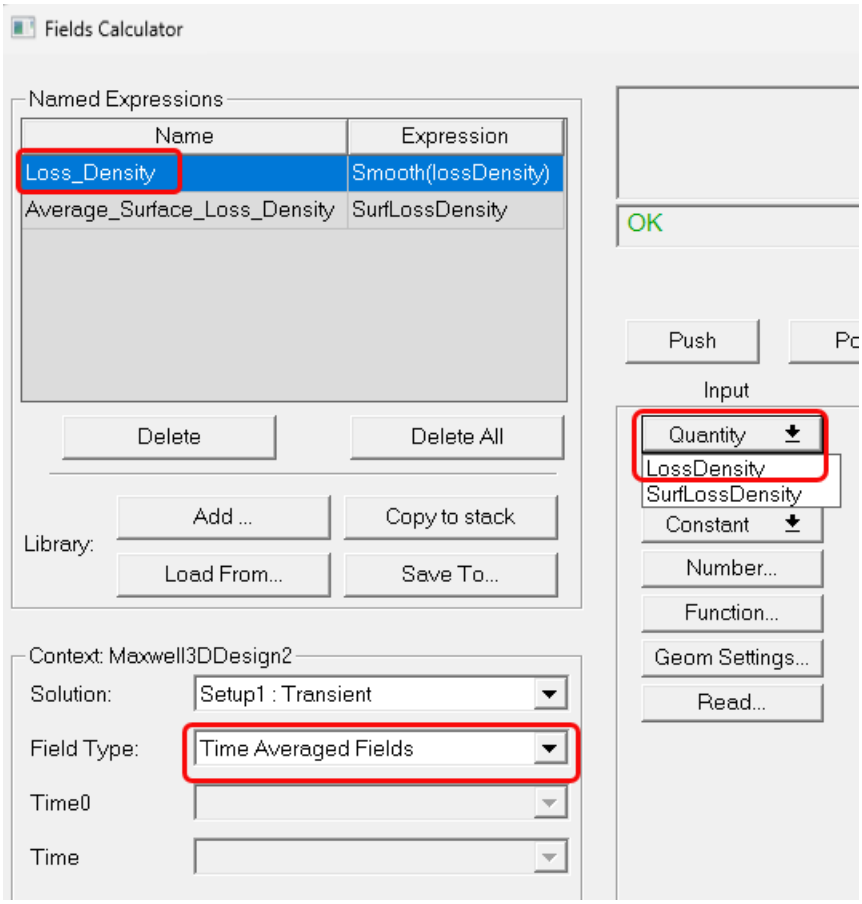


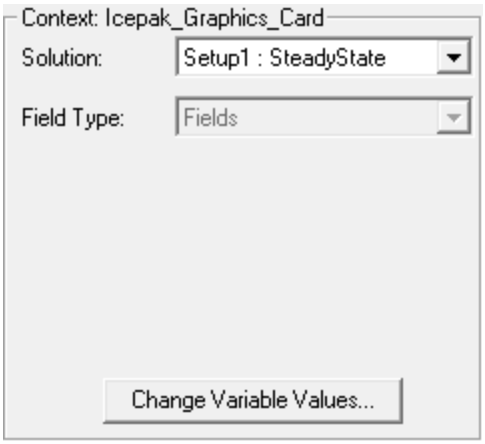
Context Area

The panel at the upper right of the window identifies the context to be used for the calculations. The top line identifies the design. The text entry boxes allow you to select the **Solution** and **Field Type**.

When coupling a Maxwell transient solution with an Icepak design, the field type must be **Time Averaged Fields**. Selecting **Time Averaged Fields** adds an additional **Time0** field, which is used when calculating time averaged losses. Selecting **Time Averaged Fields** also adds **Loss_Density** as a **Named Expression**, and **LossDensity** as a **Quantity command** selection.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





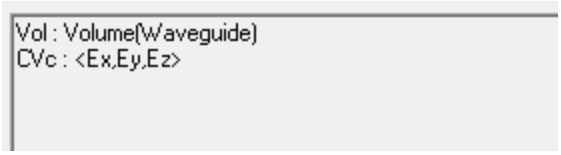
The Field Type here is not related to the edit sources. This is a general term among Ansys EM products (HFSS, Maxwell, Q3D, Icepak, and IcepakFEA). Some products have more than one field type for different solution types. If only one Field Type is available the drop-down menu is unavailable.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

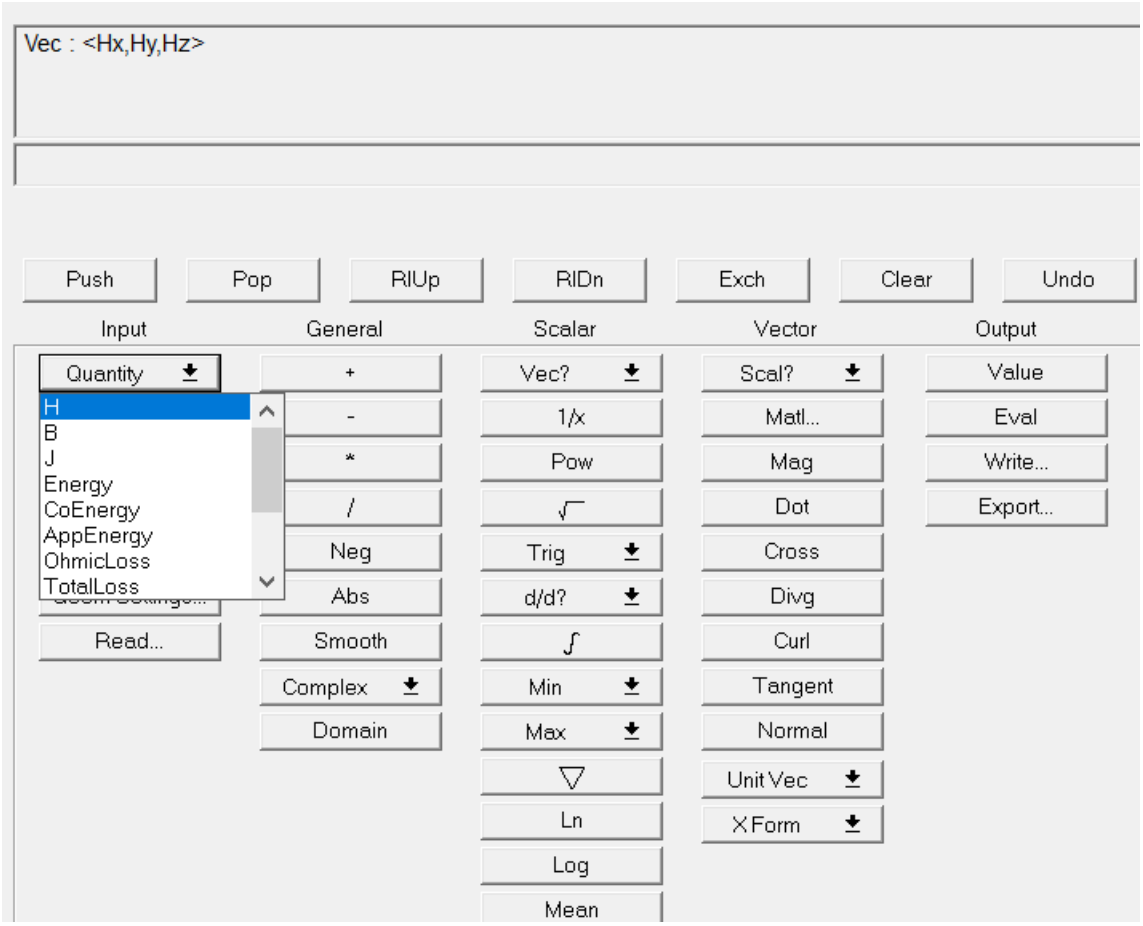


Each register can hold

- 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 2026 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 2026 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.

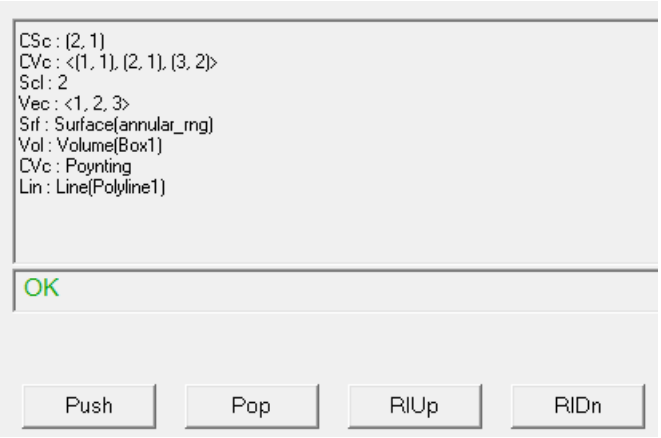
Registers

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 2026 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 Vec? drop-down menu in the Scalar column. The choices VecX, VecY, and VecZ convert the scalar data to vector data aligned with

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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, which are numerical values 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 menu 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, which 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, which 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 can 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 Surface geometry set. To select only the portion of a given data entry which exists along, on, or within a given geometry quantity,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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.

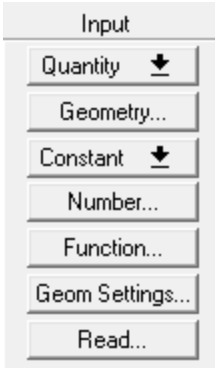
Units of Measure

Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

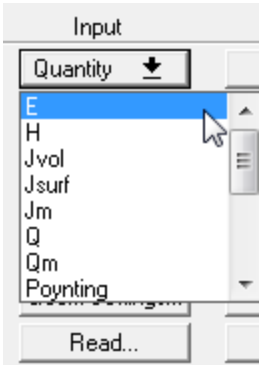
Input Commands

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 2026 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 selections, 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 2026 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	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:

Pres	The relative static pressure of the fluid and is available when flow is computed.
Temp	Available when temperature is computed.
U	The x, y, or z component of the velocity vector and is available when flow is computed.
Heat_Flow_Rate	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.
Heat_Flux	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.
Mass_Flux	Is the mass flow rate per unit area and can only be computed on the surface of model objects.
Surf_Pressure	The relative static surface pressure of the fluid and is

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	available when flow is computed.
Radiation_Flux	The radiation heat flow per unit area and can only be computed on surfaces of objects. It is available if you are solving your problem with radiation effects.
Surf_Temp	The surface temperature and is available when temperature is computed. See Icepak Post Processing Quantities for more information.
Turbulent_Viscosity_Ratio	The ratio of the turbulent viscosity to the laminar viscosity. It can be reported if any of the available turbulence models are used.
Turbulent_YPlus	is a nondimensional parameter. See Icepak Post Processing Quantities for more information.
Turbulent_Kinetic_Energy	The turbulent kinetic energy. See Icepak Post Processing Quantities for more information.
Turbulent_Epsilon	The turbulent dissipation rate. See Icepak Post Processing Quantities for more information.
Heat_Transfer_Coeff	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.
Thermal_Cond_X, Y, and Z	The x, y, or z component of the orthotropic thermal conductivity vector and is available when traces are imported.
Speed	The magnitude of the velocity vector and is available when

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	flow is computed.
Heat_Radiative_Flow	A derived scalar quantity. It is available if you are solving your problem with radiation effects. It can be computed only on surfaces of objects.
Mass_Flow_Rate	A derived scalar quantity computed from the velocity field. See Icepak Post Processing Quantities for more information.

Geometry Command

The Fields Calculator **Geometry** command opens a dialog box for selecting a geometry to load into the top register of the calculator.

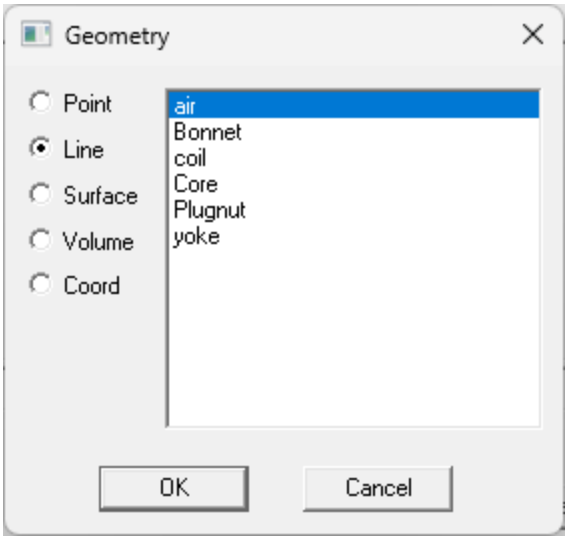
Do this to

- 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.

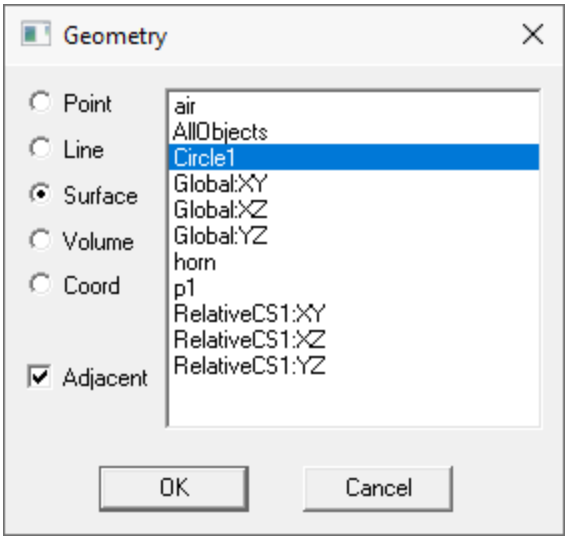
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creatable **Sheet** objects and face selections 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 box also lists an **Adjacent** command that controls the side of the sheet on which the fields are calculated.

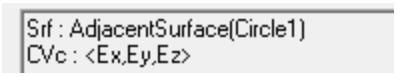
Surface



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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	adjacent side of the surface. • Support scripting, saving and loading expression with adjacent side surface geometry. 
Volume	3D objects, regions, and object selections (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:

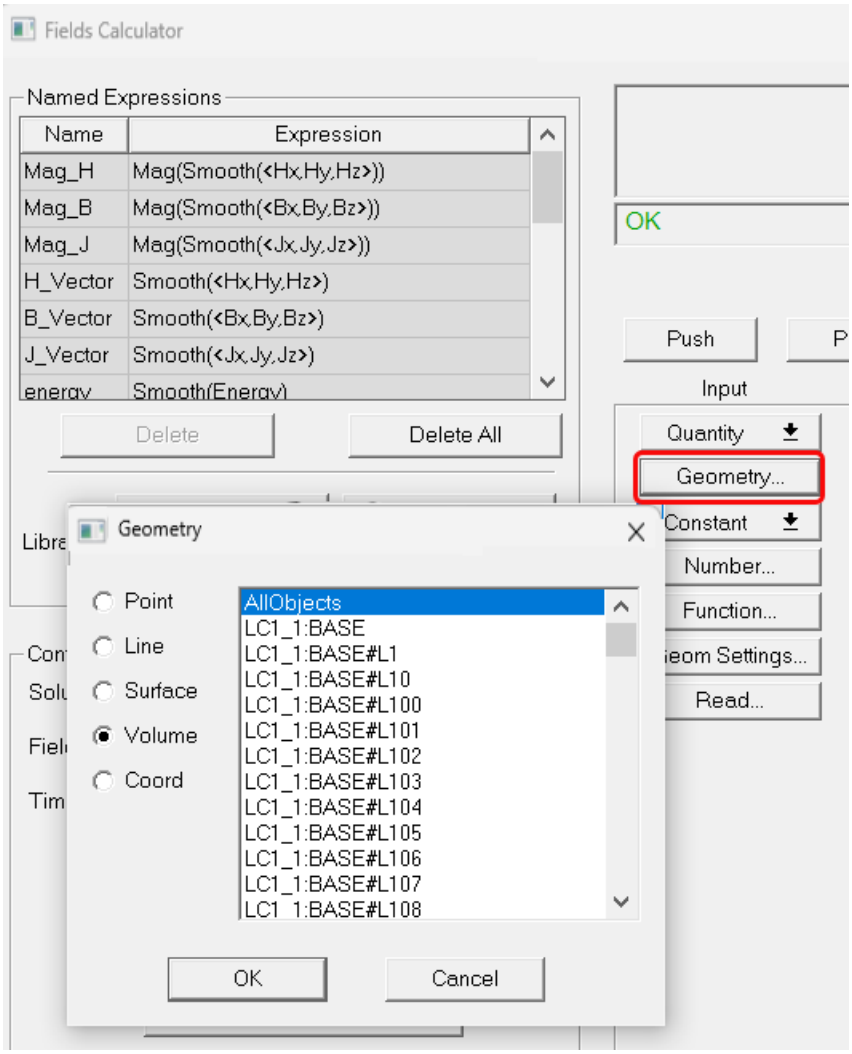
1. In the Fields Calculator, click **Geometry**.

   The **Geometry** dialog box appears.

2. Select a geometry type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 from regular 3D geometry. These additional objects are generated only after reading a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh, so they won't show up if there is no solution.

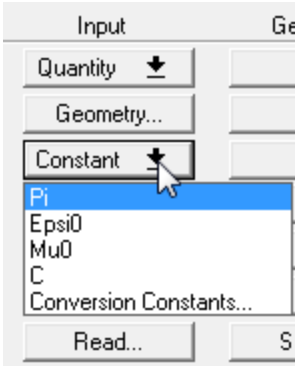


3. Click the geometry.
4. Click **OK** to load the geometry.

> **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.

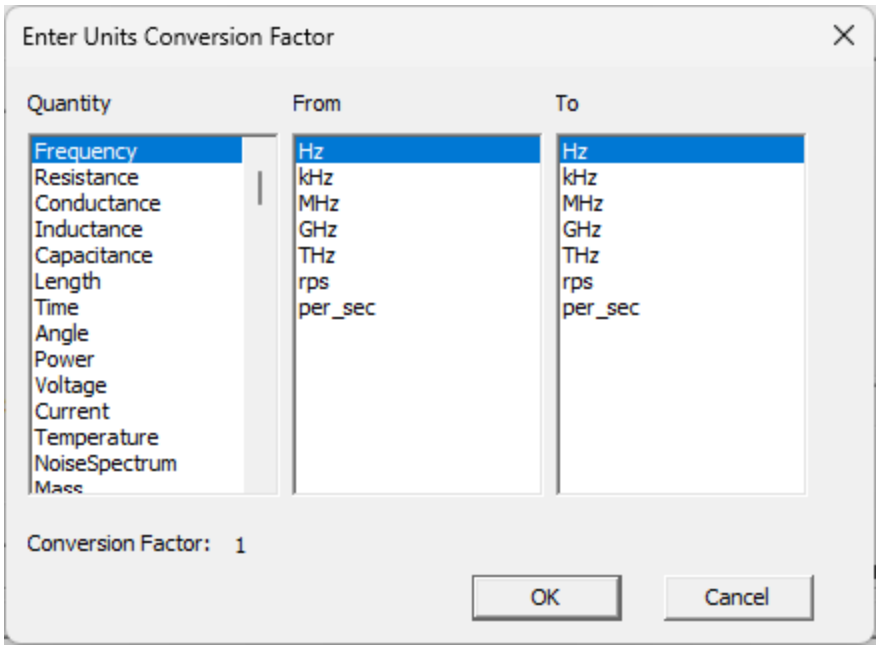
Constant Command

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 2026 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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. 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.

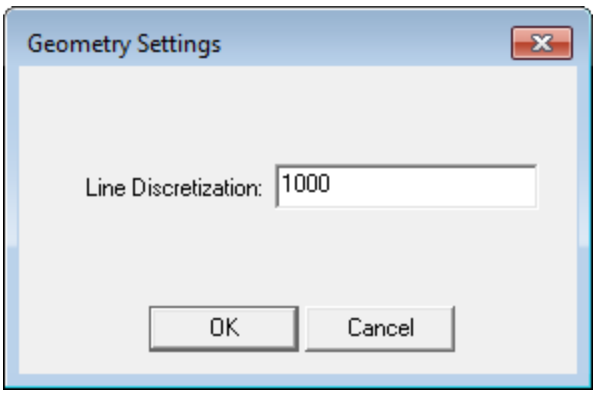
Geom Settings Command

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.



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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **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)**

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Steps for Calculating the Complex Vector Electric Field

These are the field calculator steps to obtain the real part, the imaginary part, and the magnitude of the x-directed, y-directed, and z-directed components of the phasor electric field. For each of these vector components, the magnitude should be equal to sqrt(real^2+imag^2), but the need to interpolate values and the calculation sequence means that Icepak does not give this value unless the specified location is directly on a mesh element node.

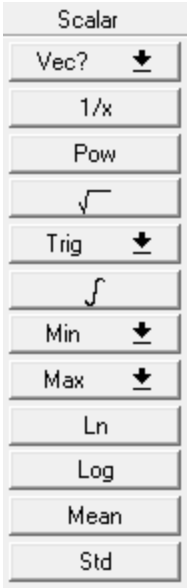
1. Calculate real part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Real
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval
2. Calculate imaginary part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Imag
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval

Use the real and imaginary components to manually calculate the magnitude as the sqrt (real^2+imag^2).

Scalar Commands

The Scalar Column contains calculator operations which can only be performed on scalar stack entries.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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. 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).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

   > **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. Icepak 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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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. • 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 **Cmplx/Real** or **Cmplx/AtPhase** 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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 one of the following: Cylindrical or Spherical.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **Warning:**
>
> The Matl command is not compatible with nonlinear material properties.

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.
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.

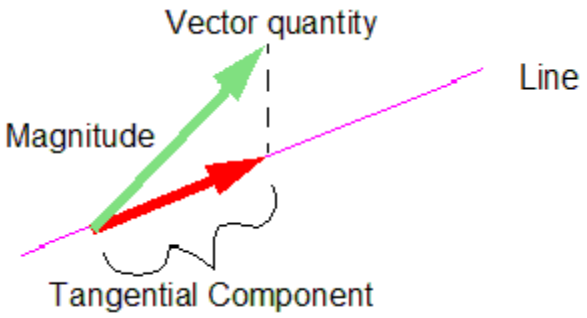
Curl Command

Takes the curl of the vector quantity in the top register.

Tangent Command

Computes the tangential component of a vector quantity along a line.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



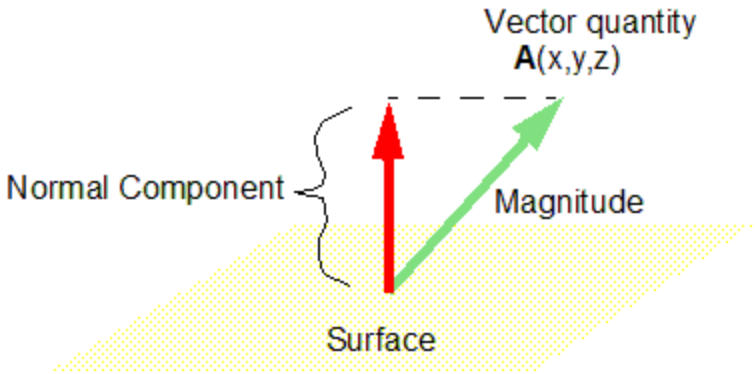
To take the tangent of a vector:

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:

$$Normal = A(x, y, z) \cdot \hat{n}$$



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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Icepak, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.

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.

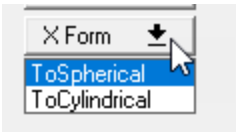
Select from the following:

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**):

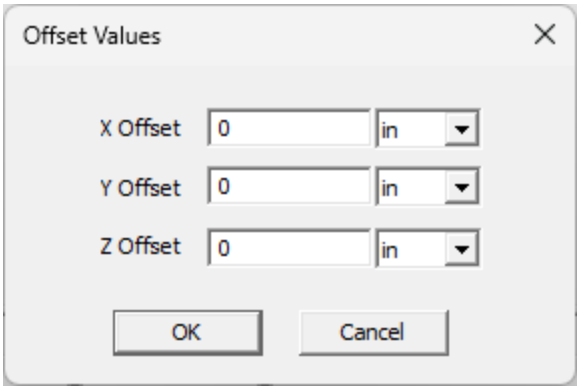
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.



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>)

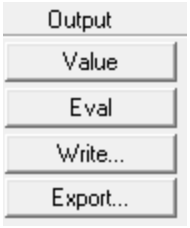
After you transform the quantities, you can then access the vector components in the Fields Calculator.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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).



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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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).

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.

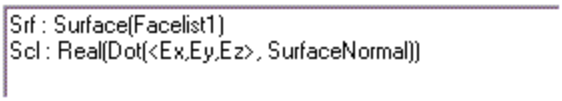
To find the value of a field quantity at a point:

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**.

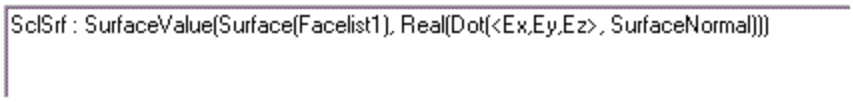
To view the numerical results of this operation, use the **Eval** command.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Now you can perform suitable operations such as Write, Integrate, etc. For PointValue you can also do **Eval**.

Eval Command

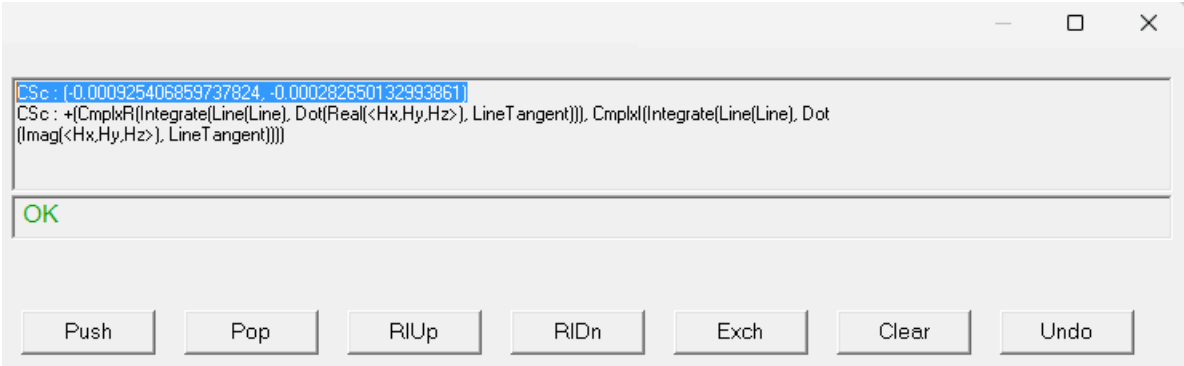
The **Eval** command finalizes computations and converts the text-string indicating the computation being performed to an actual numerical value. This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



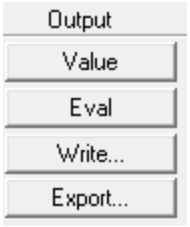
$$\boldsymbol{I} = \oint \boldsymbol{H} \cdot \boldsymbol{dl}$$

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.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

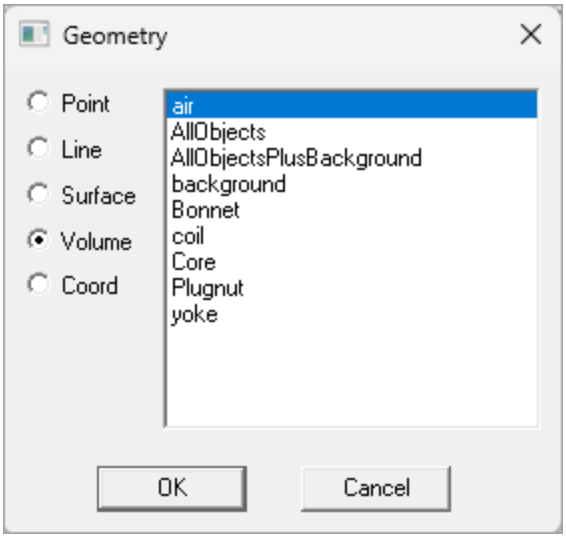
- Save registers for use during a later postprocessing session.
- Save a field quantity for use when postprocessing a different model.

> **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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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**.

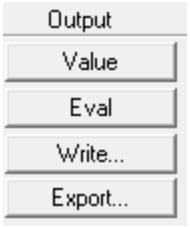
   The contents of the register are saved to the file you specified.

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 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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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:

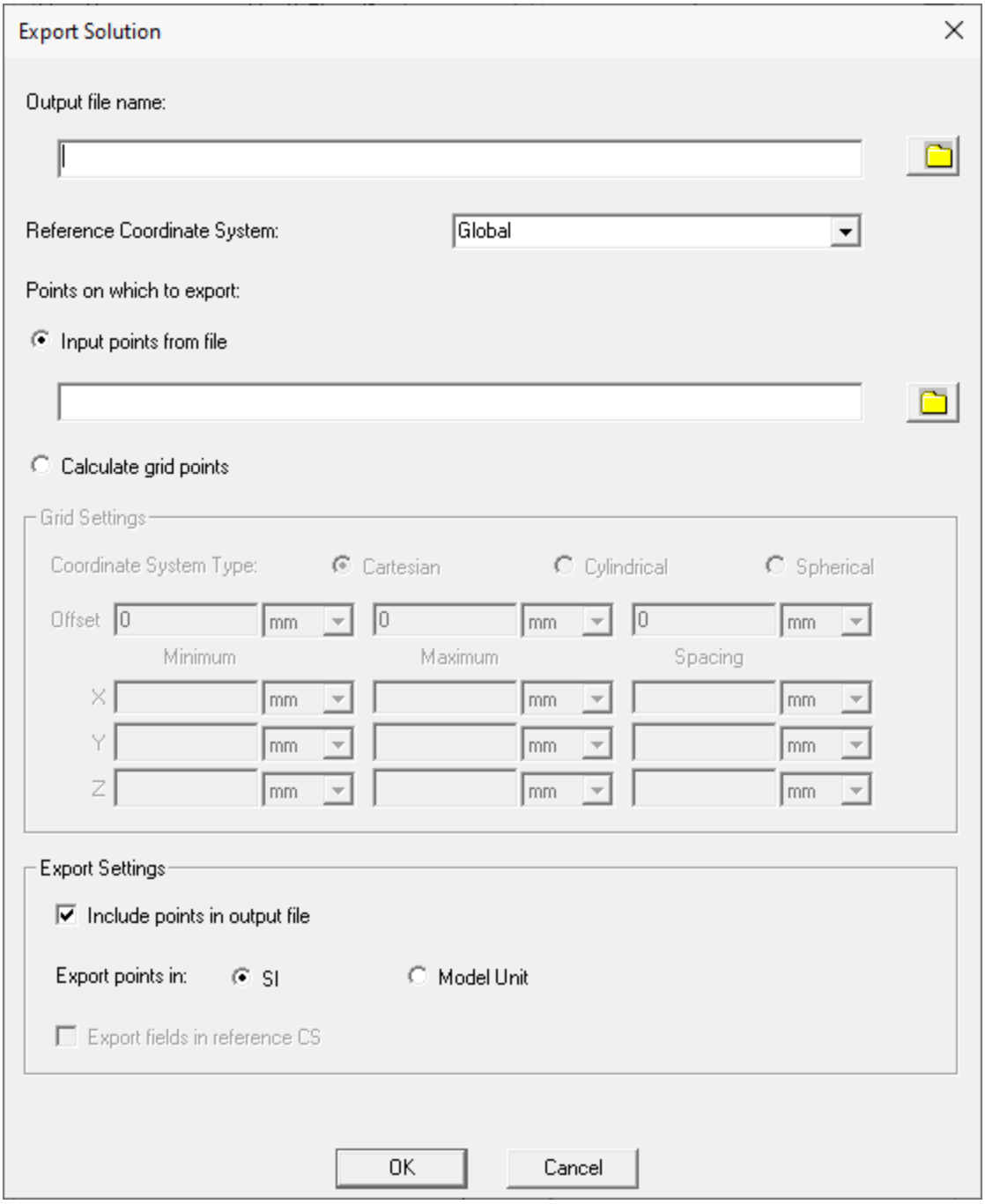
1. Load the quantity into the top register for the fields calculator, and perform any operations on it.
2. If desired, load a volume using the **Geometry** command.

   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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

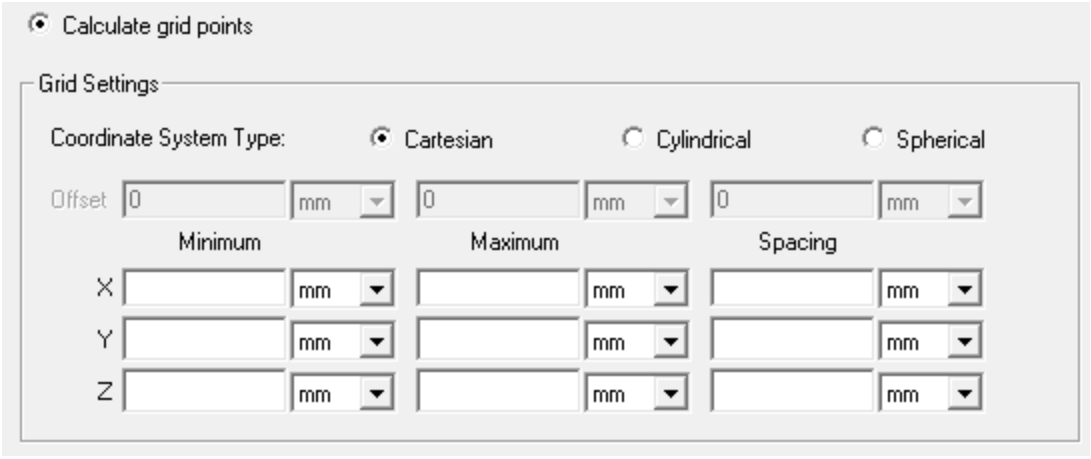
coordinate systems used and unitized in the design.



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 2026 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.

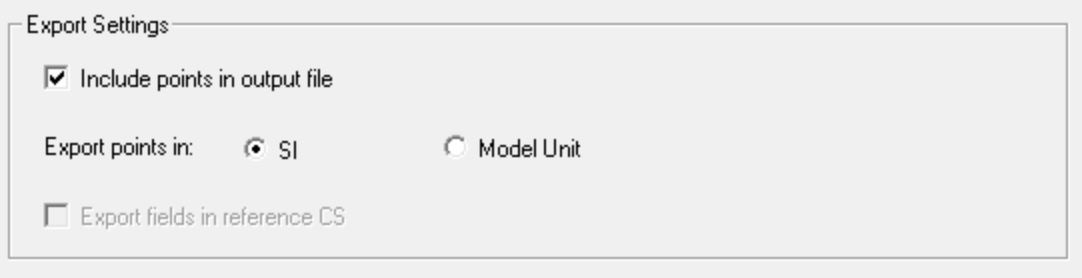
Cylindrical: for each dimension Rho, Phi, and Z, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

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.



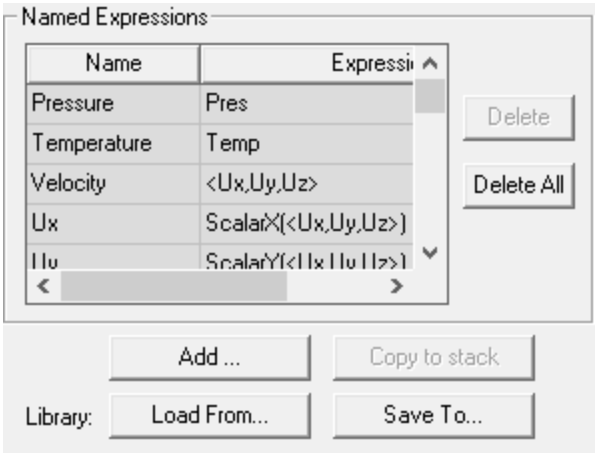
9. If field to be exported has type vector or complex vector, the 'Export Field in reference CS' checkbox is enabled and you can specify that field to be exported in reference CS. It can be helpful to evaluate the fields output when rotated coordinate systems are unitized in the design
10. Click **OK** to export the file.

    The field quantity is mapped to the grid and saved to the file you specified (**.fld** extension).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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 an Icepak design is open and a Solution Setup has been performed, the following predefined named expressions are available.

Pres	The relative static pressure of the fluid and is available when flow is computed.
Temp	Available when temperature is computed.
U	The x, y, or z component of the velocity vector and is available when flow is computed.
Heat_Flow_Rate	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

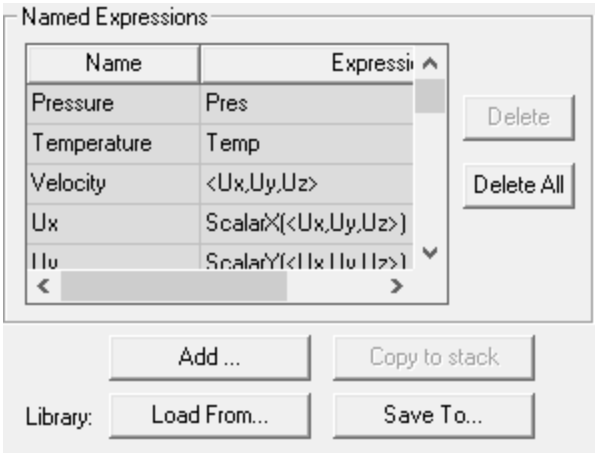
Heat_Flux	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.
Mass_Flux	Is the mass flow rate per unit area and can only be computed on the surface of model objects.
Surf_Pressure	The relative static surface pressure of the fluid and is available when flow is computed.
Radiation_Flux	The radiation heat flow per unit area and can only be computed on surfaces of objects. It is available if you are solving your problem with radiation effects.
Surf_Temp	The surface temperature and is available when temperature is computed. See Icepak Post Processing Quantities for more information.
Turbulent_Viscosity_Ratio	The ratio of the turbulent viscosity to the laminar viscosity. It can be reported if any of the available turbulence models are used.
Turbulent_YPlus	is a nondimensional parameter. See Icepak Post Processing Quantities for more information.
Turbulent_Kinetic_Energy	The turbulent kinetic energy. See Icepak Post Processing Quantities for more information.
Turbulent_Epsilon	The turbulent dissipation rate. See Icepak Post Processing Quantities for more information.
Heat_Transfer_Coeff	A derived scalar quantity computed from the temperature field. See Icepak Post Processing Quantities for more information.
Thermal_Cond_X, Y, and Z	The x, y, or z component of the orthotropic thermal conductivity vector and is available when traces are imported.
Speed	The magnitude of the velocity vector and is available when flow is computed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat_Radiative_Flow	A derived scalar quantity. It is available if you are solving your problem with radiation effects. It can be computed only on surfaces of objects.
Mass_Flow_Rate	A derived scalar quantity computed from the velocity field. See Icepak Post Processing Quantities for more information.

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 Icepak'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

To add a named expression of your own to the Fields Calculator list:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Exiting the Fields Calculator

Click **Done** to exit the Fields Calculator.

Ansys Electromagnetics Suite 2026 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 field quantities and displays the data in a table in the **Fields Summary** dialog.

> **Note:**
>
> The contents of this topic are specific to **Icepak** designs. For *Fields Summary* information specific to **IcepakFEA** designs, choose one of the following two links:
>
> - IcepakFEA – Thermal Solutions: Creating a Fields Summary
> - IcepakFEA – Structural Solutions: Creating a Fields Summary

1. From the **Icepak** menu, select **Fields**>**Create Fields Summary**. The **Fields Summary** and **Setup Calculation** dialog boxes are displayed.

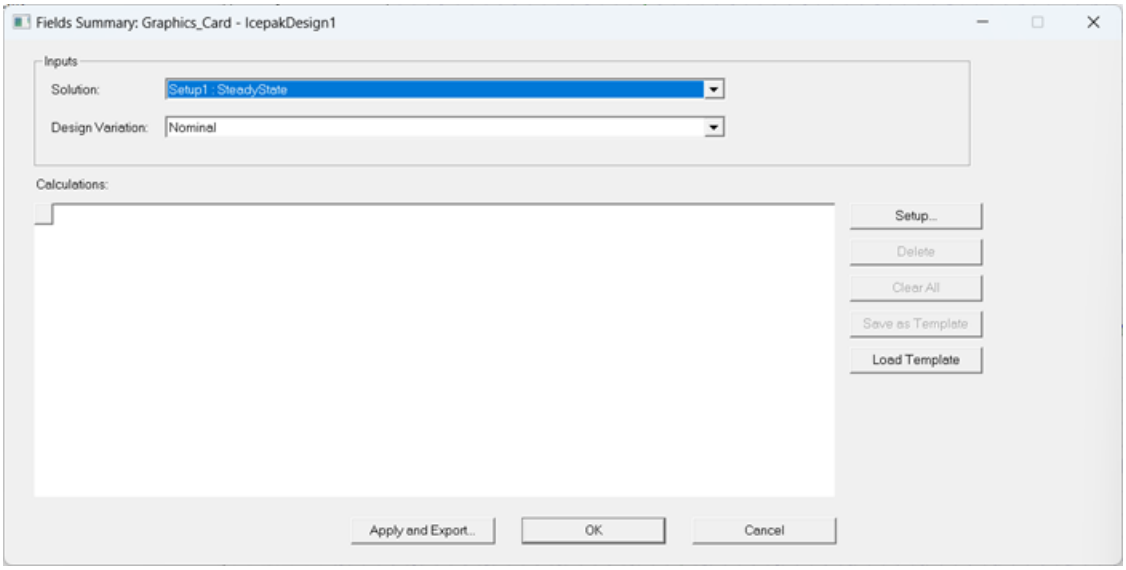
   > **Note:**
   >
   > Alternatively, right-click **Field Overlays** in the **Project Manager** and select **Create Fields Summary**.

   > **Note:**
   >
   > If you've previously created a fields summary template, you can click **Load Template** to select and load the template file.

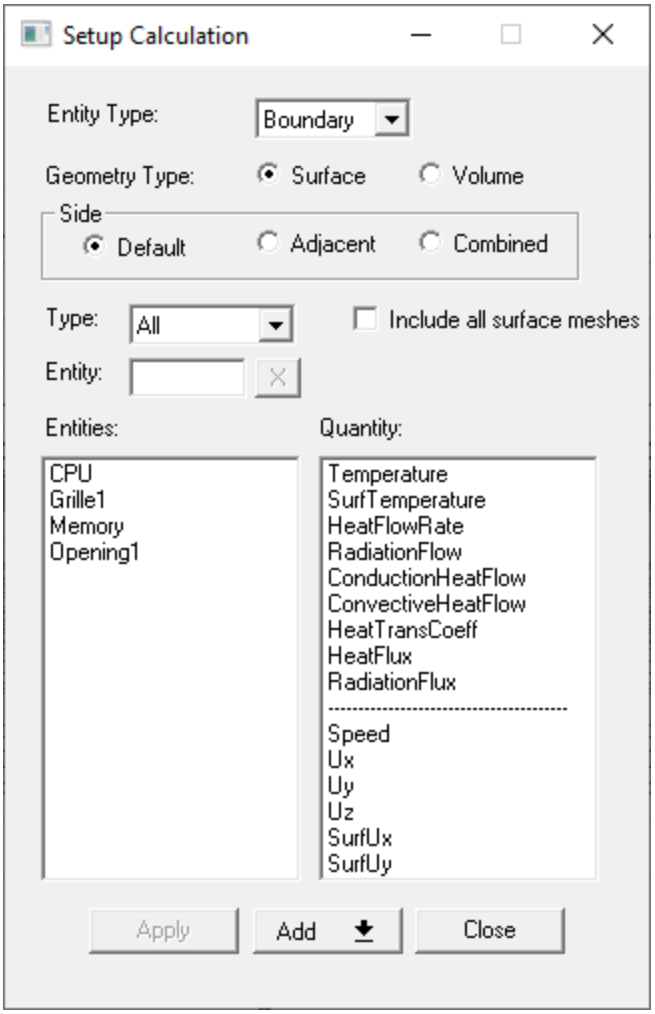
2. In the **Fields Summary** dialog under **Inputs**, select the **Solution** and **Design Variation** for which you want to create a fields summary. For **Design Variation**, you can select a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

single variation or **All**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the **Setup Calculation** dialog box, select the **Entity Type** and **Geometry Type**.
   - **Entity Type**: select **Boundary** (thermal boundary condition), **Object** (geometry), or **Monitor** (face or point monitor).

     > **Note:**
     >
     > When **Monitor** is selected, the entities displayed depends on the selected **Geometry Type**. When **Surface** is selected, face monitors are displayed. When **Volume** is selected, point monitors are displayed. Results for point monitors use the centroid of the selected object or face (not the mesh).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Geometry Type**: select **Volume** or **Surface**.

> **Note:**
>
> When **Volume** is selected:
>
> - Only thermal boundary conditions that can be assigned to a solid object are available (e.g., Block, Resistance, EM Loss).

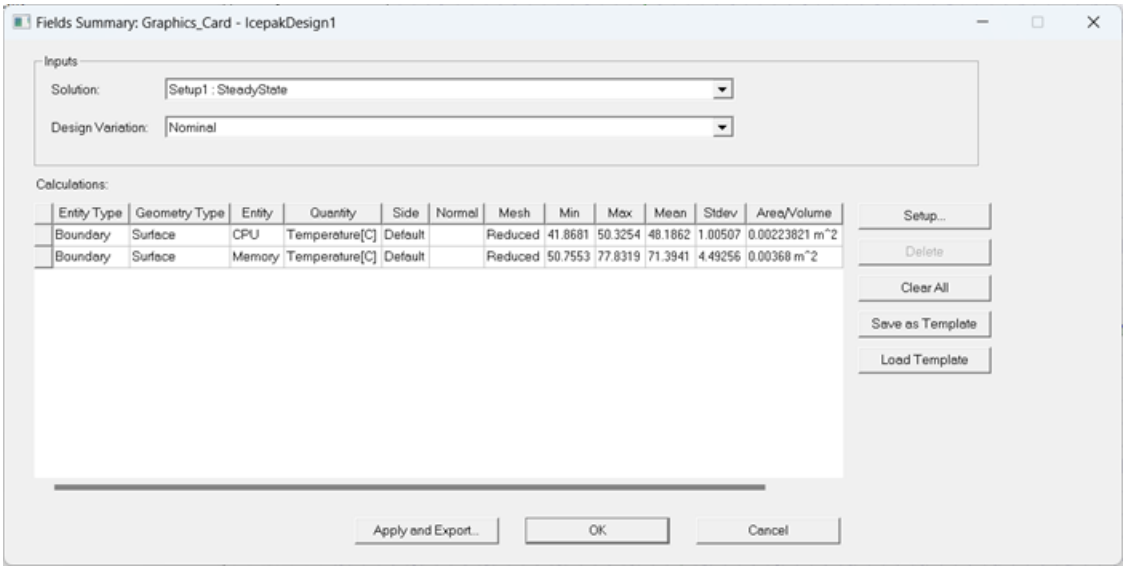
4. If needed, for 2D geometry or boundary conditions or the faces of 3D geometry or boundary conditions, select the **Adjacent side** check box. See Reporting Sides for information on the data displayed.
5. Select the **Entities** and **Quantity** you want to display in the fields summary. To filter the entities displayed, use the filtering options. For more information, see Filtering the Entities List.
   - **Entities**: select one or more entity from the list. Press and hold **Shift** or **Ctrl** to select multiple entities and quantities.

> **Note:**
>
> When **Volume** is selected:
>
> - Only quantities reported at nodes are available (e.g., Pressure, Temperature, RadTemperature, Ux, Uy, Uz).

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 row of the selected quantity for each selected entity.

> **Note:**
>
> When adding an entity list to a fields summary report as a single calculation, one line item is added. When adding one as multiple calculations, line items for each object or face in the entity list are added.

Ansys Electromagnetics Suite 2026 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.

7. If needed, click **Export** to open the **Save As** dialog and save a comma separated (.csv) file of the fields summary data in the desired directory.

SolutionName	Setup1 : SteadyState									
DesignVariation	Nominal									
Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev	Area/Volume
Object	Volume	ALPHA_MAIN_PCB	Temperature[C]	Default	-N/A-	20.3166	57.7065	32.3294	11.0919	2.81597e-05 m^3
Object	Surface	MEMORY1	Temperature[C]	Default	-N/A-	52.2261	57.8633	56.0828	1.58979	0.00184 m^2
Boundary	Volume	CPU	Temperature[C]	Default	-N/A-	41.9437	46.0303	44.7922	0.775189	1.52166e-06 m^3
Boundary	Surface	Opening1	Temperature[C]	Default	-1	20	20	20	0	0.0154 m^2

8. Do one of the following:

> **Note:** You can click **Save as Template** to save the current fields summary calculations as a template file.

- Click **OK** to save the settings and close the **Fields Summary** dialog box. When you open the **Fields Summary** dialog again, the data for the saved settings is displayed.
- Click **Cancel** to close the **Fields Summary** dialog without saving.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean and Standard Deviation Calculations

Mean

Surfaces

When calculating the mean of a selected quantity (e.g., Kx, Ky, or Kz) for surfaces, Ansys Icepak uses the following formula:

$$\frac{\sum(Qi*Ai)}{\sum(Ai)}$$

where

Ai = the area of element i of the selected geometry

Qi = the selected quantity value at element i of the surface

Volumes

When calculating the mean of a selected quantity (e.g., Kx, Ky, or Kz) for volumes, Ansys Icepak uses the following formula:

$$\frac{\sum(Qi*Vi)}{\sum(Vi)}$$

where

Vi = the volume of element i of the selected geometry

Qi = the selected quantity value at element i of the volume

Standard Deviation

Surfaces

When calculating the standard deviation of a selected quantity for surfaces, Icepak uses the following formula:

$$\frac{\sum(Qi*Qi*Ai)}{\sum(Ai)} - Mean*Mean$$

where

Ai = the area of element i of the selected geometry

Qi = the selected quantity value at element i of the surface

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volumes

When calculating the standard deviation of a selected quantity for volumes, Ansys Icepak uses the following formula:

$$\frac{\sum(Qi*Qi*Vi)}{\sum(Vi)} - Mean*Mean$$

where

Vi = the volume of element i of the selected geometry

Qi = the selected quantity value at element i of the volume

Reporting Sides

When reporting on 2D geometry or boundary conditions or the faces of 3D geometry or boundary conditions, which side (outward normal direction) of the geometry or boundary condition reported depends on the data quantity selected and whether or not the **Adjacent side** check box in the **Setup Calculation** dialog box is enabled.

The following table lists which side of a face is reported for 3D object faces, 2D sheet geometry, and a face selection with one planar face.

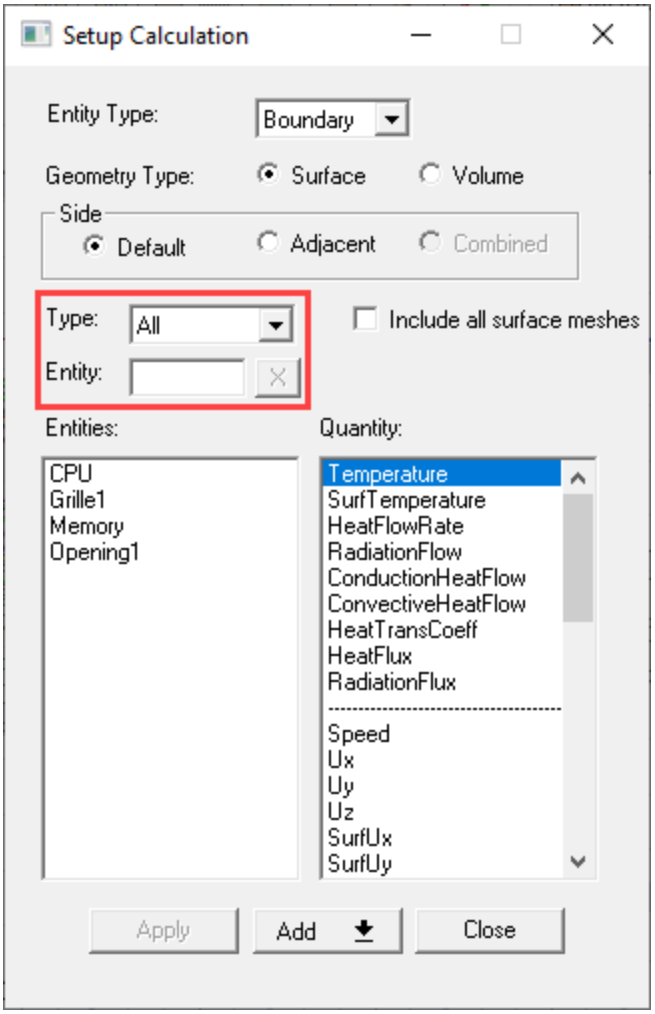
Quantities	Adjacent Side	Reported Side
SurfTemperature, SurfVelocity, Surface Ux, Uy, Uz	Disabled	Opposite direction of the face normal.
SurfTemperature, SurfVelocity, Surface Ux, Uy, Uz	Enabled	Direction of the face normal.
SurfPressure, HeatFlowRate, HeatFlux, RadiationFlux, Heat TransferCoeff	Disabled	Direction of the face normal.
SurfPressure, HeatFlowRate, HeatFlux, RadiationFlux, Heat TransferCoeff	Enabled	Opposite direction of the face normal.

> **Note:**
>
> If a face selection contains more than one face or only one non-planar face, normal is not applicable and displayed as **N/A**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filtering the Entities List

When creating a fields summary with numerous boundary conditions, object geometries, and monitors, you can use the filtering options in the **Setup Calculation** dialog box.



Type

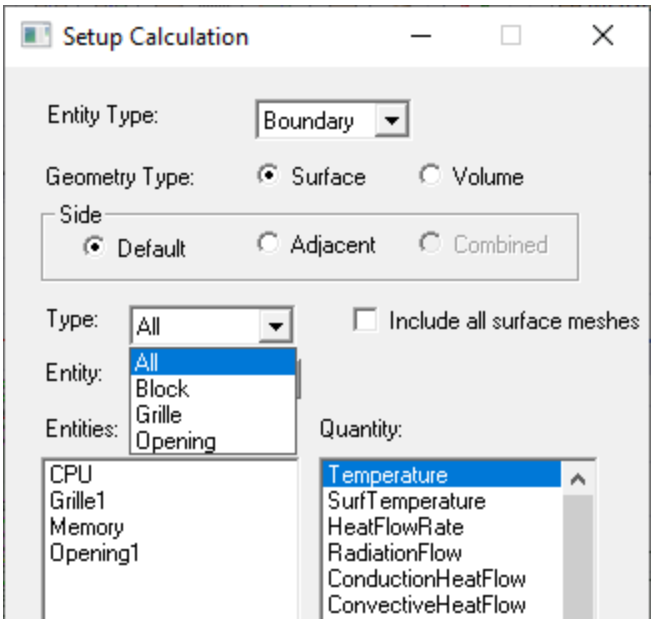
From the **Type** drop-down list, select an option. The options vary based on **Entity Type** and **Geometry Type** settings.

Boundary

When the **Entity Type** is set to **Boundary**, assigned boundary conditions are displayed in the **Entities** list. When the **Geometry Type** is set to **Surface**, boundary conditions assigned to 2D

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

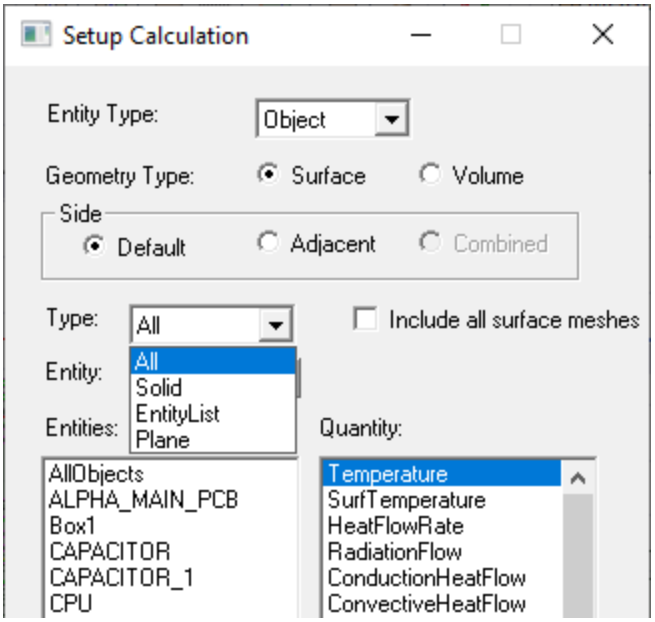
and 3D geometry are displayed. When **Volume** is set, only boundary conditions assigned to 3D geometry are displayed. Use the **Type** drop-down list to filter by boundary condition type.



Object

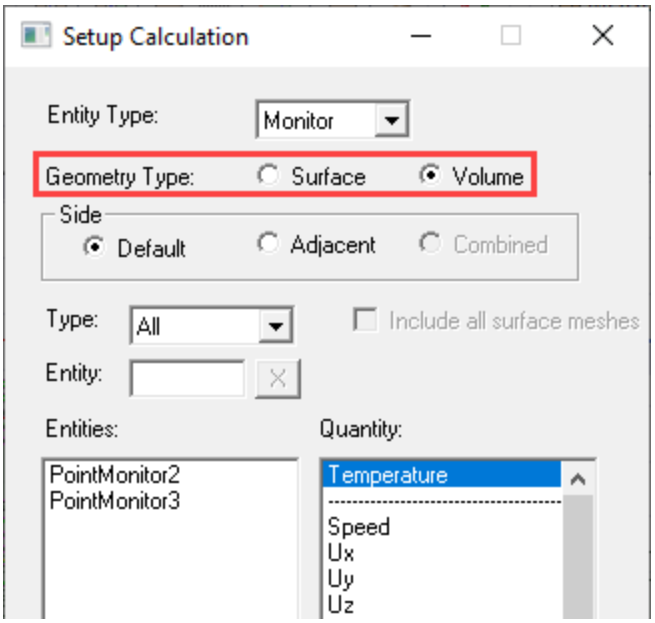
When the **Entity Type** is set to **Object**, objects are displayed in the **Entities** list. When the **Geometry Type** is set to **Surface**, 2D and 3D objects are displayed. When **Volume** is set, only 3D objects are displayed. Use the **Type** drop-down list to filter by object type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Monitor

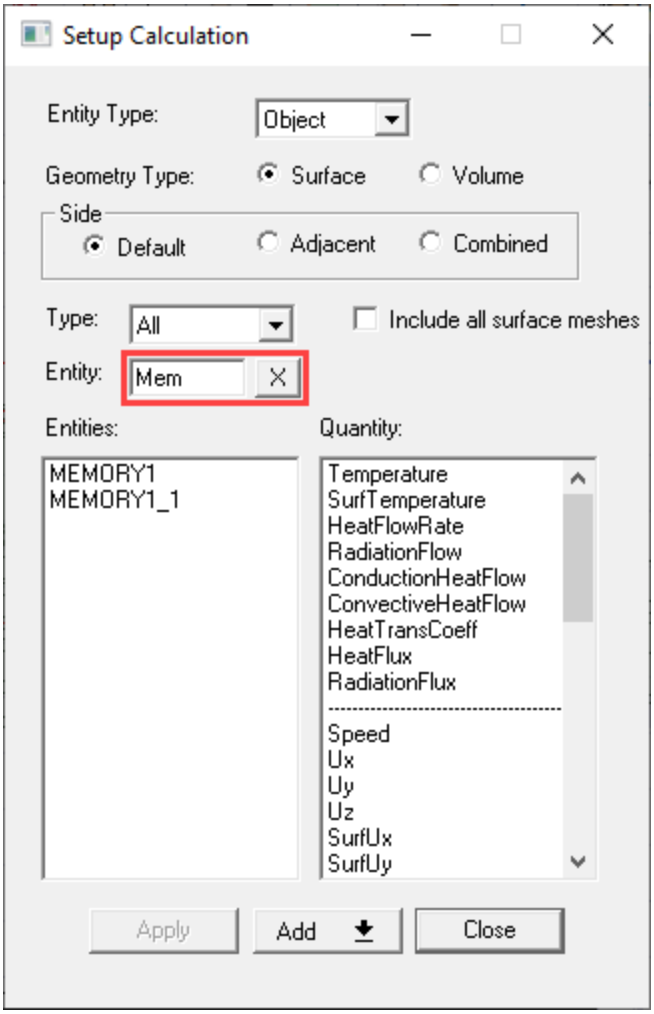
When the **Entity Type** is set to **Monitor**, face or point monitors are displayed in the **Entities** list. When the **Geometry Type** is set to **Surface**, face monitors are displayed. When **Volume** is set, point monitors are displayed.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity

In the **Entity** field, you can enter text to further filter the entities displayed. For example, in the following image, typing *Mem* displays the two memory objects.



Exporting Fan Operating Point Data

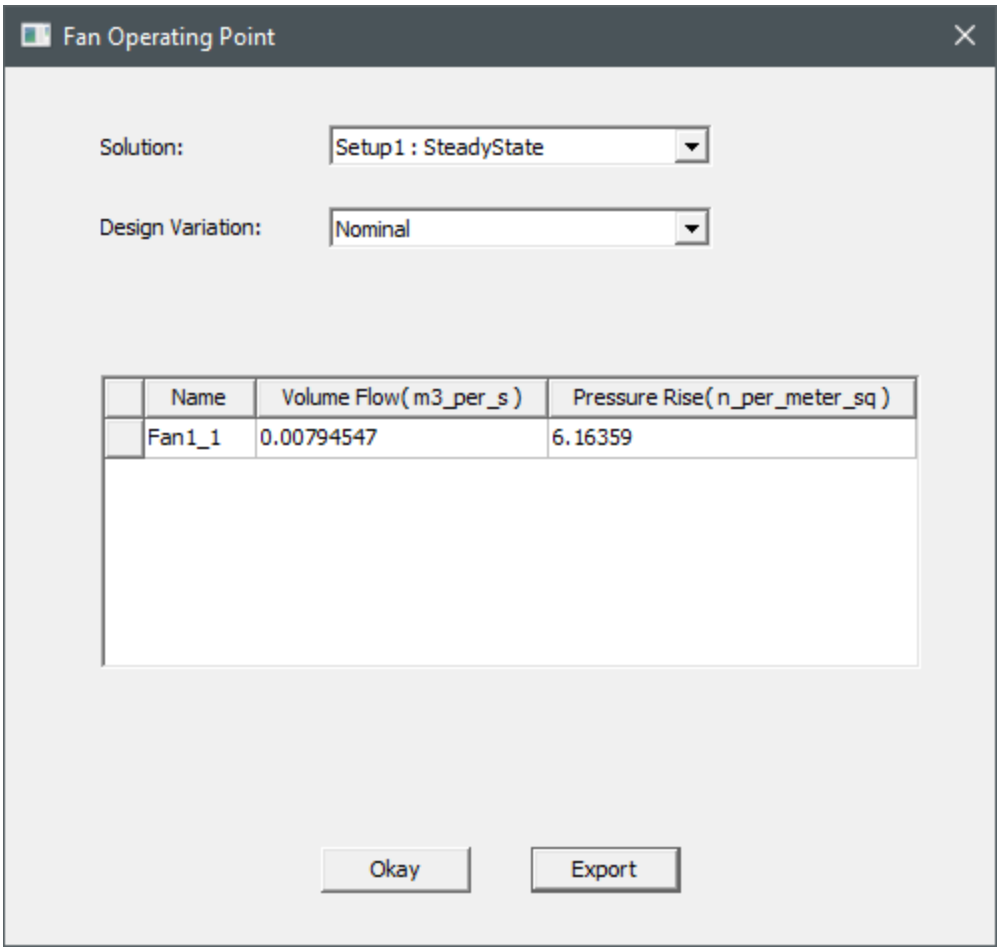
For models with a Fan component that is defined by a fan curve dataset, you can export the fan operating point data (in .csv format). The Fan operating point data reports the volume flow rate and pressure rise for the characteristic curve of fans in your model.

1. From the **Icepak** menu, select **Fields**>**Fan Operating Point**. The **Fan Operating Point** dialog box is displayed.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Alternatively, right-click **Field Overlays** in the **Project Manager** and select **Fan Operating Point**.

2. In the **Fan Operating Point** dialog under **Inputs**, select the **Solution** and **Design Variation** for which you want to export the fan operating point data.



3. Click **Export** to open the **Save As** dialog.
4. Save the file in the desired location. An example of the data is shown below.

Solution Name: Setup1 : SteadyState		
Design variation:		
Fan Instances	Volume Flow (m3_per_s)	Pressure Rise (n_per_meter_sq)
Fan1_1	0.00794547	6.16359

5. Click **Okay** to close the **Fan Operating Point** dialog box.

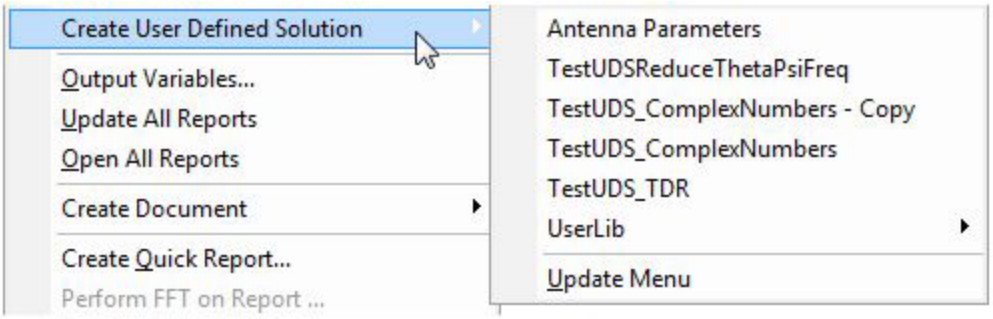
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



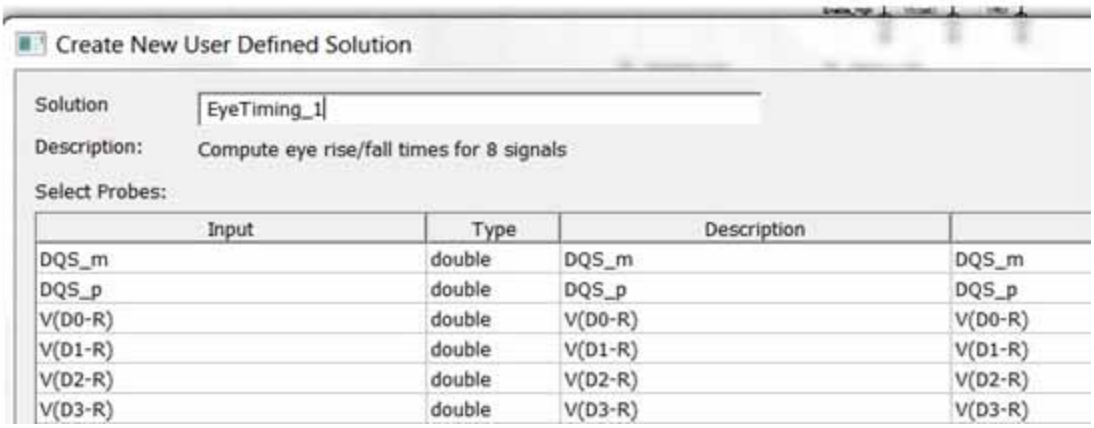
Named Probes and Properties in User Defined Outputs

For more information on user-defined outputs (UDOs), please visit the User-Defined Outputs (UDOs).

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.

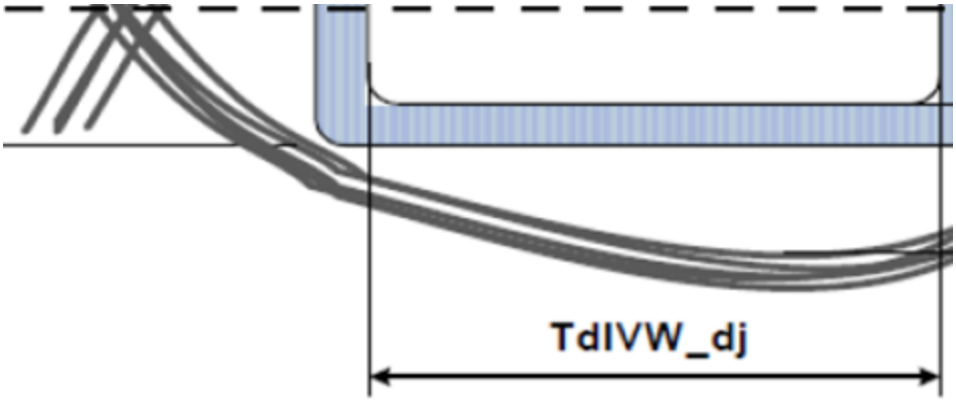
For example, you could create a user-defined solution called EyeTiming_1.

Ansys Electromagnetics Suite 2026 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.

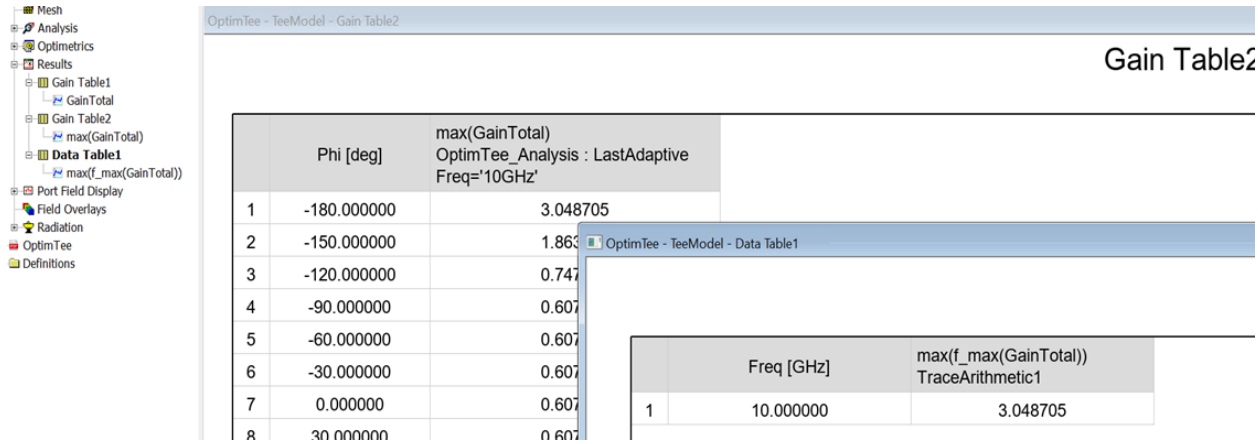


In another example, you could create a user-defined solution called **UDS Distance Trace Arithmetic Result1**:

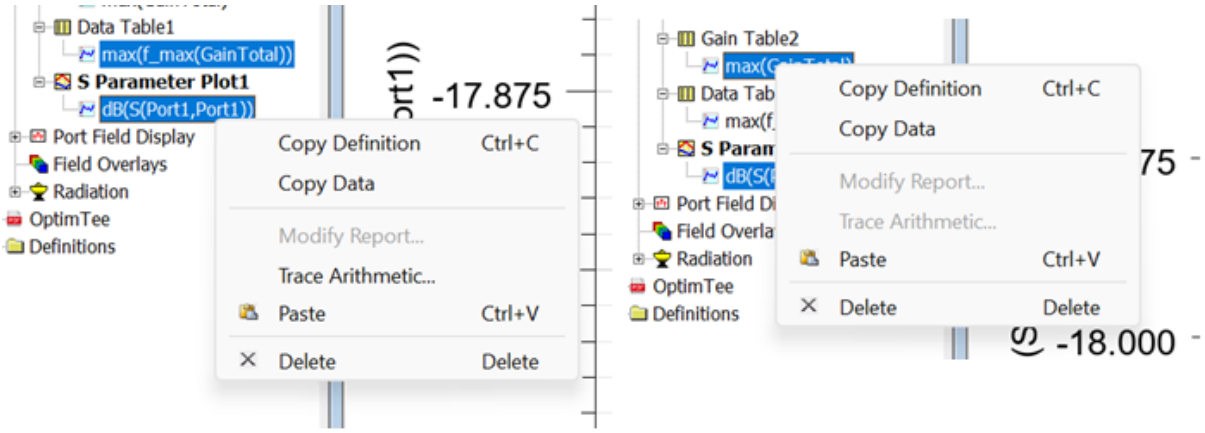
Then access this solution in the **Create User Defined Solution** menu:

The **Trace Arithmetic** feature is built into the User Defined Output capability to allow quantities from multiple Solution Contexts to be combined into a single User Defined Solution (and outputs probe data without modification). Previously, sweep reduction functions were only applied to values to the primary sweep within a trace; with the Trace Arithmetic feature, reductions across sweet are now achievable.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

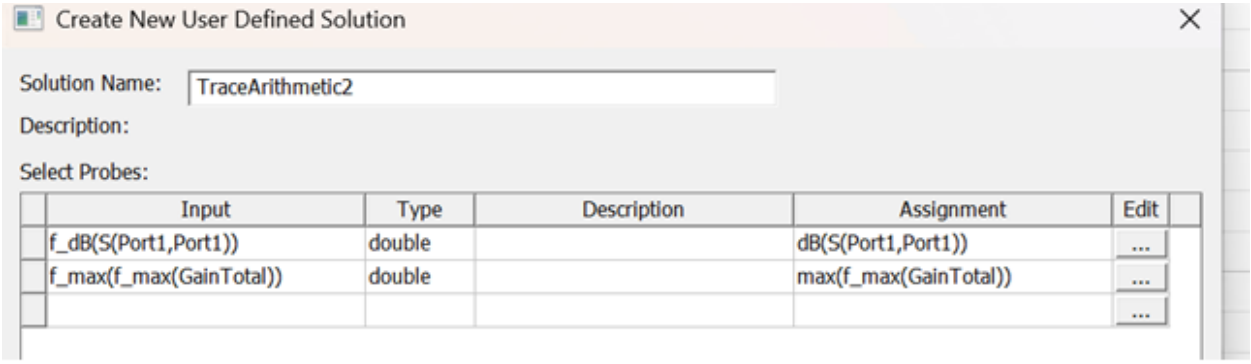


To create a Trace Arithmetic solution, select one (or more) trace and then right click on the selected trace to open the **Trace Menu** window. The Trace Arithmetic menu item is enabled if the selected traces have identical sweep sets.

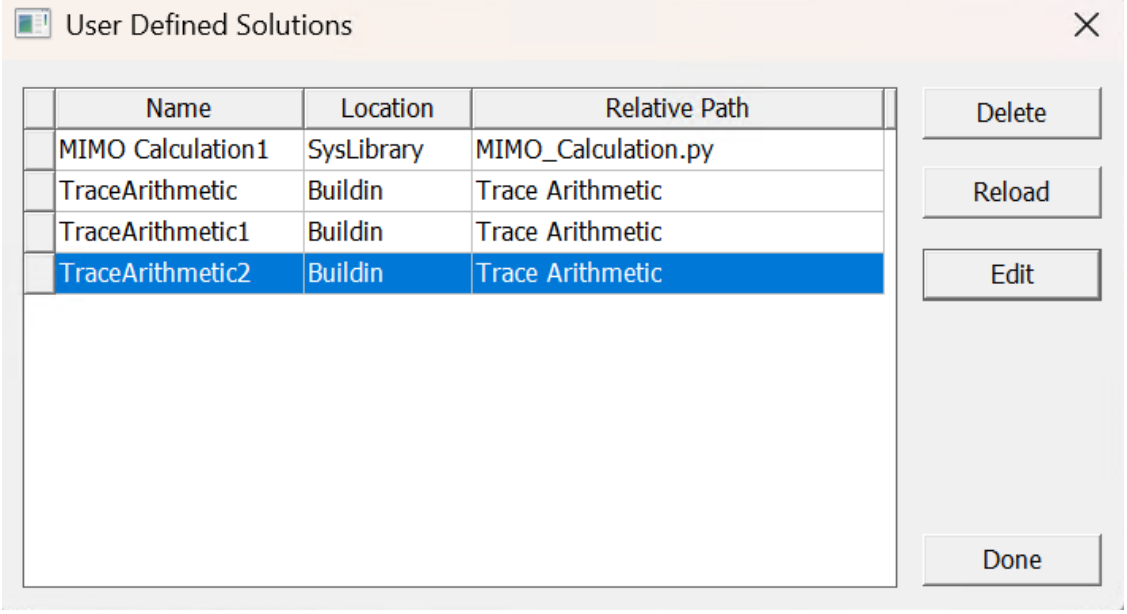


Clicking the Trace Arithmetic item in the **Trace Menu** window brings up the **Create User Defined Solution** window that show the selected traces as the input probes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

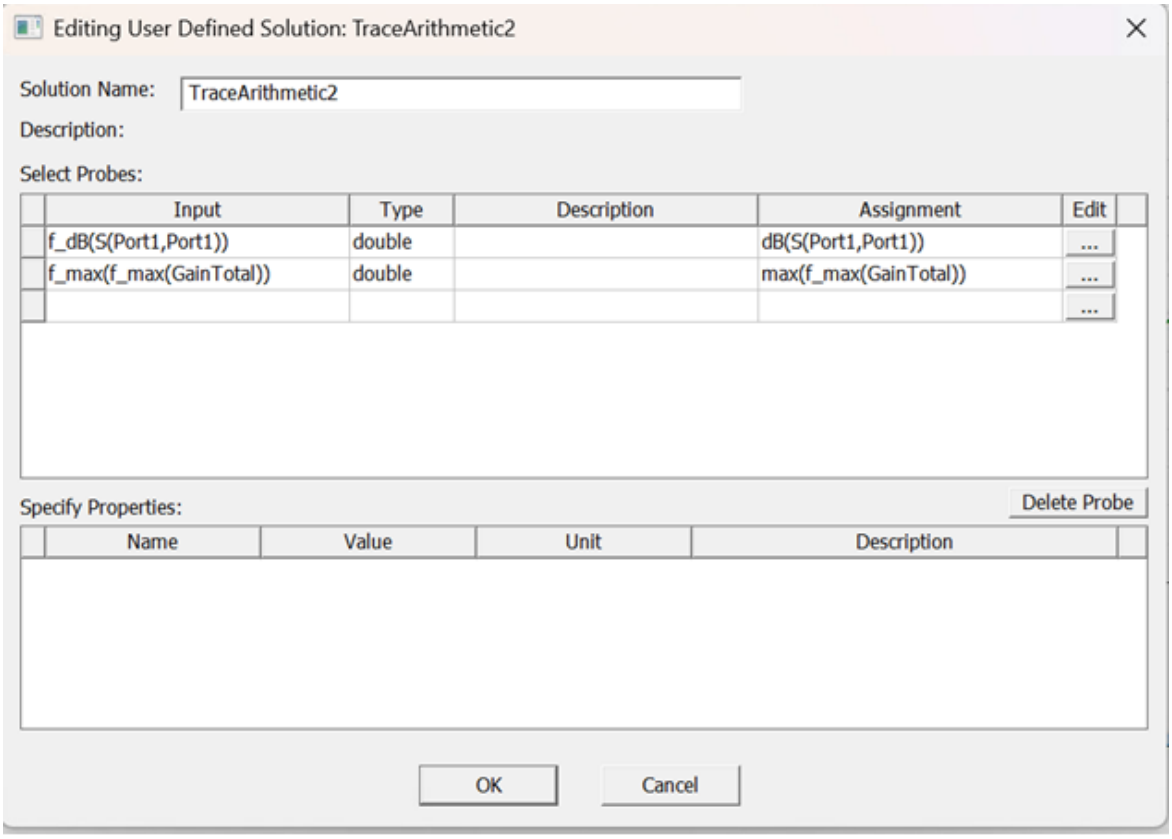


As a Trace Arithmetic solution is a built-in UDS, it can be listed, viewed, and/or edited as a UDS.



Unlike a traditional UDS, a Trace Arithmetic probe can be deleted or added, and the input probe name is editable. Click any of the "..." radial buttons under the **Edit** column to edit or add the assignment expression.

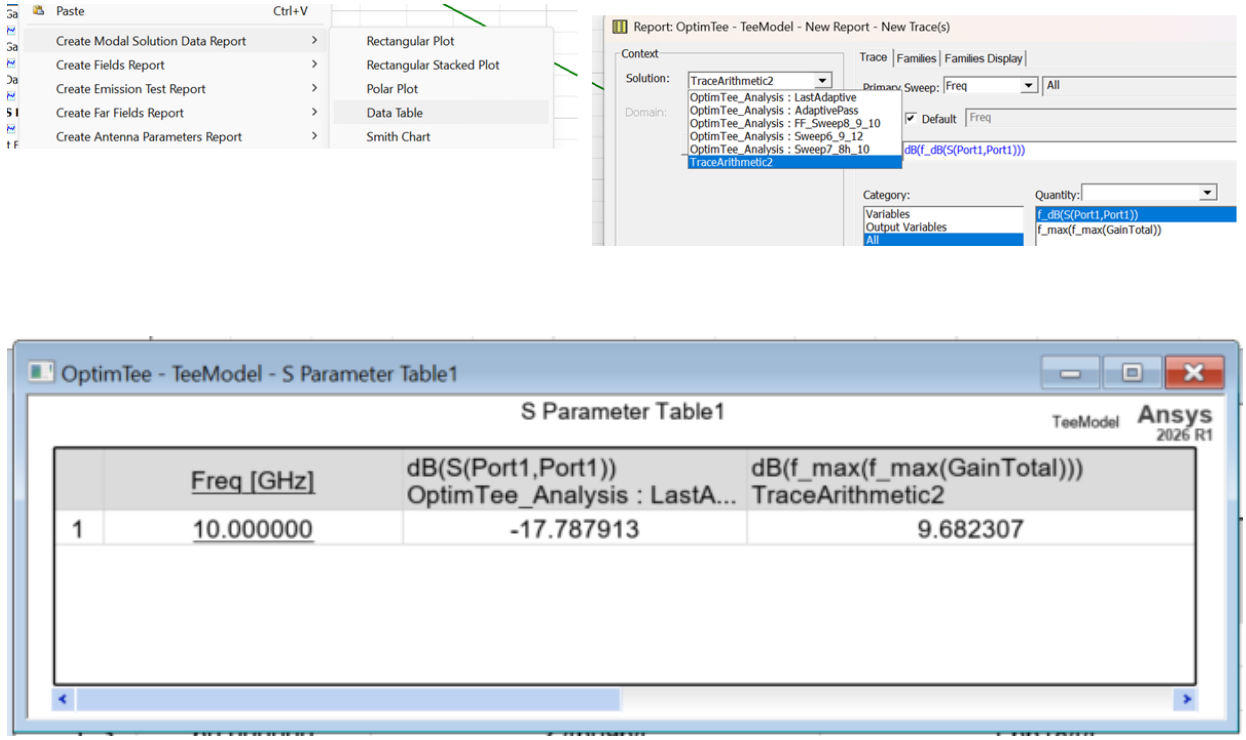
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A probe assignment can be an expression using a quantity from import solution, user defined solution, and any report type solutions that have the same sweep set as the first probe.

To create a report/trace using Trace Arithmetic solution, create the trace using the Trace Arithmetic solution from the report type of the first probe of the Trace Arithmetic solution.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Trace Arithmetic solution maintains the same set of sweep variables as the input probes. For the first three sweeps:

- Sweep values are unions within the range intersection of their respective probes ([0, 15, 30, 45, 60, 75], [-10, 0, 10, 20, 30, 40, 60] -> [0, 10, 15, 20, 30, 40, 45, 60]}
- Missing quantity values are calculated by interpolating between neighboring values ([1.5, 2. 3.2, 5, 6, 5] -> [1.5, 1.75, 2, 2.6, 3.2, 3.6, 5, 6], [0, 2,1, 2, 5, 3, 6] -> [2, 1, 1.5, 2, 5, 3, 3.75, 6])

For the fourth sweep and up:

- Sweep values are taken directly from the command values of their probes ([0,15,30,45,60, 75], [-10, 0, 10, 20, 30, 40,60] -> [0,30])
- Extra quantity values are removed ([1.5, 2, 3.2, 5, 6, 5] -> [1.5, 3.2], [0, 2, 1, 2, 5, 3, 6] -> [2, 5])

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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

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 …
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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 []`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Syntax	GetCategoryNames
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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`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 <*quantityName*>	Type String	Description Name of quantity
Return Value	Object of type QuantityInfo.		

Python Syntax	GetQuantityInfo(<*quantityName*>)

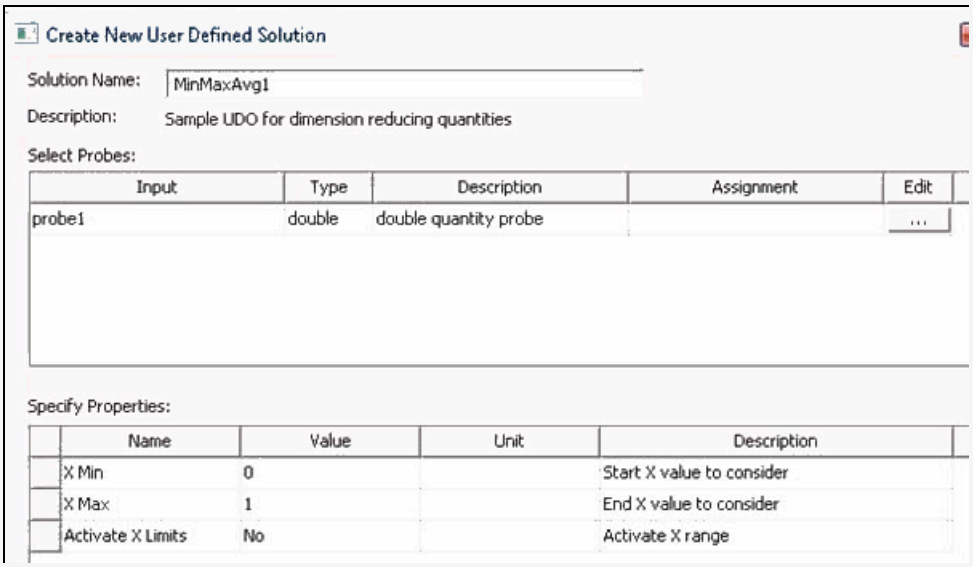
Python Example	`# Returns an instance of QuantityInfo for the qtyName supplied or None if such a`
	`# quantity could not be found`
	`def GetQuantityInfo(self, qtyName):`
	`  # All the quantities we have are simple doubles`
	`  # we can leave them unitless`
	`  return QuantityInfo(Constants.kDoubleParamStr)`

GetInputUDSParams

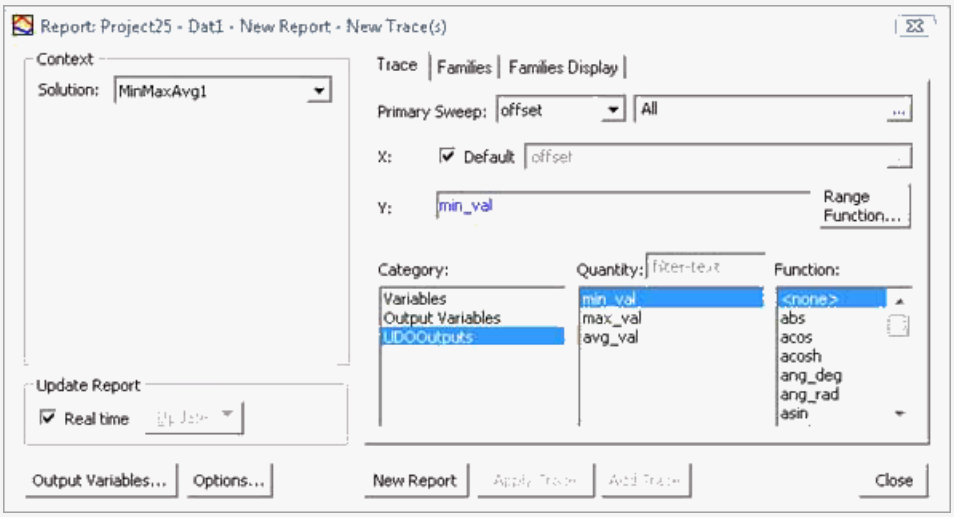
This is the main definition method of the UDO. The supplied arguments populate details of the parameters to which the UDO user will specify value, specify the probe names and their types as well as the dynamic probe selections.

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 2026 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 *<udsParams>*	Type List< UDSProbeParams >	Description 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			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 method.
	< *userSelectionForDynamicProbes* >	List< UDSProbeParams >	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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  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"

  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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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	Type	Description
	<inData>	IUDSInputData	Used to get the input probe data.
	<outData>	IUDSOutputData	Used to set the UDO solution quantity and sweep data.
	<propList>	IPropertyList	Used to get the user entered values for each of the properties defined during the GetInputUDSParams call.
	<progressMonitor>	IProgressMonitor	This can be used to set progress for long running calculations, check for 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`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  # 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
    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
```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
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
```

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
- 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*>)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.
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		

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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):`
	`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	`# sweepNames is C# Array of strings`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example	`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()
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:`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	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 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 <*sweepName* > <*sweepData*>	Type String List<floats>	Description Name of the sweep 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*>)
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)`

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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*>)

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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.
- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

- 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',
```


Ansys Electromagnetics Suite 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 i 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 ecosystem. 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`

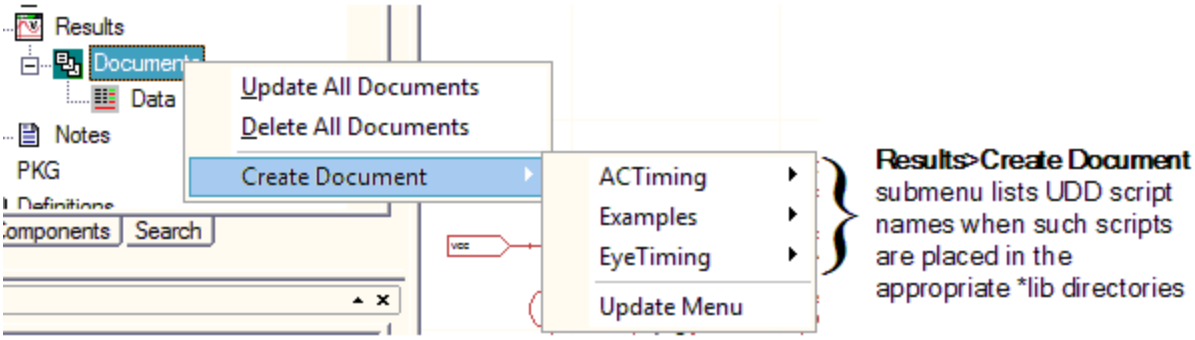
If for some reason you cannot place the .NET assemblies into a Lib directory under *UserDefinedOutputs*, you need to do a couple more steps before step 3 listed above.

```
Import sys
sys.path.append("full path to your .NET assembly location")
```


Ansys Electromagnetics Suite 2026 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.

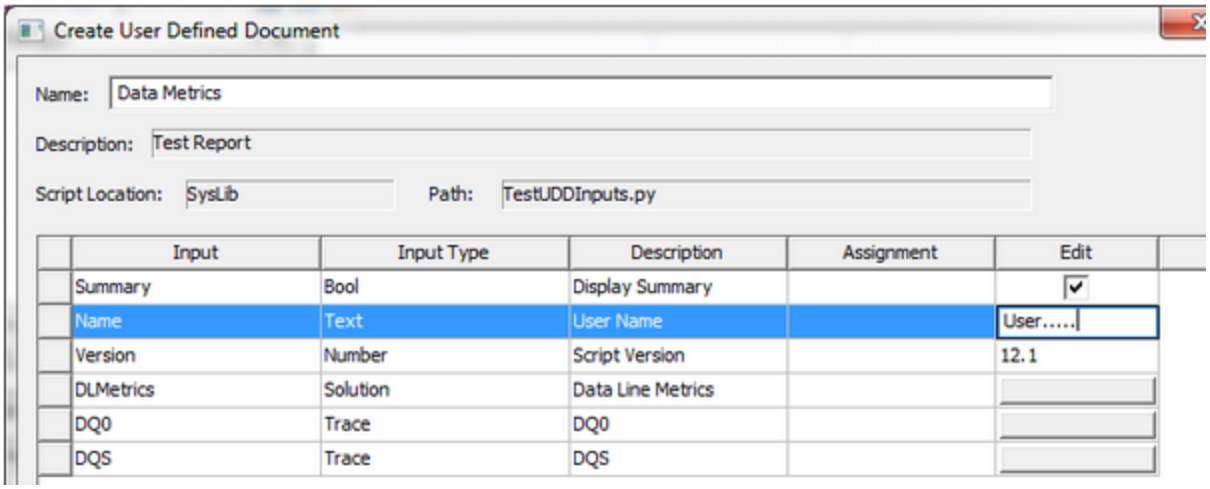
Use **Icepak** > **Results** > **Create Document** > **Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib, or Personallib, or to exclude them if they have been deleted after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments, or Personallib/UserDefinedDocuments are available through the **Icepak** > **Results** > **Create Document** menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create User-Defined Document Inputs

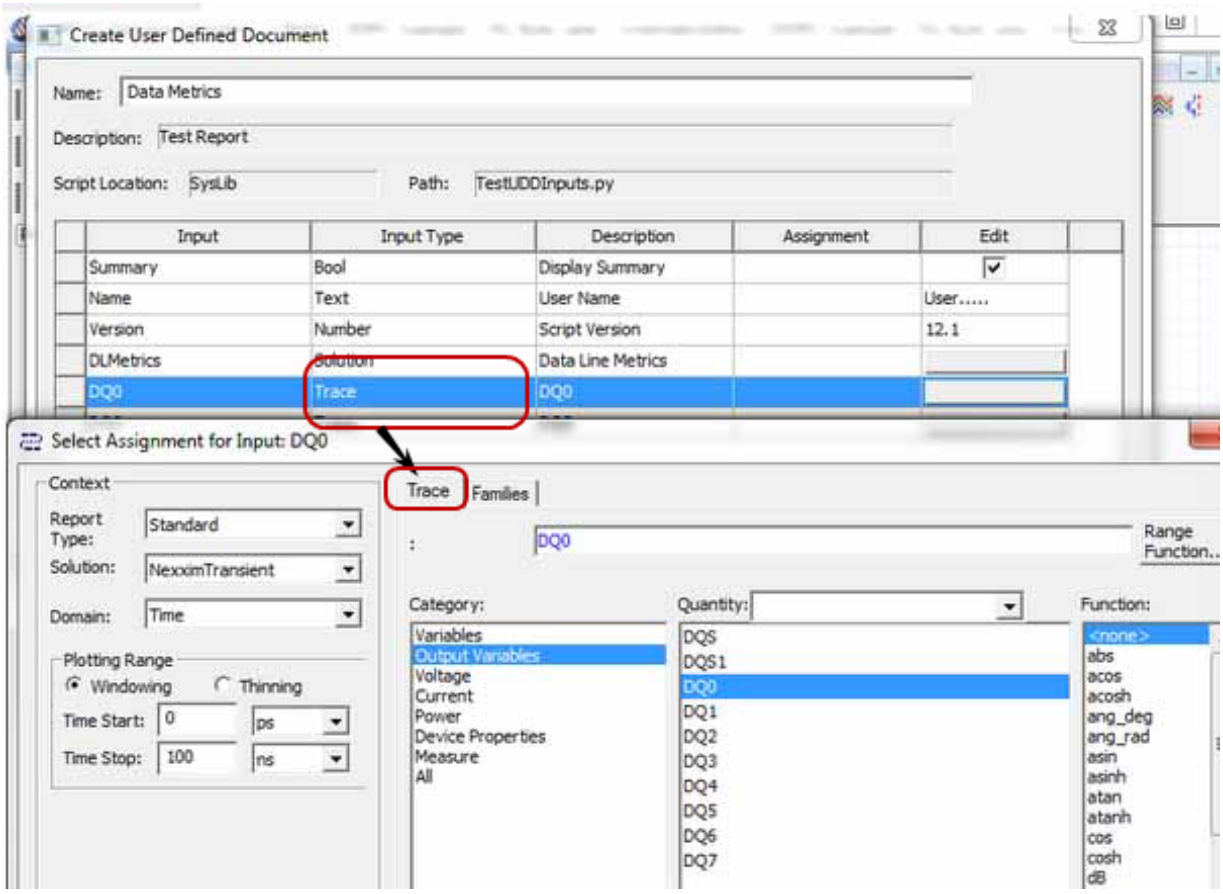
User-defined documents allow data from traces, solutions, and report types as inputs. A UDD can specify the named inputs, which you select or enter the values of in the *Create User Defined Document* dialog box that displays when you run **Icepak** > **Results** > **Create Document** > <*scriptName*>.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

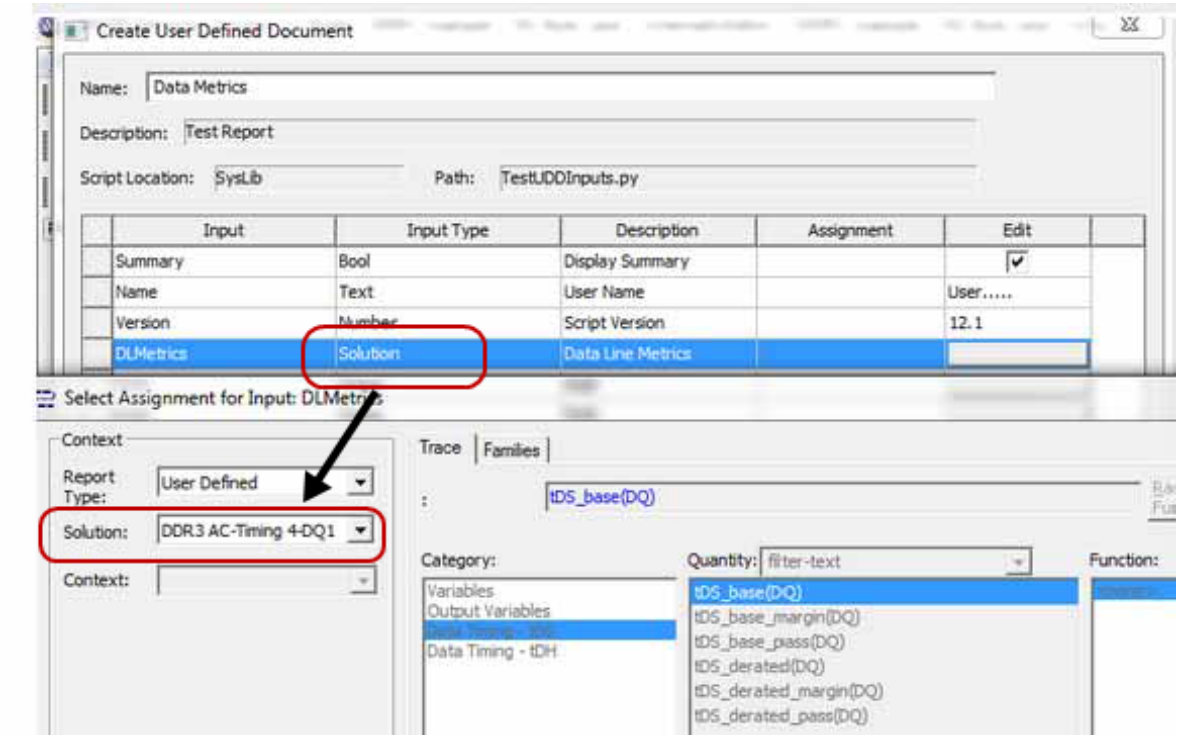


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety is now available to the user and can be accessed from the Python script.

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 box is disabled for user input.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

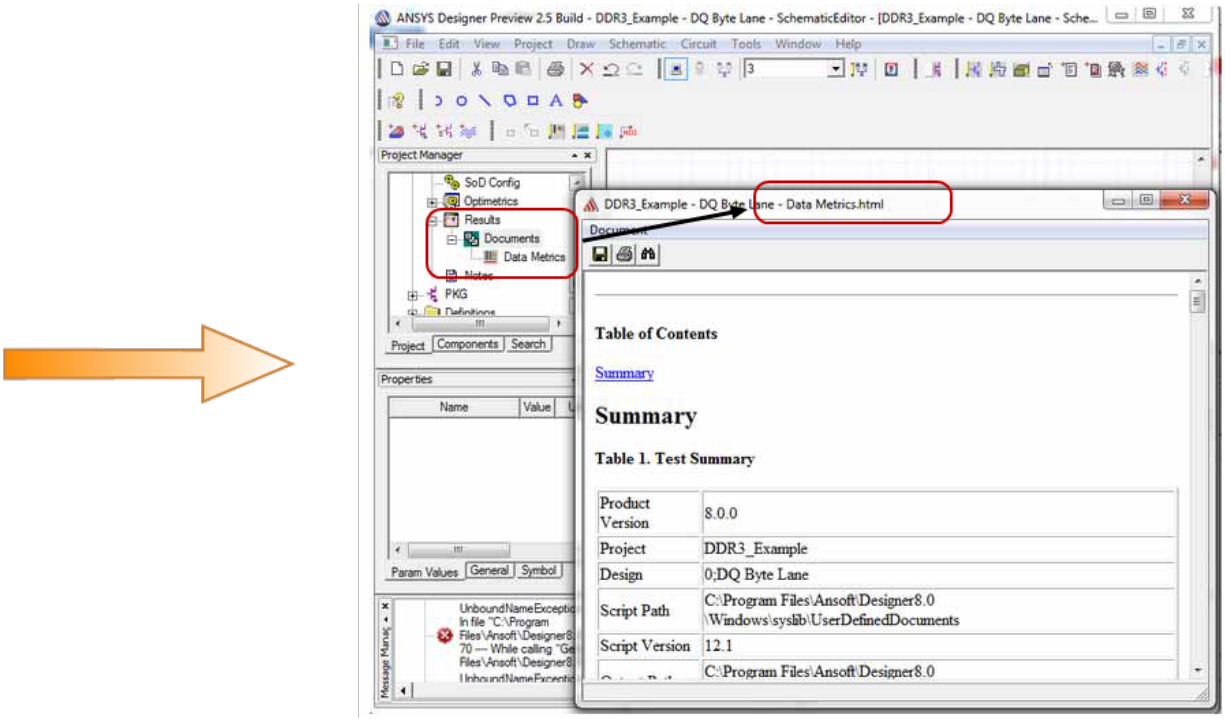


UDD Document Creation and Display

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.

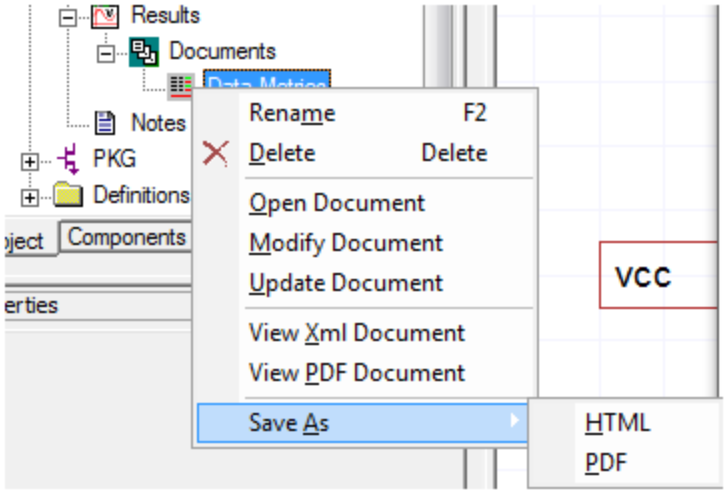
The created document will be placed under a new folder named **Documents** under the **Results** folder. All documents that are created by the user for the design will be placed under this folder.

Ansys Electromagnetics Suite 2026 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

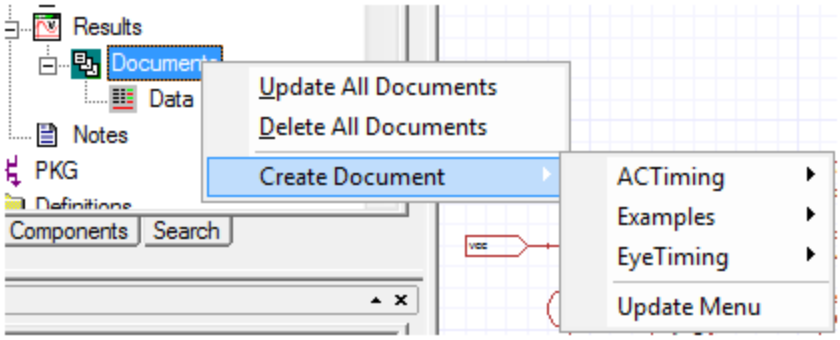
Right-click a user-defined document displayed in the Project Manager tree to bring up a menu where you can rename, delete the document. **Open document** opens the web browser with the html document. **Modify document** opens the setup dialog box, where you can change the selections for the input. To 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Documents Folder's Context Menu

The Documents folder has the right-click menu options to **Update All Documents** or **Delete All Documents**. It also provides the option of creating a document from here.

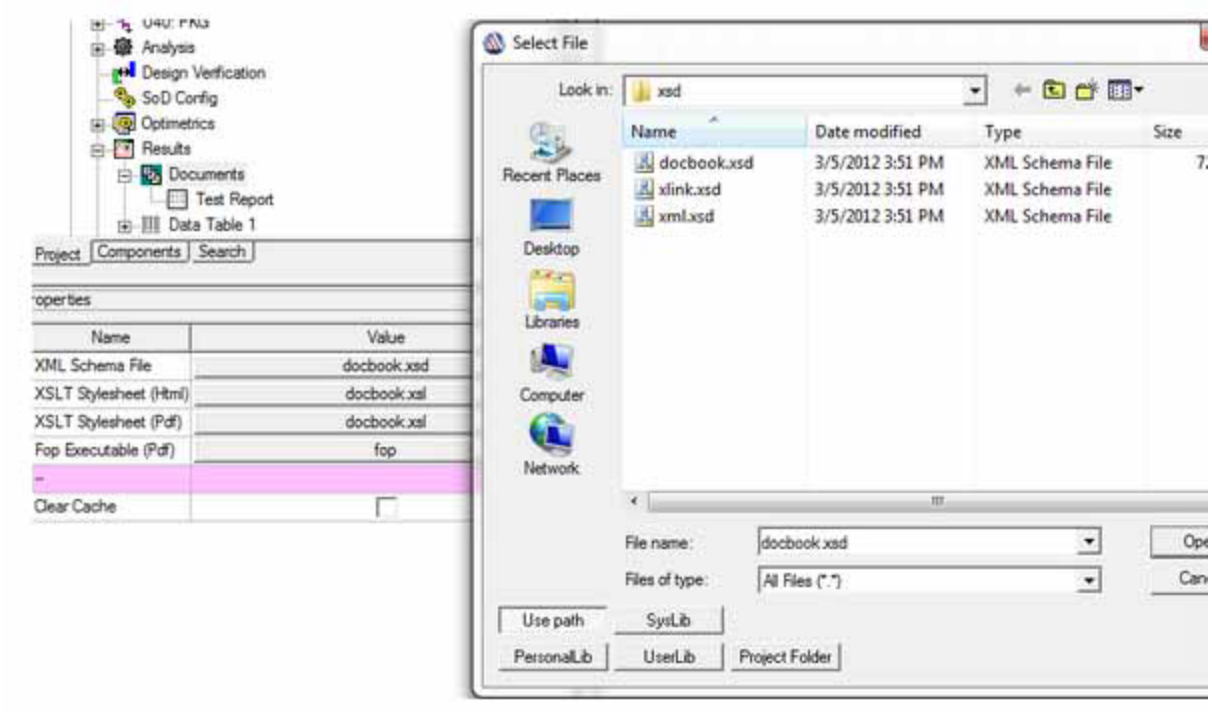


Document Folder's Property Window

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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

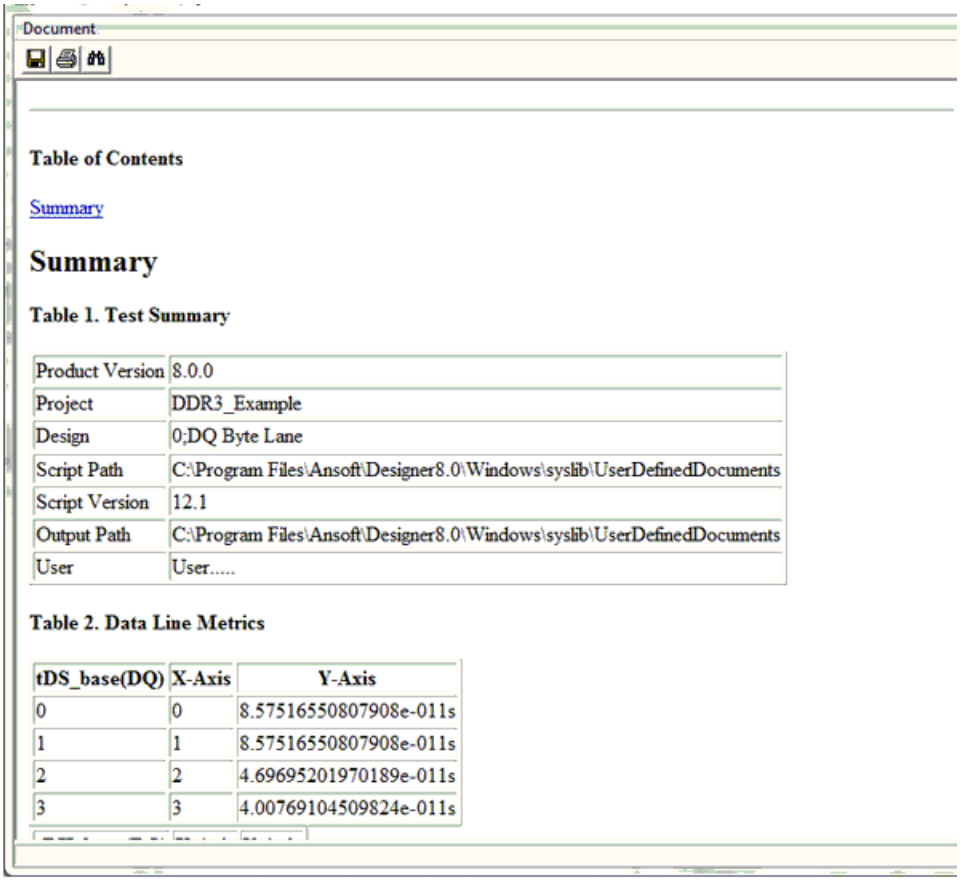

Document

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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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.

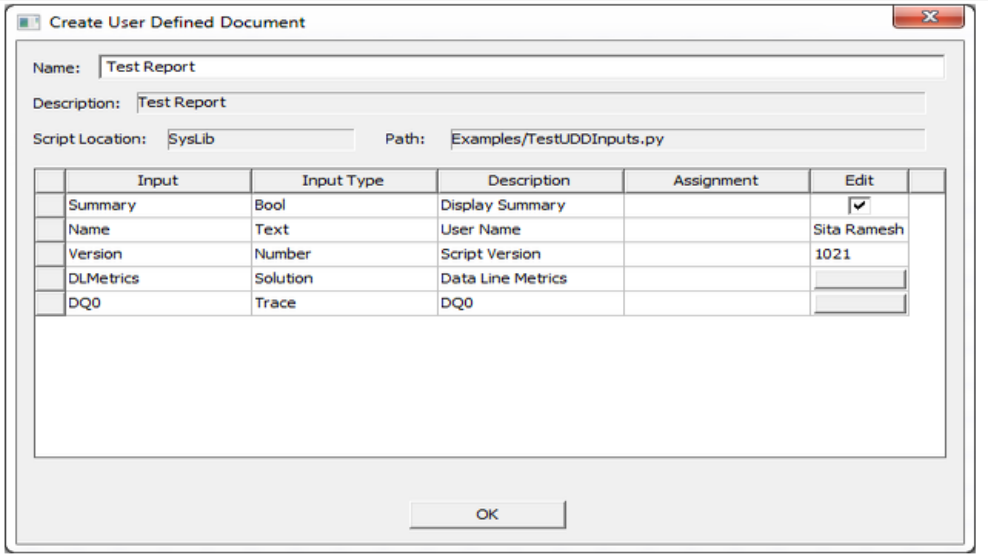
Python Syntax	ShowDefaultSetupDialog()
Python Example	`def ShowDefaultSetupDialog(self):` `  return True`

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 2026 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 **Icepak**> **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 2026 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 2026 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 2026 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 2026 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 2026 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\\v261\\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 2026 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\\v261\\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\\v261\\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 2026 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\\v261\\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 2026 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 2026 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 2026 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 2026 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 2026 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 2026 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()
```

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:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- oApp = self.GetUDDAppContext()

  Gets the application context. Use this interface to get the active project ad the version of the product.

  ```
  oDesktop = oApp.GetAppDesktop()
    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();
     ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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")
     ```
  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")
     ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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:

  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)
     ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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")
     ```
  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; it 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; it 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. It 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. It takes a column index, row index and a text string and takes in a row number because a table row can have multiple sub rows.
     ```
     IUDDTextElement trow1.Add(int column, int subrow, string
     text)
     ```


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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)
   ```
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 two columns; can be cell content or header text. It 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. It 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)
    ```

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 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18 - Transient Simulations

Icepak can solve the equations for conservation of mass, momentum, and energy in time-dependent form. Thus Icepak can be used to simulate a wide variety of time-dependent phenomena, including transient heat conduction and convection.

Activating time dependence is sometimes useful when attempting to solve steady-state problems that tend toward instability (for example, natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

Icepak uses a fully-implicit time-integration scheme for transient analysis. This chapter provides details about setting up a transient simulation in Icepak and post-processing the results.

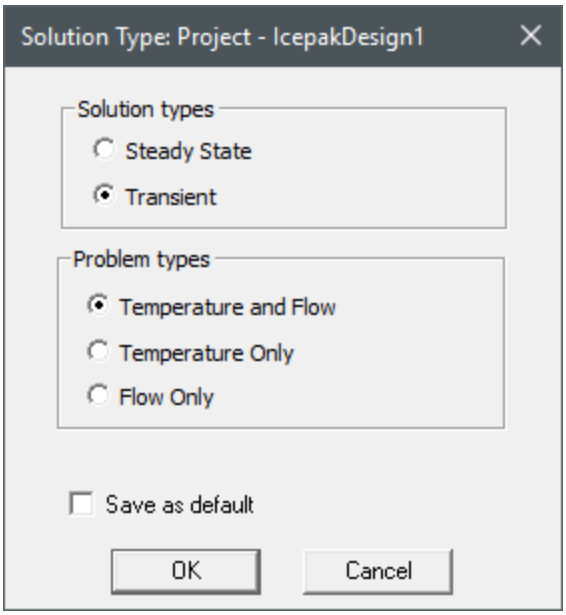
> **Note:**
>
> See Getting Started with Icepak - Transient Simulation for a workflow related to this feature.

Selecting the Transient Solution Type for Icepak

1. Click **Icepak** > **Solution Type**.

   The **Solution Type** dialog box appears.

2. Under **Solution types**, select **Transient**.

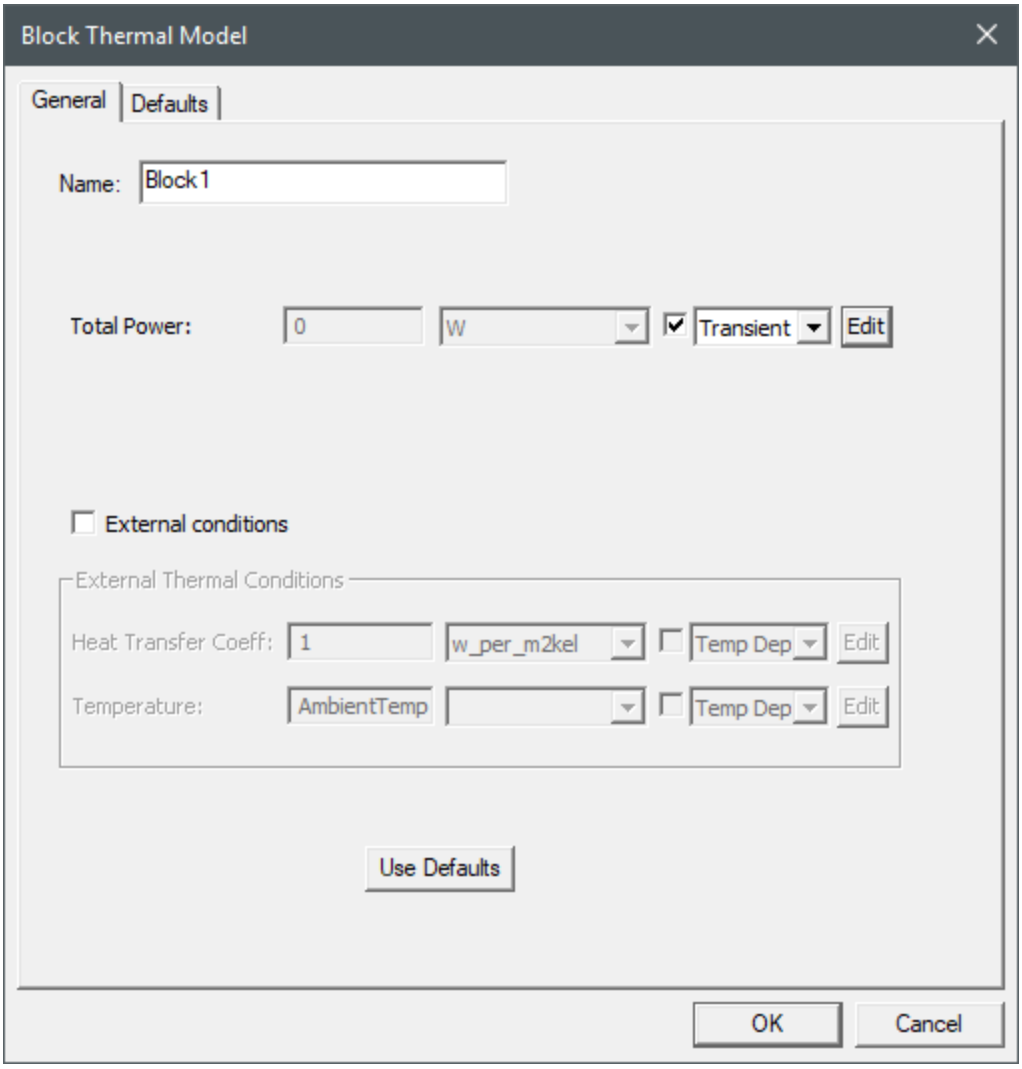


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

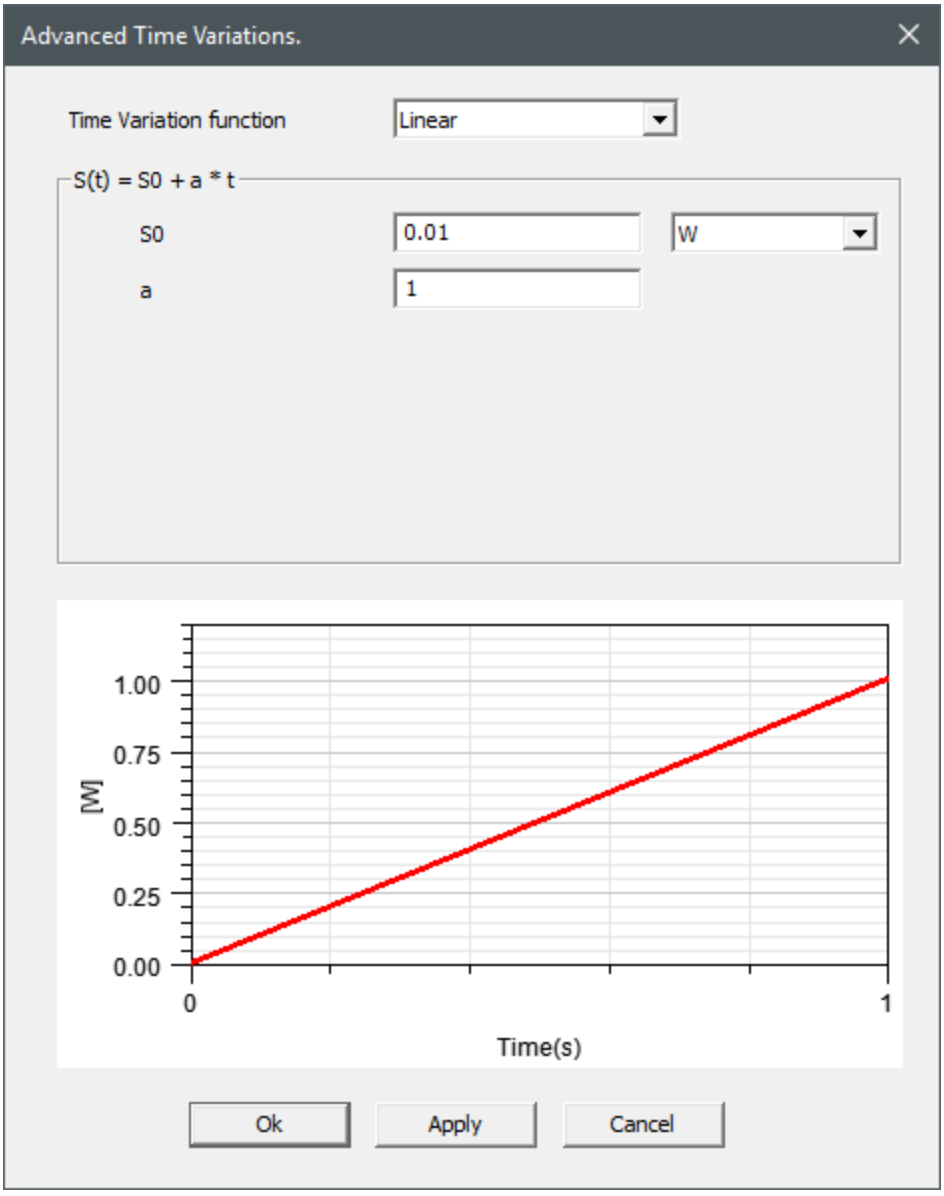
3. Under **Problem types**, select one of the following:
    - **Temperature and Flow**: For solving the energy equation for the temperature distribution together with the flow equations.
    - **Temperature Only**: For solving the energy equation for the temperature distribution only.
    - **Flow Only**: For solving the problem for flow only.

Specifying Parameters as a Function of Time

For transient simulations, you can specify ambient temperature (in Design Settings) and certain thermal boundary condition parameters vary as a function of time. You can define different types of variations, including linear, power law, exponential, sinusoidal, piecewise linear, and square wave. For example, in the block boundary condition, you can specify the **Total Power** parameter as **Transient** and click **Edit** to open the **Advanced Time Variations** dialog box.



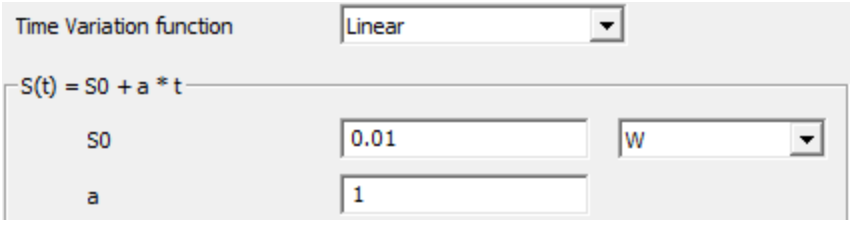
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Linear

$s_t = s_0 + at$

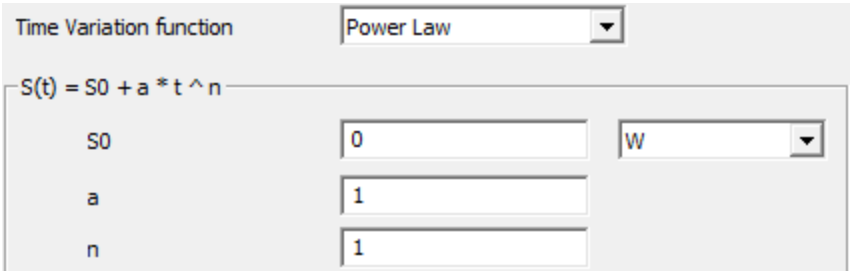
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



where t is the time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0, and a is a constant.

Power Law

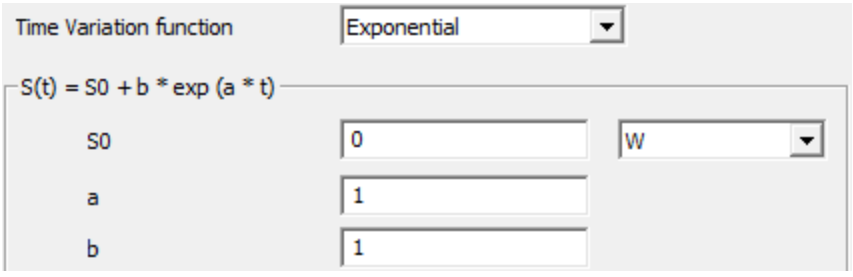
$s_t = s_0 + at^n$



where t is the time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0, and a and n are constants.

Exponential

$s_t = s_0 + be^{at}$

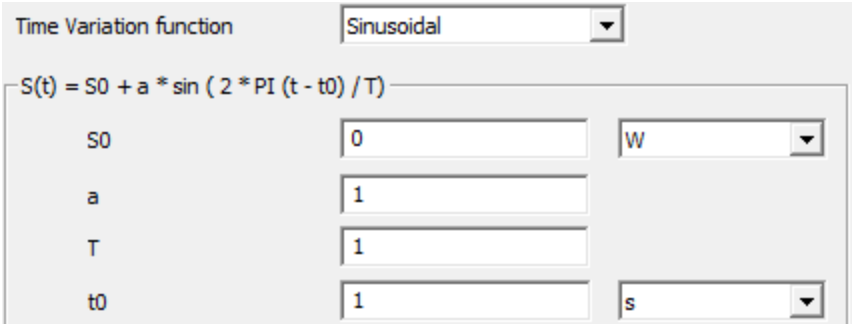


where t is the time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0, and a and b are constants.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sinusoidal

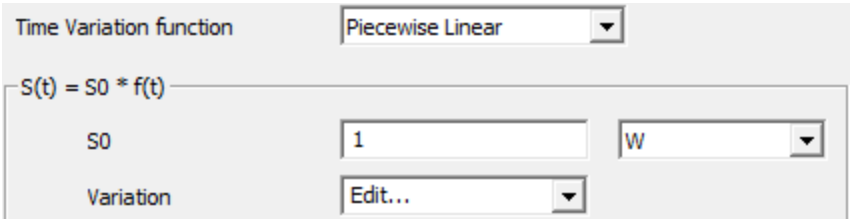
$$s_t = s_0 + asin\left[\frac{2\pi}{T}(t - t_0)\right]$$



where t is the time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0, T is the period, t_0 is the phase shift, and a is a constant.

Piecewise Linear

$$s_t = s_0 f(t)$$

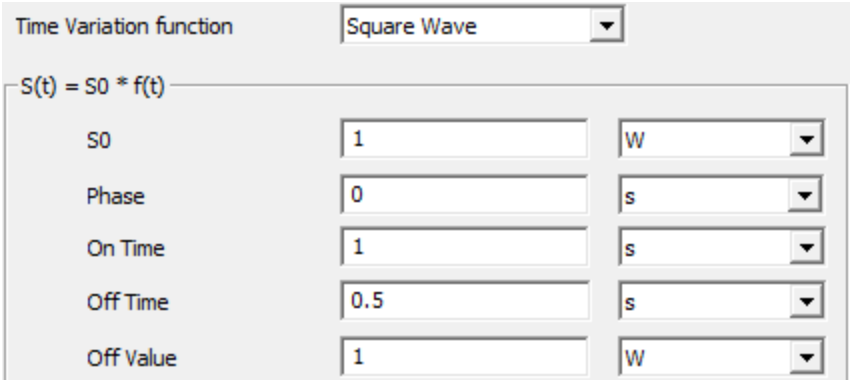


where $f(t)$ is a function of time, s_t is the value of the variable at time t, s_0 is the multiplier value applied to the variable parameter defined in the piecewise linear dataset (y-axis value). See Using Datasets to Define Transient Parameters for information on defining $f(t)$ using the **Add Dataset** dialog box.

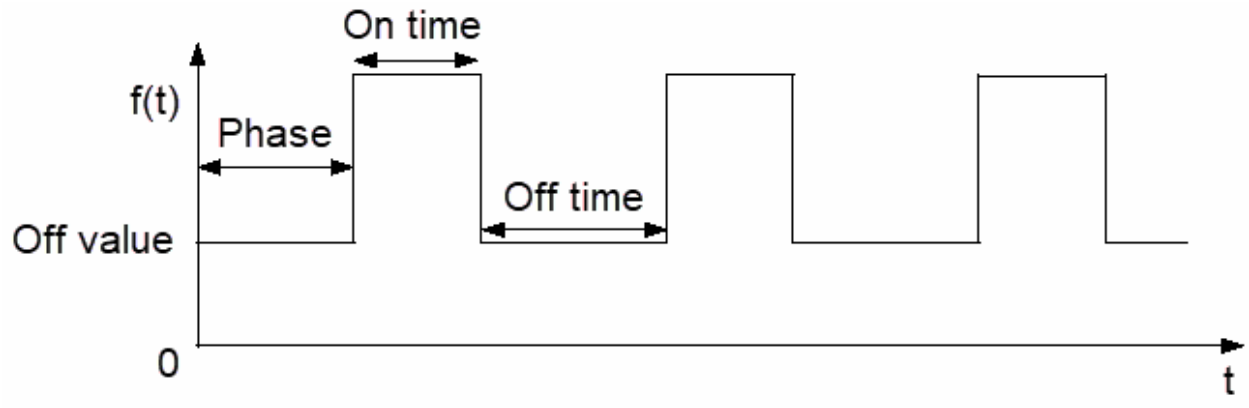
Square Wave

$$s_t = s_0 f(t)$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



where s_t is the value of the variable at time t and s_0 is the value of the variable at t = 0. $f(t)$ is a function of time as shown in the following figure.



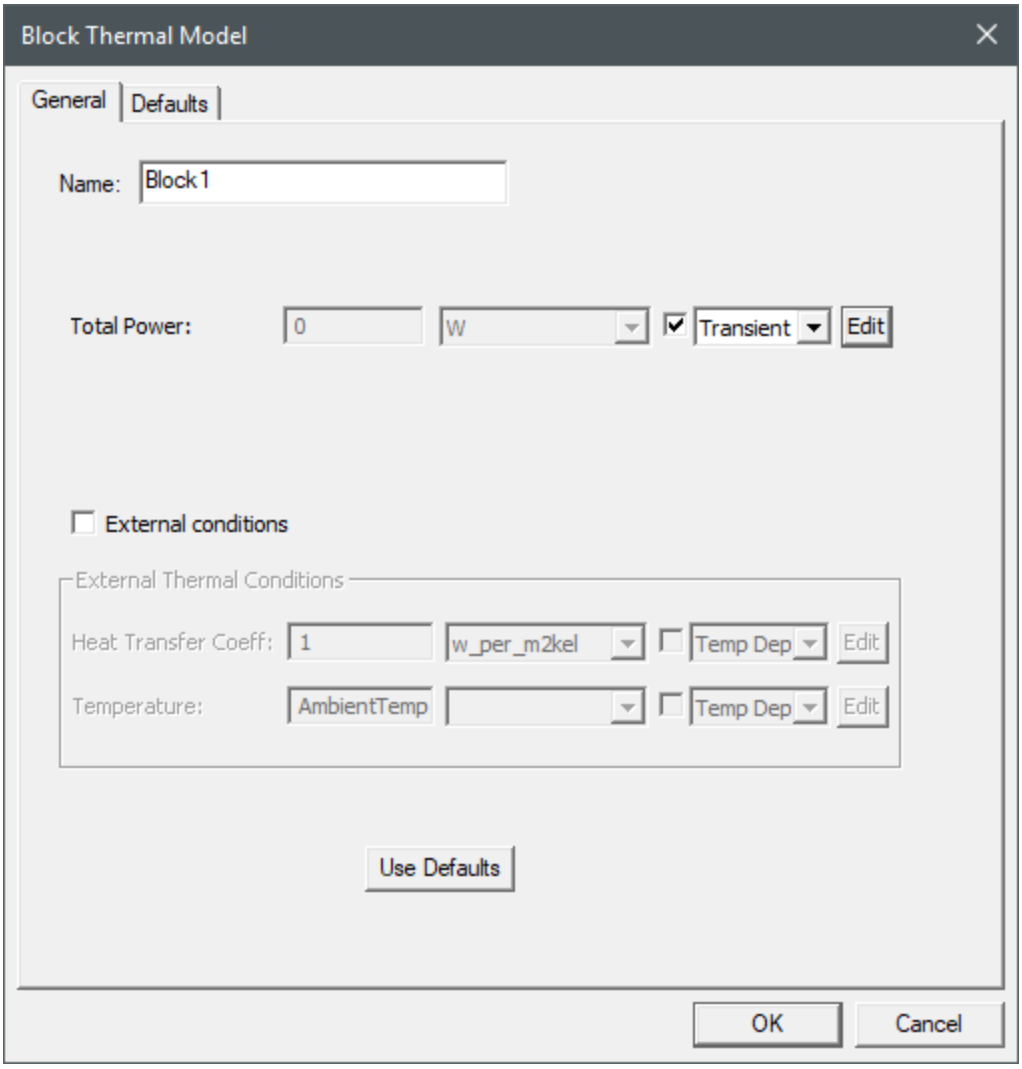
The **Phase** is the time between t = 0 and the first peak of the square wave. The **On Time** is the time that the square wave is at its peak value. The **Off Time** is the time between peak values of the square wave. The **Off Value** is the value of the square waves between the peaks of the wave. The peak value is the specified value of the transient quantity

Using Datasets to Define Transient Parameters

Some parameters for Icepak thermal boundary conditions and design settings can be assigned a transient value using a dataset. For a complete list of transient parameters, see Transient Thermal Properties Dependency. You can access the **Add Dataset** dialog two ways:

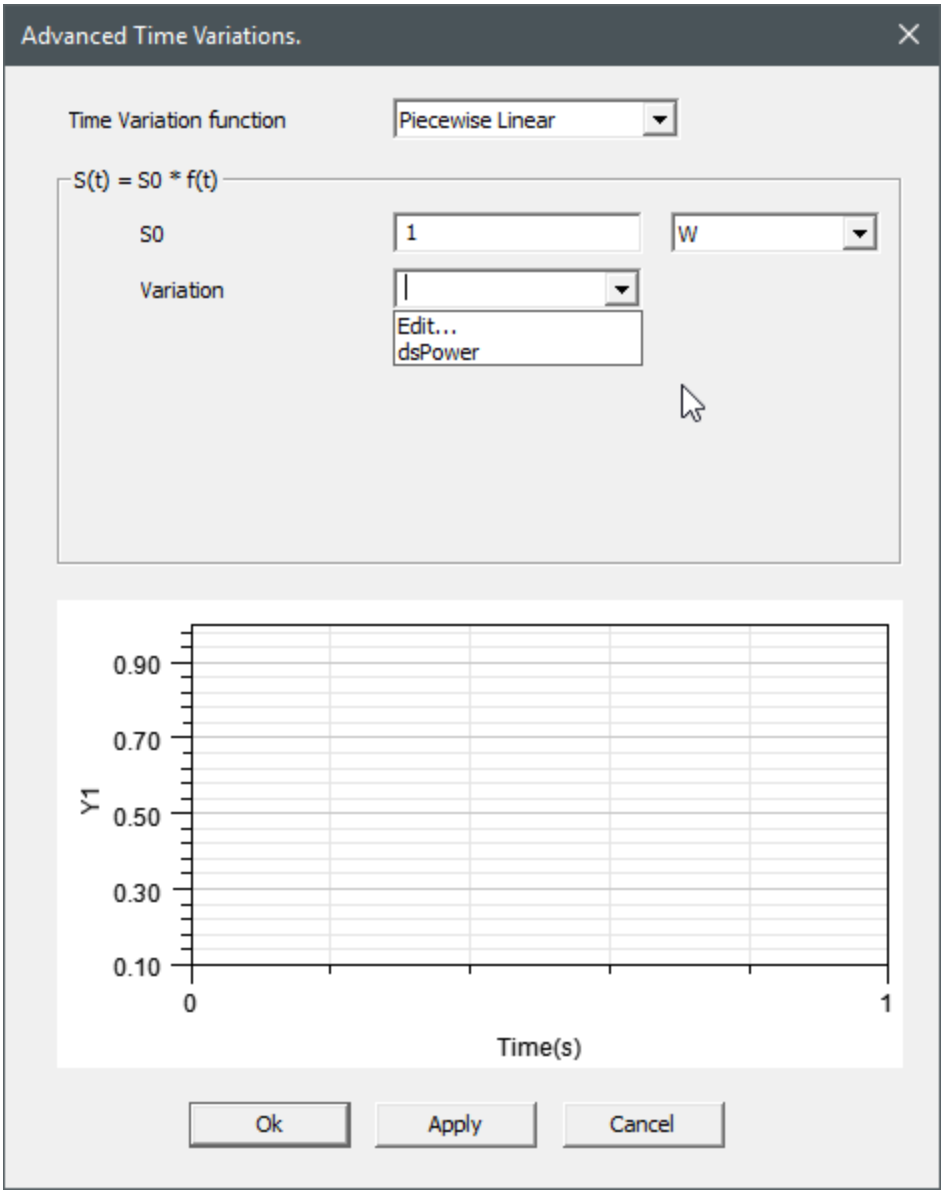
- Create a dataset prior to defining thermal boundary conditions or design settings by adding a design dataset. See Adding Datasets for more information.
- When defining thermal boundary conditions or design settings, enable the transient option for the parameter and define the dataset. See "Defining a Dataset for Transient Parameters" on page 18-8 for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: Datasets you have previously created are available as options in the **Variation** drop-down list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

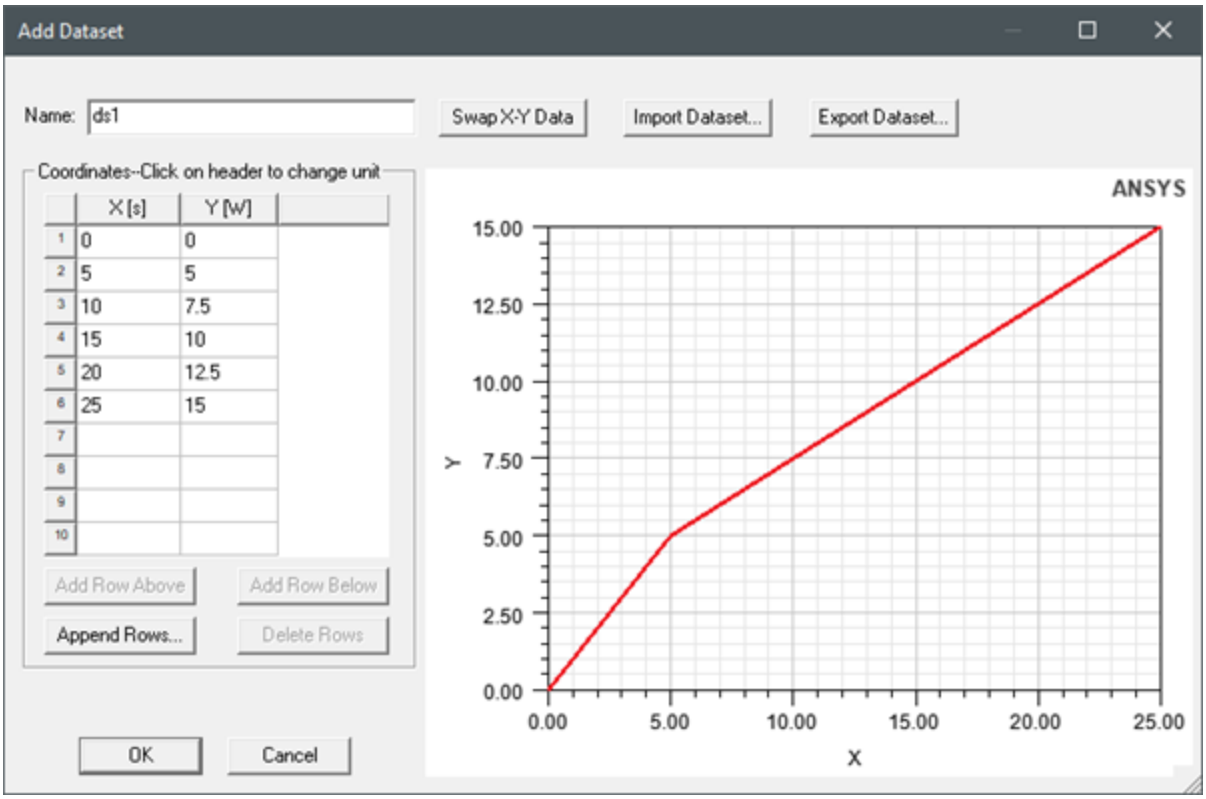


Defining a Dataset for Transient Parameters

1. In the boundary condition or design settings dialog box, enable the check box next to the parameter.
2. Select **Transient** from the drop-down list and click **Edit**.
3. In the **Advanced Time Variations** dialog box, select **Piecewise Linear** from the **Time Variation Function** drop-down list.
4. Specify a value for **S0**, which is the multiplier value applied to the variable parameter defined in the piecewise linear dataset (y-axis value).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. From the **Variation** drop-down list. select **Edit**.
6. In the **Datasets** dialog box, click **Add** > **1D Dataset**.
7. In the **Add Dataset** dialog box, enter a **Name**.
8. In the **Coordinates** area, click the **X** column header and select **Time** and then click it again and select a unit of time.
9. Click the **Y** column header and select a parameter category (e.g., **Power**) and then click it again and select a unit for the parameters (e.g., **W**).
10. In the **X** and **Y** columns enter the values for time and the time-dependent parameter.



11. Click **OK** to save the dataset.
12. In the **Datasets** dialog box, click **Done**.

Transient Thermal Properties Dependency

In the Electronics Desktop, you can create datasets, which are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. In Icepak, you can create a dataset to define time-dependent, temperature-dependent, or spatially-dependent thermal properties, as detailed in the following tables.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Setting

Settings	Thermal Property	Transient Dependency
Design Settings	Ambient Temperature	Time

Icepak 3D Native Component

Native Component	Thermal Property	Transient Dependency
Fan	Transient Strength	Time

Boundary Conditions

Boundary Condition	Thermal Property	Transient Dependency
Fluid Block	Total Power	Temperature, Time, Spatial
	Heat Transfer Coefficient	Temperature, Time
	Temperature	Time
Hollow Block	Total Power	Temperature, Time, Spatial
	Heat Flux	Temperature, Time
	Temperature - Fixed Temperature	Time
	Heat Transfer Coefficient	Temperature, Time
	Temperature - Internal Conditions	Time
Solid Block	Total Power	Temperature, Time, Spatial
	Heat Transfer Coefficient	Temperature, Time
	Temperature	Time
Openings	Temperature	Time, Spatial
	Static Pressure/Total Pressure	Time, Spatial
	X Velocity	Time, Spatial
	Y Velocity	Time, Spatial
	Z Velocity	Time, Spatial
Plate (Conducting Thin)	Power	Temperature, Time
Resistance	Power	Time, Spatial
3D Source	Power	Temperature, Time
	Surface/Volume Flux	Temperature, Time, Spatial

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Source	Power	Temperature, Time
	Surface/Volume Flux	Spatial
Wall (Stationary)	Heat Flux	Time, Spatial
	Temperature	Time, Spatial
	Heat Transfer Coefficient	Temperature, Time, Spatial
Network	Internal Node Power	Temperature, Time
	Boundary Node Temperature	Temperature, Time

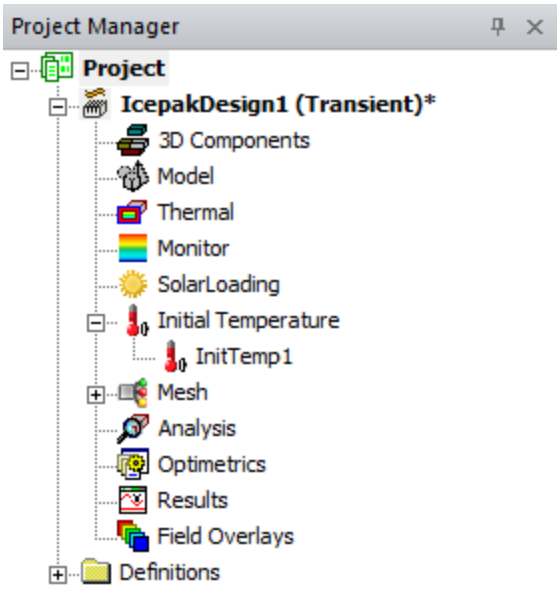
Assigning Initial Temperatures

For transient simulations, you can assign an initial temperature (temperature patch) to 3D geometry in the design, including the 3D geometry composing Icepak native 3D components (e.g., a fan). The temperature is assigned to the starting time step. At the start of a transient solution, the selected geometries begin with assigned temperatures.

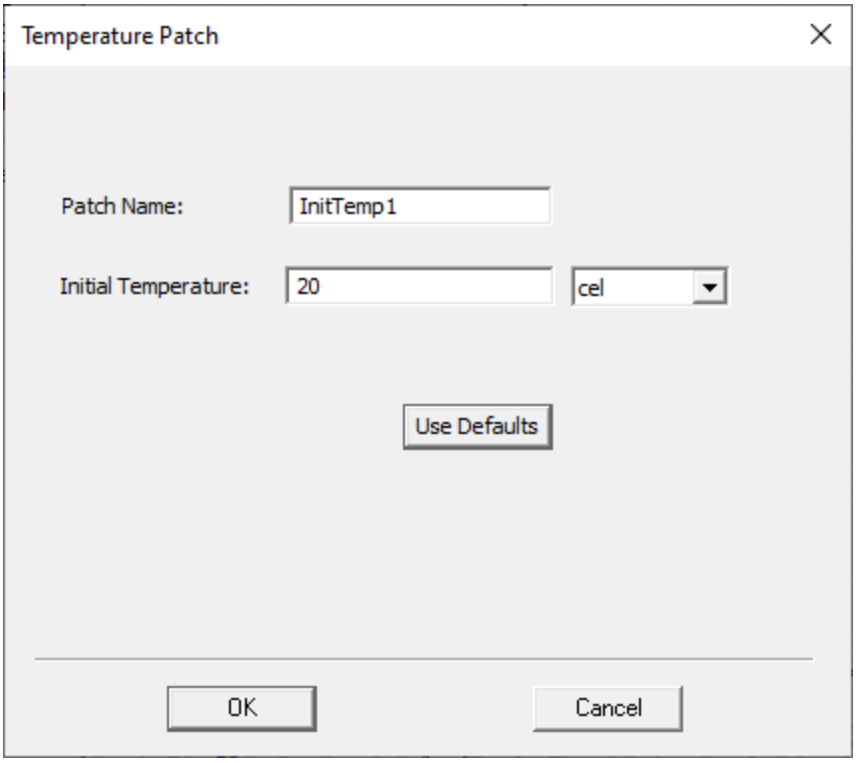
> **Note:** When **Start/Continue from a previously solved setup is** enabled, Icepak does not use the assigned temperature patch to start the simulation.

> **Note:** Temperature patches can be assigned variables for use in parametric trials.

Assigned temperature patches appear under **Initial Temperature** in the **Project Manager**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Patch Name	The **Patch Name** of the temperature patch appears in the **Project Manager** window and **3D Modeler** window.
Initial Temperature	The **Initial Temperature** specifies the temperature the selected object is assigned at the start of a transient simulation.

Assign an Initial Temperature

To assign a temperature patch and define its properties:

1. Press O to enable object-selection mode.
2. Select and right-click a three-dimensional geometry component in the **3D Modeler** window or the history tree.
3. From the right-click menu, select **Assign Initial Temperture**.
4. In the **Temperature Patch** dialog box, enter a **Patch Name** for the temperature patch.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter an **Initial Temperature** value and select a unit.

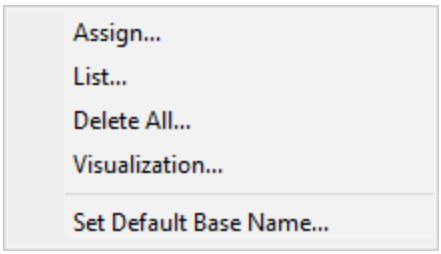
> **Note:**
>
> You can define the temperature patch using a 3D dataset (representing a spatial or volumetric profile). See Adding 3D Datasets for more information.

6. Click **OK**.

> **Note:**
>
> The **Temperature Patch** dialog has a **Defaults** tab, which contains information related to default settings for the temperature patch properties and the following buttons:
>
> - **Save Defaults** saves the currently defined properties as the default for newly created temperature patches.
> - **Revert to Standard Defaults** sets the temperature patch properties to the saved default values.

Initial Temperature Context Menus

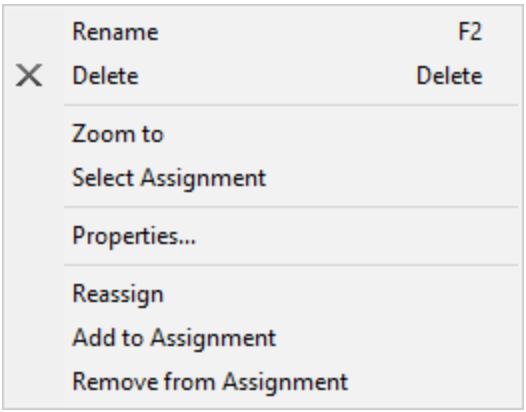
In the **Project Manager**, right-click the **Initial Temperature** module to display the context menu.



Assign	**Assign** opens the **Temperature Patch** dialog box to assign a temperature patch. 3D geometry must be selected to assign a temperature patch.
List	**List** opens the **Design List** dialog box on the **Initial Temperature** tab. See Viewing the Design List for Icepak for more information.
Delete All	**Delete All** removes all temperature patches from the design.
Visualization	**Visualization** opens the **Visualization Options** dialog box. See Initial Temperature Visualization Options for more information.
Set Default Base Name	**Default Base Name** opens the **Initial Temperature Base Name** dialog box. See Setting Initial Temperature Base Names for more information.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project Manager**, expand **Initial Temperature** and right-click a temperature patch to display the context menu.

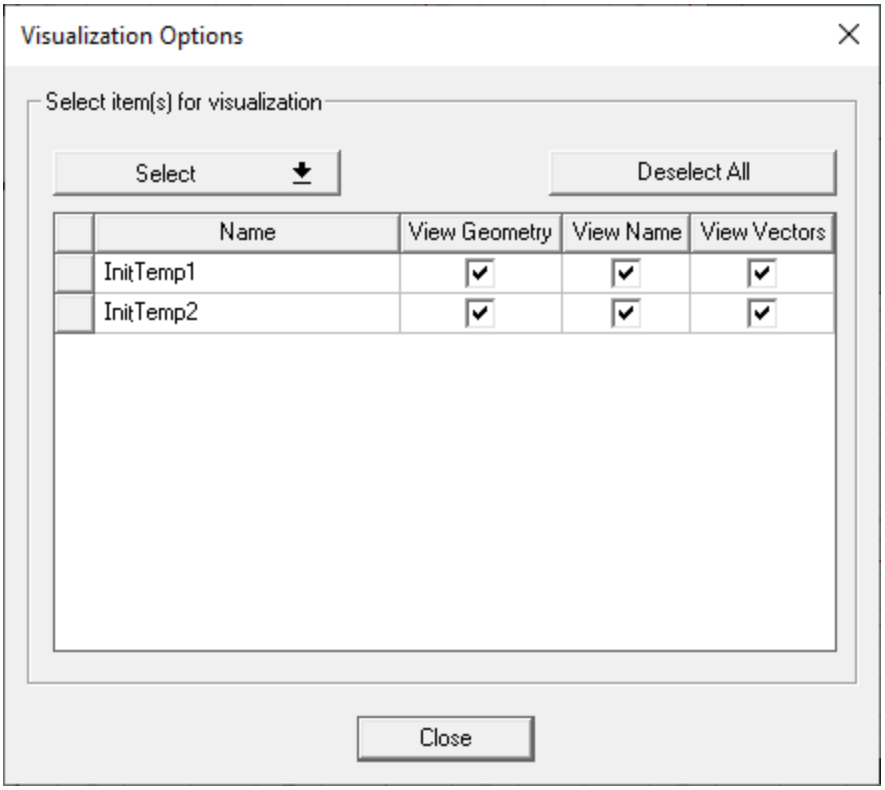


Rename	**Rename** allows you to edit the name of the temperature patch.
Delete	**Delete** removes the temperature patch from the design.
Zoom To	**Zoom To** allows you to zoom to the selection.
Select Assignment	**Select Assignment** selects the geometry assigned to the temperature patch in the **3D Modeler** window and the history tree.
Properties	**Properties** opens the **Temperature Patch** dialog box.
Reassign	**Reassign** assigns the temperature patch to the currently selected geometry .
Add to Assignment	**Add to Assignment** adds the currently selected geometry to the temperature patch.
Remove from Assignment	**Remove from Assignment** removes the selected geometry from the temperature patch.

Initial Temperature Visualization Options

Temperature patch information displayed in the **3D Modeler** window can be enabled or disabled using the **Visualization Options**, which can be accessed from the **Icepak**>**Initial Temperature** menu or by right-clicking **Initial Temperature** in the **Project Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select	The **Select** drop-down contains options to select **All Initial Temperature** or **By Name**. Select **By Name** to open the **Select By Name** dialog box in which you can enter a temperature patch name and click **OK** to select it.
Deselect All	Click **Deselect All** to deselect all temperature patches.
Name	The **Name** of the temperature patch appears in the **3D Modeler** window.
View Geometry	Enable **View Geometry** for the temperature patches for which you want to display geometry in the **3D Modeler**.
View Name	Enable **View Name** for the temperature patches for which you want to display names in the **3D Modeler**.
View Vectors	Enable **View Vectors** for the temperature patches for which you want to display vectors in the **3D Modeler**.
Close	Click **Close** to close the **Visualization Options** dialog box.

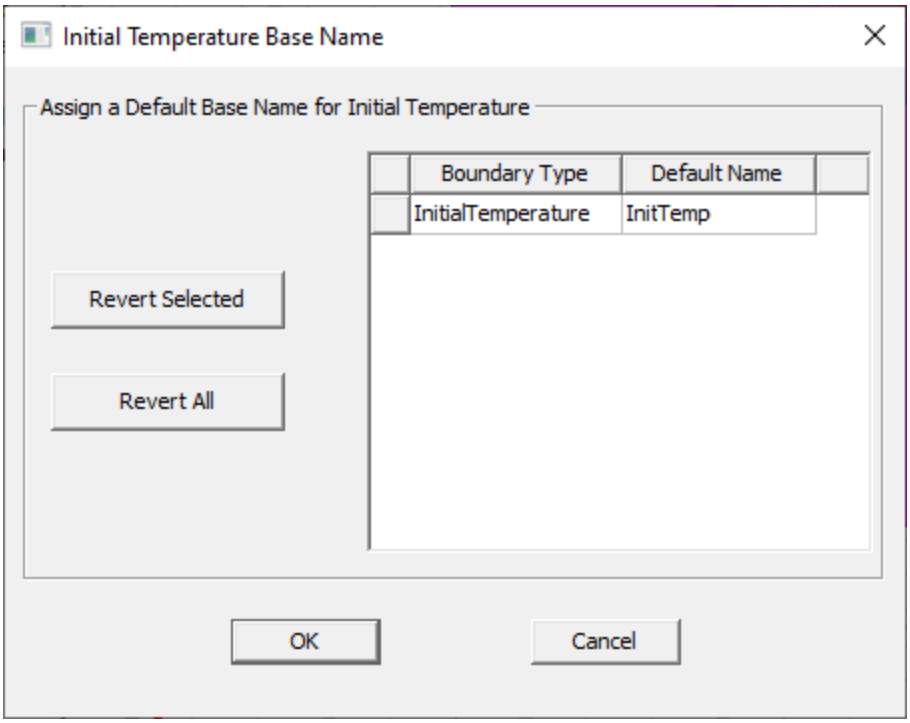
Setting Initial Temperature Base Names

When creating a temperature patch, a name are assigned by default. The default naming convention can be customized using the **Initial Temperature Base Name** dialog box, which can be accessed from the **Icepak**>**Initial Temperature** menu or by right-clicking **Initial Temperature** in the **Project Manager**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Temperature patch names can be changed by selecting them in the **Project Manager** and pressing F2 or right-clicking them in the **Project Manager** and selecting **Rename**.



Boundary Type	**Boundary Type** specifies the type of temperature patch.
Default Name	Enter a **Default Name** to use as the base name for new temperature patches.
Revert Selected	Click **Revert Selected** to restore the default value for the **Default Name** of the selected temperature patch type.
Revert All	Click **Revert All** to restore the default value for the **Default Name** of all temperature patch types.
OK	Enable **View Name** for the temperature patches for which you want to display names in the **3D Modeler**.
Cancel	Click **Cancel** to close the **Initial Temperature Base Name** dialog box without saving any changes.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Icepak Solve Setup Dialog (Transient)

A solution setup includes the settings you need to define for the Icepak simulation. For transient simulations, the **Icepak Solve Setup** dialog box contains the following tabs:

- **General**
- **Save Fields**
- **Simulation Control (Dynamic Thermal Modeling Beta)**
- **Convergence**
- **Solver Settings**
- **Radiation**
- **Solar Radiation**
- **Defaults**

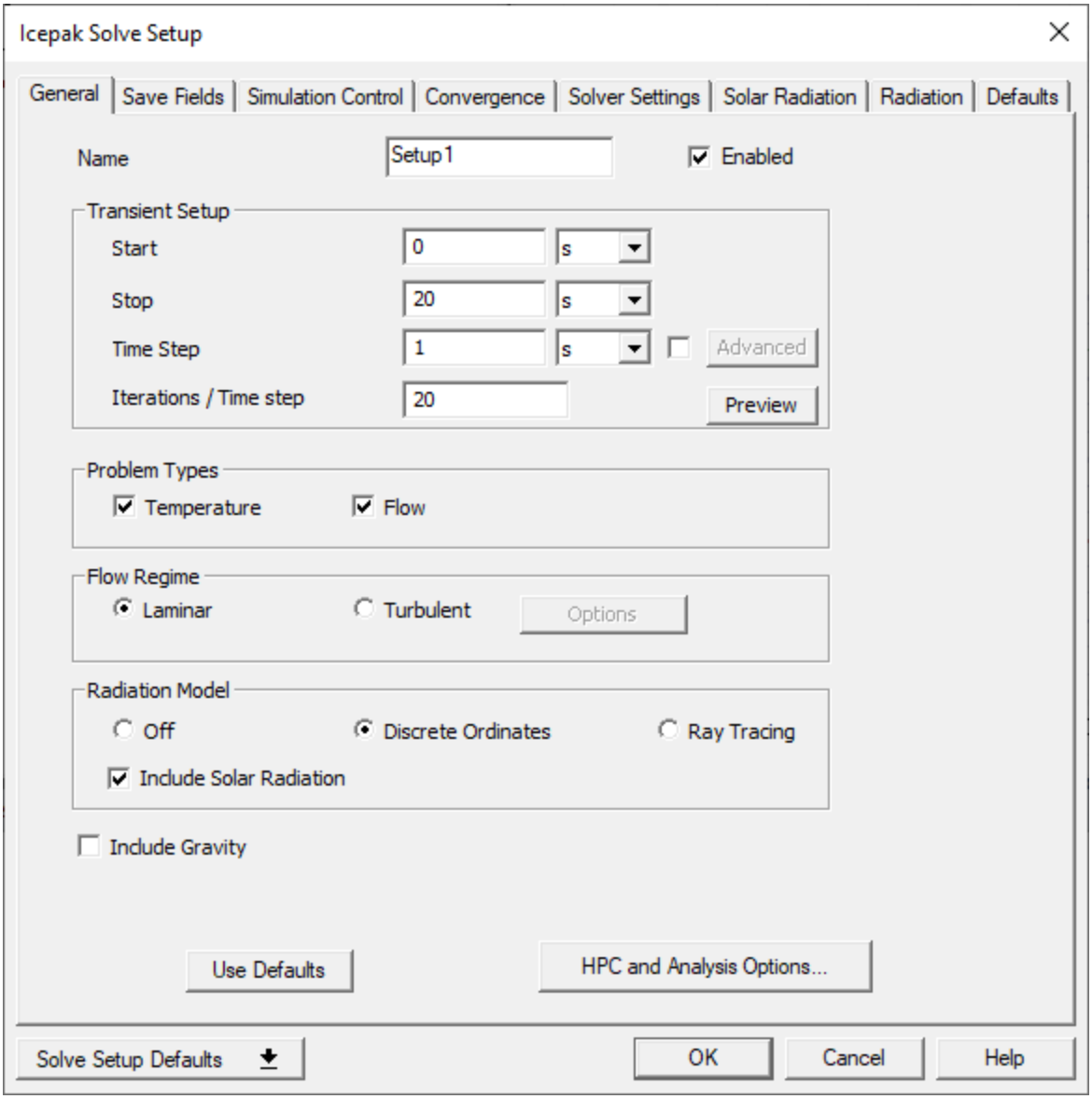
> **Note:** To enable Dynamic Thermal Modeling on linux systems, you must set the following environment variable:
>
> export fluent_python=$<*Install_Dir*>/ansys_inc/v261/AnsysEM/commonfiles/CPython/3_10/linx64/Release/python/runpython

The **Solve Setup Defaults** drop-down list contains the following options that automatically configure settings based on your selection. See Transient Icepak Solve Setup Defaults for more information.

- **Solve Setup Defaults**
- **Forced Convection Defaults**
- **Natural Convection Defaults**
- **Mixed Convection Defaults**
- **Conduction Only Defaults**
- **LTI ROM Defaults**

> **Note:**
>
> Some settings are enabled or disabled based on other setting selections. For example, the **Radiation** tab is only displayed if radiation is enabled by setting the **Radiation Model** to **Discrete Ordinates** or **Ray Tracing**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	The **Name** of the setup appears in the **Project Manager** window.
Maximum Number of Iterations (Steady State only)	**Maximum Number of Iterations** is the number of solution iterations to be performed in a simulation. The simulation will stop when these iterations have been performed or the **Convergence Criteria** are satisfied, whichever happens first. For relatively simple models, the default of 100 should be sufficient for the solution to converge, but for more complex models you may need to increase this value.
Enabled	The **Enabled** check box disables/enables a setup. Only

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	enabled setups are run when you select **Analyze All**.
Transient Setup (Transient only)	
Start	**Start** defines the start time for the simulation.
Stop	**Stop** defines the ending time for the simulation.
Time Step	**Time Step** defines the duration of a time step. To define varying time steps, enable the check box to open the **Advanced Time Variations** dialog box. See Specifying Varying Time Steps for Transient Solution Setups for more information.
Advanced	Click **Advanced** to open the **Advanced Time Variations** dialog box. See Specifying Varying Time Steps for Transient Solution Setups for more information.
Iterations/Time step	**Iterations/Time step** defines the maximum number of iterations to be performed per time step. If the convergence criteria are satisfied before this number of iterations is performed, the simulation advances to the next time step. The default value of 20 iterations/time step should be acceptable for most cases. If you find that the solution is not converging at each time step, you can increase **Iterations/Time step** value and/or decrease **Time Step**.
Preview	Click **Preview** to display the **Variations Preview** dialog box, which contains a preview of all transient advanced time variations in the design.
Problem Types	
Temperature	When enabled, the **Temperature** check box instructs the solver to include thermal calculations in the analysis.
Flow	When enabled, the **Flow** check box instructs the solver to include fluid calculations in the analysis.
Flow Regime	
Laminar	**Laminar** flow is smooth, regular, deterministic, and steady. In laminar flow, fluid mixing and heat transfer take place on a molecular level. The molecular (or dynamic) viscosity and the thermal conductivity are the quantities that measure the amount of mixing and heat transfer. **Laminar** is the default flow regime.
Turbulent	**Turbulent** flows are characterized by fluctuating velocity fields. In turbulent flow, the degree of fluid mixing and heat transfer is much greater than in laminar flow, and takes place on a global, or macroscopic, level rather than on a molecular level. The amount of fluid mixing is measured by an effective

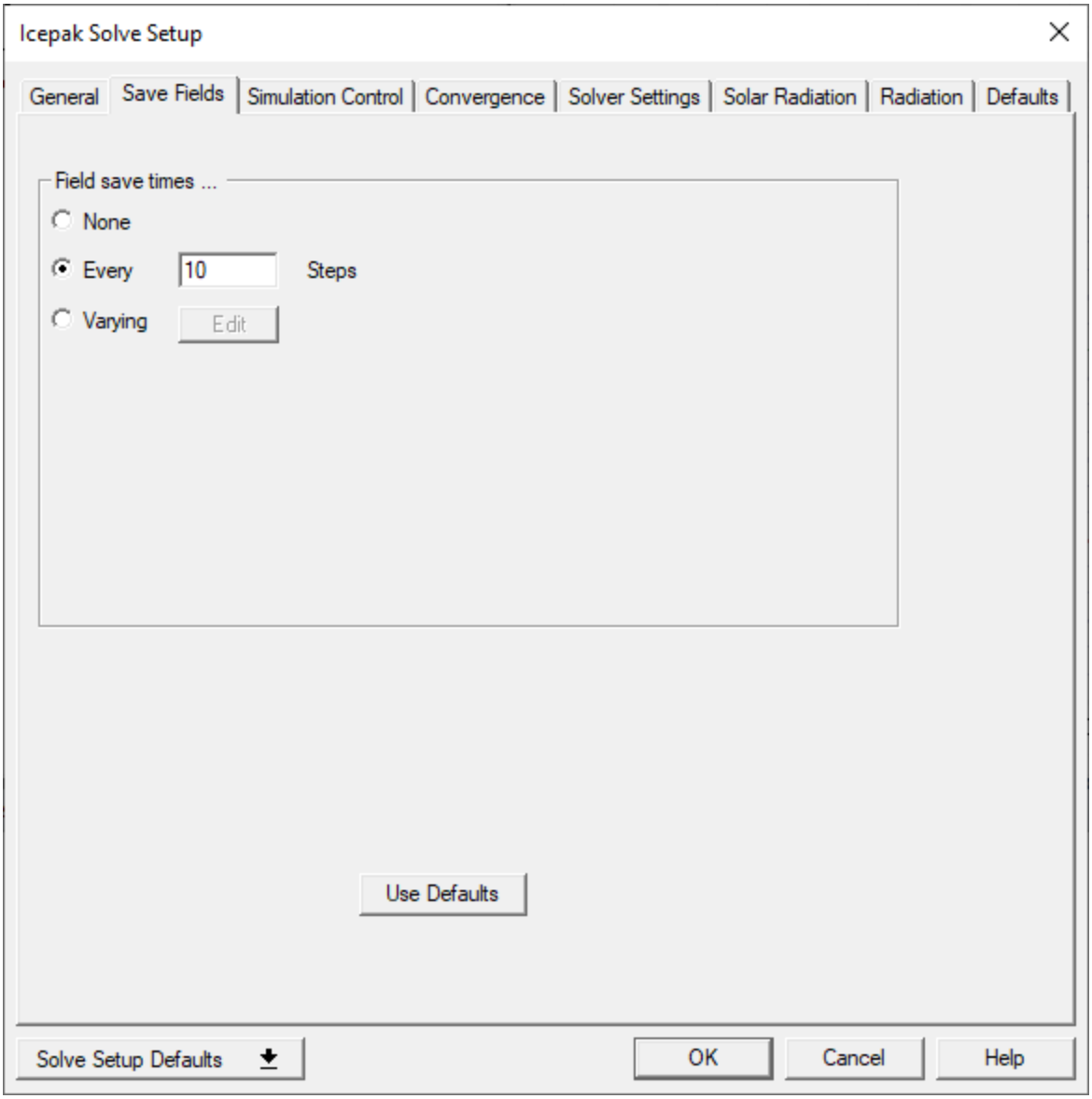
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	viscosity, which is the sum of the dynamic viscosity and the turbulent eddy viscosity. If you select **Turbulent**, click **Options** to open the **Turbulent Flow Model** dialog and select a model. The default turbulent flow model is **Zero Equation**.
Radiation Model	
Off	**Off** disables radiation for the solution. Off is the default setting.
Discrete Ordinates	The **Discrete Ordinates** (DO) radiation model solves the radiative transfer equation (RTE) for a finite number of discrete solid angles, each associated with a vector direction $\vec{s}$ fixed in the global Cartesian system (x, y, z). The DO model solves for as many transport equations as there are directions $\vec{s}$. The solution method is identical to that used for the fluid flow and energy equations.
Ray Tracing	With the **Ray Tracing** radiation model, the ray paths are calculated and stored prior to the fluid flow calculation. At each radiating face, rays are fired at discrete values of the polar and azimuthal angles. To cover the radiating hemisphere, θ is varied from 0 to $\frac{\pi}{2}$ and ϕ from 0 to 2π. Each ray is then traced to determine the control volumes it intercepts as well as its length within each control volume. This information is then stored in the radiation file (.s2s.gz), which is then read in before the fluid flow calculations begin.
Include Gravity	To include the effect of gravity in the simulation for natural convection models, select **Include Gravity**. Define the gravity vector settings in the Icepak Design Settings. Natural convection models require **Include Gravity** to be enabled.
HPC and Analysis Options	High performance computing (HPC) enables a range of different technologies in Icepak that allows efficient simulation of extremely large and complex problems. See High Performance Computing for more information.

Note:

Ray tracing radiation is not calculated on the faces of mesh regions that touch other objects.

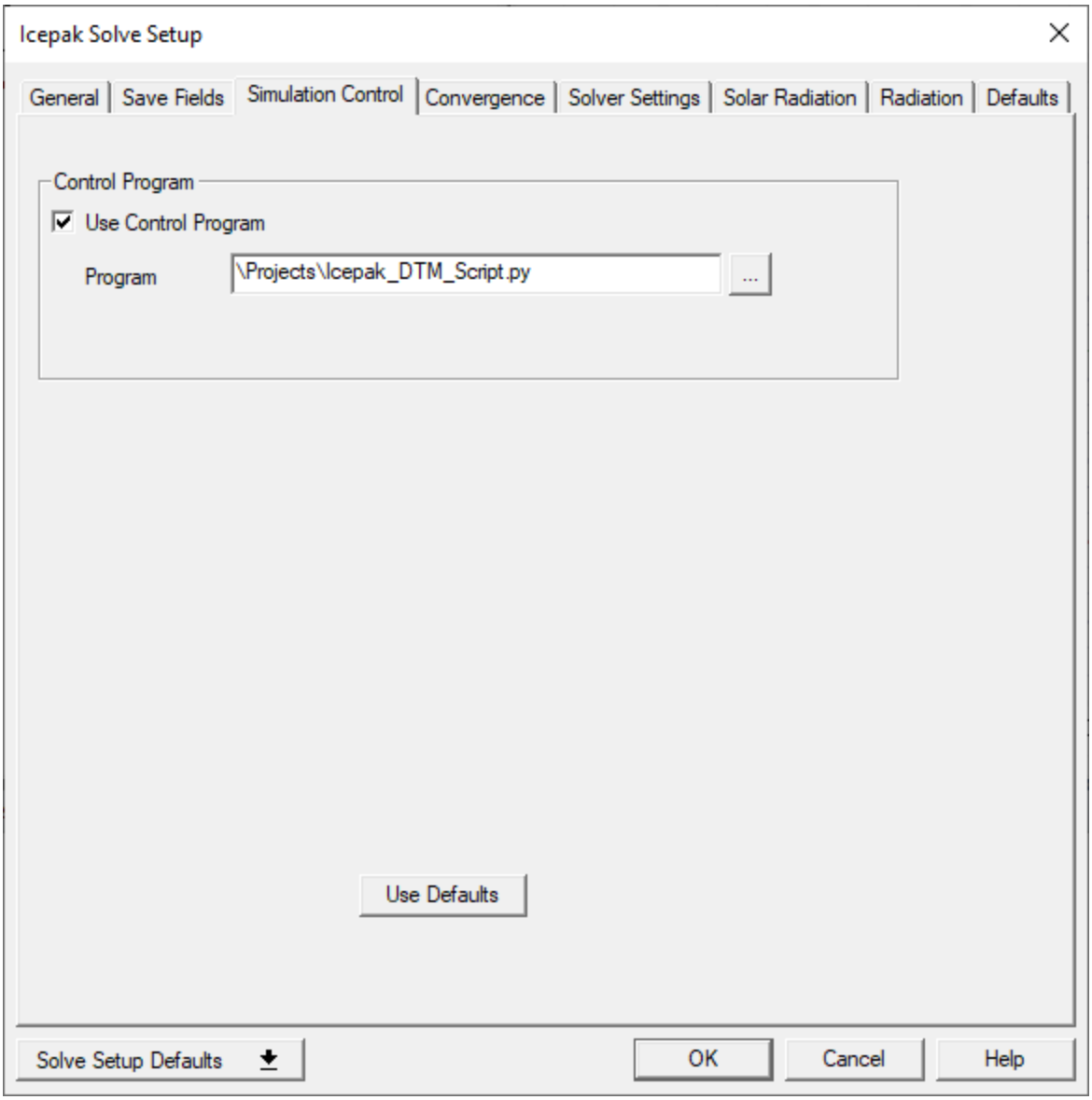
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Field Save Times (Transient only)	
None	**None** specifies that Icepak field quantity data will not be saved at any interim time points during the simulation.
Every # Steps	**Every # Steps** specifies the frequency at which Icepak field quantity data is saved for creating post-processing (fields calculations, plots, reports, etc.). The amount of save times is determined by the simulation start and stop times and duration of a time step defined on the **General** tab.
Varying	**Varying** displays the **Save Field - Transient Variation** dialog

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

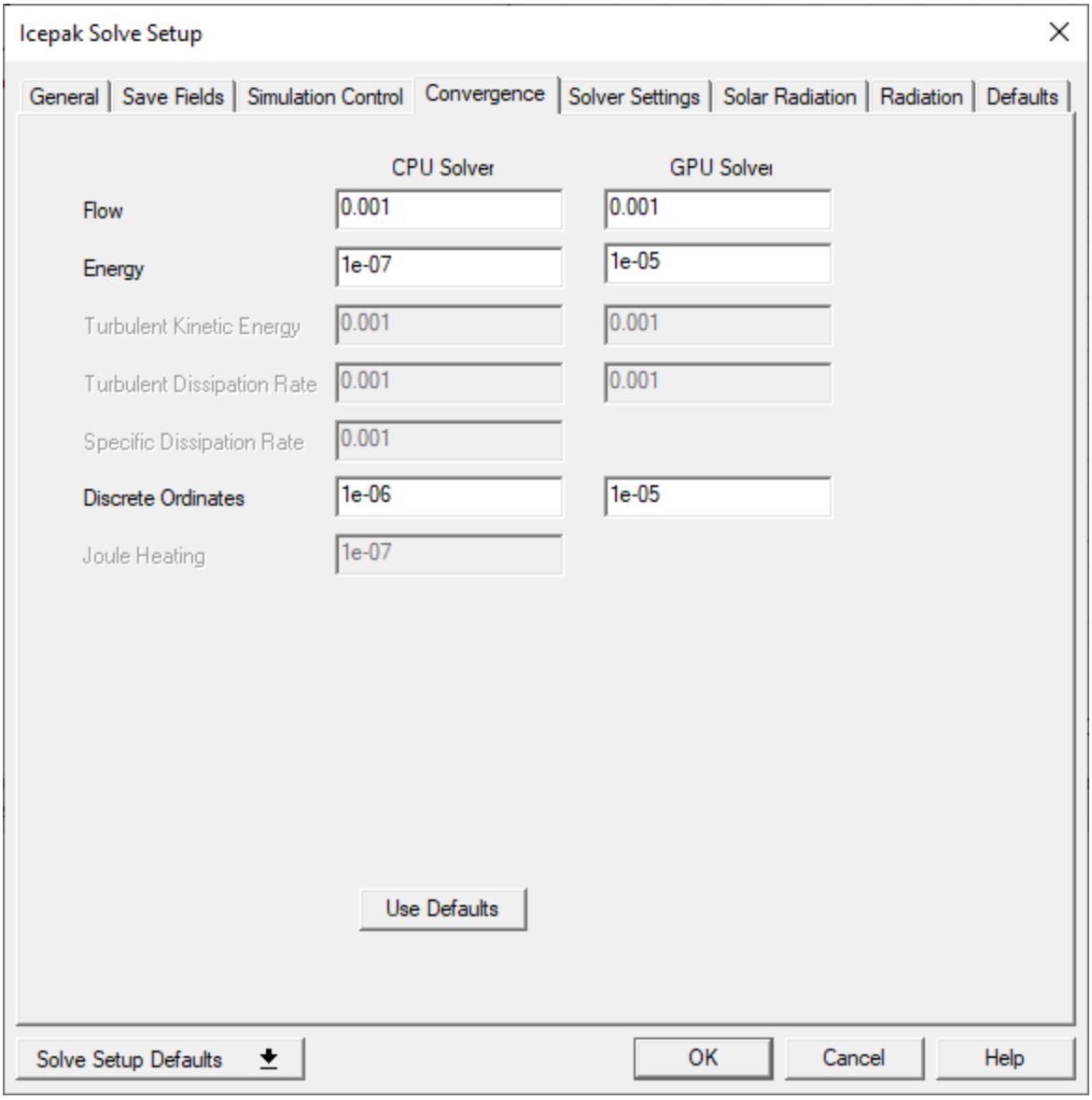
	box, in which you can specify varying save times using a piecewise constant variation function. Click **Edit** to make changes to the variation function.



Control Program (Dynamic Thermal Modeling)	
Use Control Program	**Use Control Program** allows you to select the **Program** script you previously created using the Thermostat for Source Control toolkit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

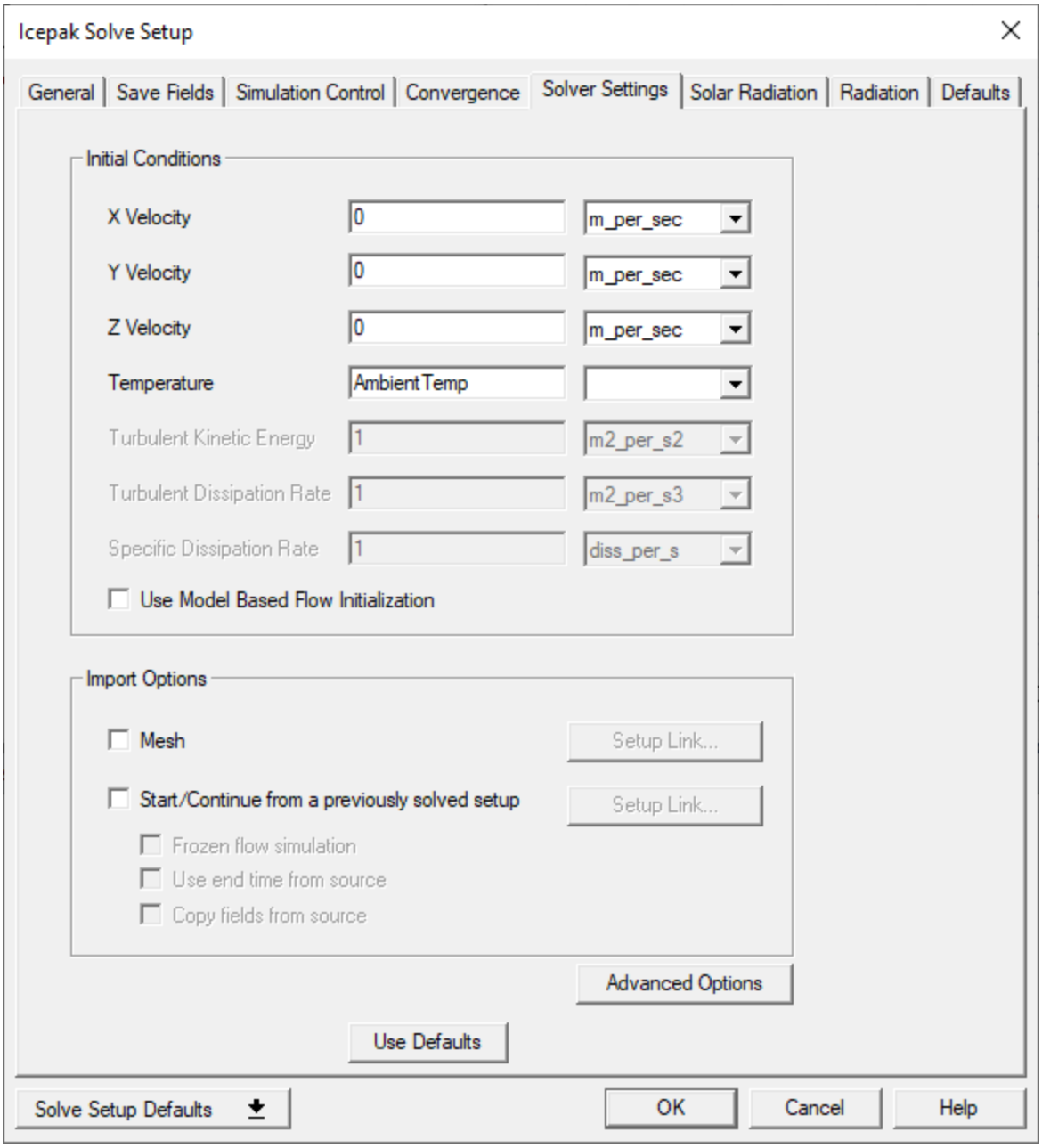
Note: You can use the DTM Python Verification Form toolkit to ensure there are no errors in the control program script.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow	These settings are the solution-residual values used to determine convergence. Solution residuals measure the error or imbalance in the conservation equations that Icepak solves. When all solution residuals are less than or equal to their specified convergence criteria, the solution will be considered converged. **Turbulent Kinetic Energy**, **Turbulent Dissipation**, and **Specific Dissipation Rate** are enabled by the selection of certain turbulent flow models. **Discrete Ordinates** is enabled if the **Radiation Model** is set to **Discrete Ordinates**. **Joule Heating** is enabled if the power for a block boundary condition is defined as joule heating.
Energy	
Turbulent Kinetic Energy	
Turbulent Dissipation	
Specific Dissipation Rate	
Discrete Ordinates	
Joule Heating	

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



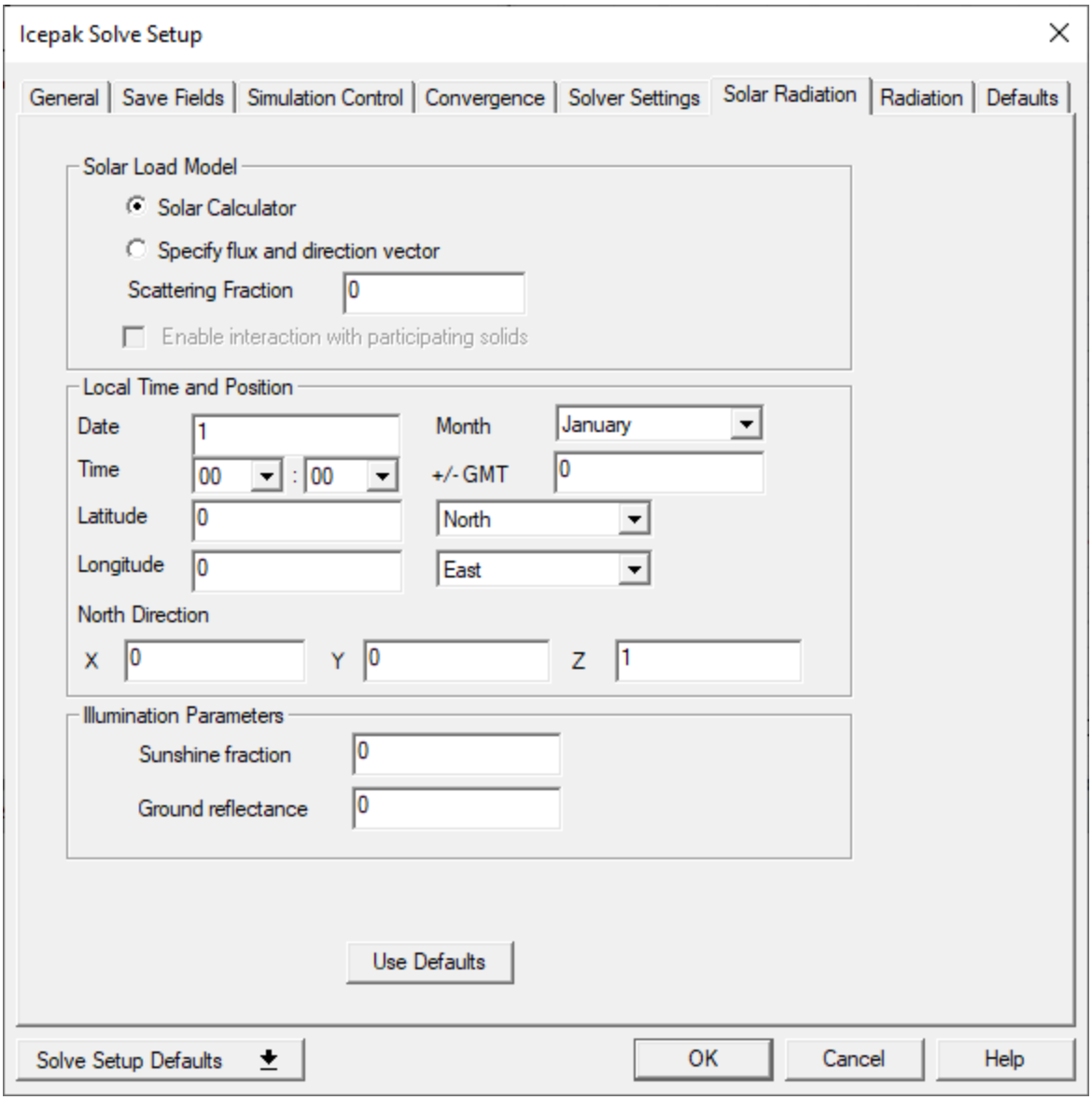
Initial Conditions

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z Velocity **Temperature** **Turbulent Kinetic Energy** **Turbulent Dissipation** **Specific Dissipation Rate**	These settings specify the initial conditions for the fluid in the flow region. The initial conditions are the initial guess for the various solution fields used by the solution procedure. **Turbulent Kinetic Energy**, **Turbulent Dissipation**, and **Specific Dissipation Rate** are enabled by the selection of certain turbulent flow models.
Use Model Based Flow Initialization	**Use Model Based Flow Initialization** instructs the Electronics Desktop to automatically evaluate inlet and outlet conditions to use optimal values for solution variables.
Import Options	
Mesh	**Mesh** allows to use mesh from an existing Icepak design and is automatically enabled after using the Add a Mesh Linked Solution Setup option. Mesh from the selected design is imported and used in the simulation. The mesh from the source design must conform to the target design's model geometry. Seeing Importing Mesh for more information.
Start/Continue...	**Start/Continue from a previously solved setup** allows you to begin or resume a simulation from a design you have previously solved. See Transient Import Options Requirements for more information.
Note: If the design has a mesh link and solve setup link, if you remove the solve setup link, the mesh link is not automatically removed.	
Frozen flow simulation	**Frozen flow simulation** allows you to disable the flow of the linked steady-state or transient design. If enabled, only the energy equation is solved during the analysis. If the linked solution did not solve for flow, this option is not active.
Use end time from source (Transient only)	**Use end time from source** allows you to start the analysis from the linked solution's stop time. For example, if the linked solution solved from 0 to 10 seconds and the target solution is set to solve from 0 to 20 seconds, the target solution starts solving at 10 seconds and stops at 20 seconds.
Copy fields from source (Transient only)	When **Use end time from source** is enabled, **Copy fields from source** is available. Enable this option to copy fields data from the source design to make it available for post-processing in the target design. Using the example in the definition above, fields data from the target design (from 0 to 10 seconds) will be available to post-process in the target design.
Advanced Options	Click **Advanced Options** to display the **Advanced Solver**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Settings dialog box. To review advanced solver settings, see The Icepak Solve Setup Dialog.



Solar Load Model	
Solar Load Model	Select **Solar Calculator** to compute the solar radiation based on time and geographic parameters or **Specify flux and direction vector** to specify solar radiation values and a solar radiation direction vector.
Scattering Fraction	**Scattering Fraction** specifies the amount of direct solar radiation that is reflected from opaque objects in your model. The reflected

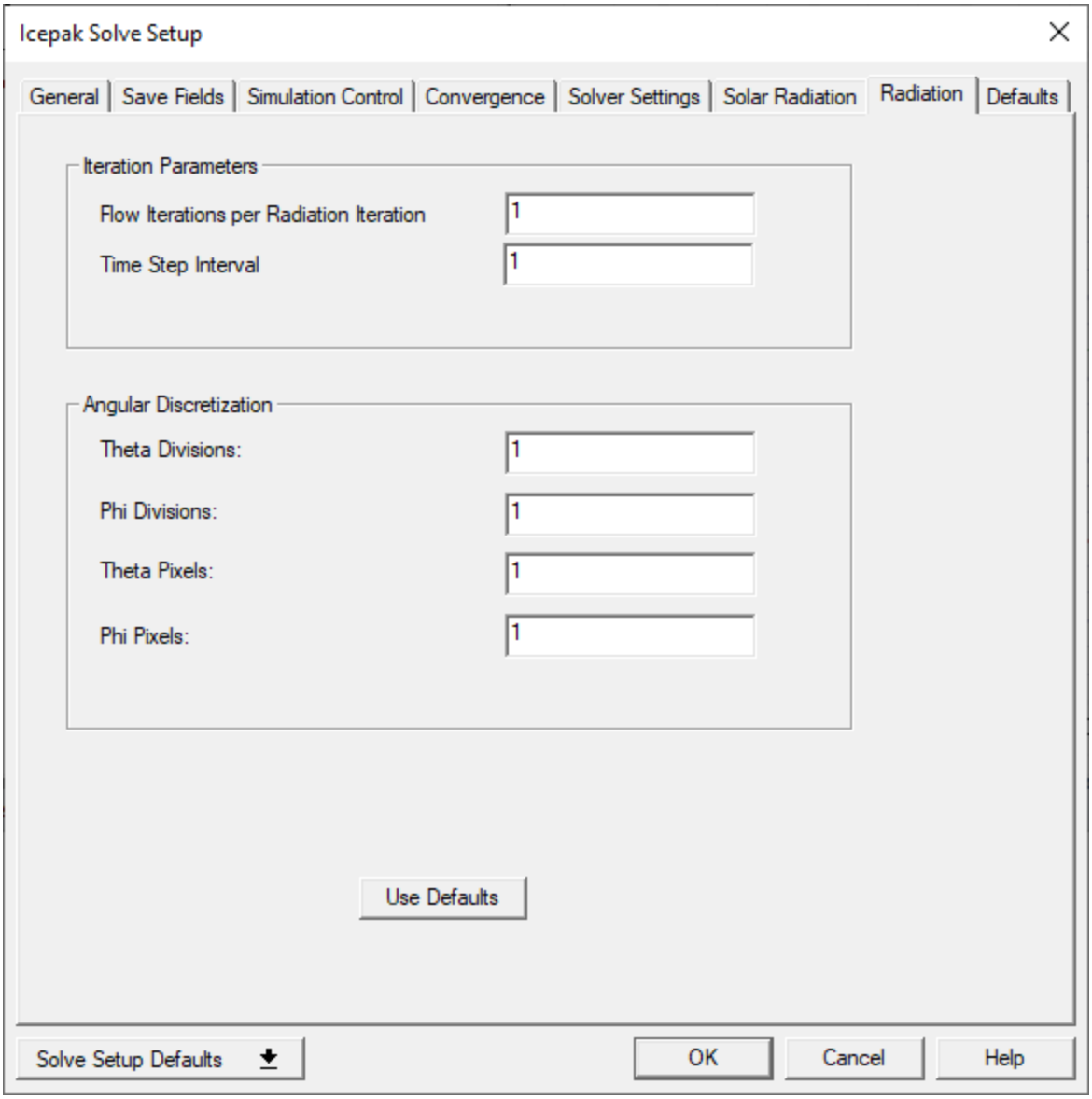
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	radiation is evenly distributed among all objects that participate in solar loading.
Enable interaction with participating solids	When **Discrete Ordinates** is defined as the radiation model, select **Enable interaction with participating solids** to model the solar irradiation as discrete ordinate fluxes at the global region boundaries. This allows semi-transparent walls and participating solids to absorb, refract and scatter the incident solar irradiation.
Local Time and Position (Solar Calculator)	
Date and **Month**	Specify a **Date** and **Month**.
Time	**Time** specifies the local time at the desired location. Select the hour and minute from the drop-down lists. The time is based on a 24-hour clock, therefore acceptable values range from 0 h 0 min (12:00 a.m.) to 23 h 59 min (11:59 p.m.).
+/- GMT	**+/- GMT** specifies the local time zone of the desired location. If the time you enter is a local time, specify the current time zone by providing the offset in the +/- GMT entry. If the time you enter is already in GMT, then +/- GMT should be set to 0 (zero). For example, enter -5 to specify Eastern Standard Time (EST).
Latitude	Specify the **Latitude** of the desired location. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator. Select the hemisphere (N or S) from the menu to the right of the Local Longitude entry field.
Longitude	Specify the **Longitude** of the desired location. The longitude is approximated if you specify the local time zone, but you can enter a more precise value if you know it. Any value you enter here will take precedence over the time zone. Values may range from 0° to 180°. Select the hemisphere (W or E) from the menu to the right of the Local Longitude entry field.
North Direction Vector	The **North Direction Vector** specifies the northward direction relative to the model. Enter the appropriate values in the **X**, **Y**, and **Z** fields. The default northward direction is in the Z direction.
Illumination Parameters (Solar Calculator)	
Sunshine fraction	**Sunshine fraction** is a factor between 0 and 1 used to account for the effects of clouds that may reduce the direct solar irradiation. Clear sky is modeled by setting the value equal to 1 and complete cloud cover is modeled by setting the value equal to 0. Partial cloud cover is modeled by setting the value to be between 0 and 1. The default value is 1.0.
Ground reflectance	**Ground reflectance** is a parameter that is used in determining the contributions of reflected solar radiation from ground surfaces.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Reflected solar radiation from ground surfaces is a function of the direct normal irradiation, the time of the year, the tilt angle of the surface, and the ground reflectance. If is treated as part of the total diffuse solar irradiation. Ground reflectance values can vary depending on the ground surface (that is, concrete, grass, rock, gravel, asphalt). The default value is 0.2.
Solar Flux and Direction Vector	
Direct solar irradiation	**Direct solar irradiation** specifies the amount of energy per unit area due to direct solar irradiation. This value may depend on the time of the year and the clearness of the sky.
Diffuse solar irradiation	**Diffuse solar irradiation** specifies the amount of energy per unit area due to diffuse solar irradiation. This value may depend on the time of year, the clearness of the sky, and also on ground reflectivity.
Solar Direction	**Solar Direction** specifies the direction of solar irradiation relative to the model. Enter the appropriate values in the **X**, **Y**, and **Z** fields.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iteration Parameters	
Flow Iterations per Radiation Iteration	The **Flow Iterations per Radiation Iteration** parameter is set to 1 by default. This implies that the radiation calculation is performed once every iteration of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.
Maximum Radiation Iterations (Ray Tracing only)	The **Maximum Radiation Iterations** controls the maximum number of iterations of the radiation calculation during each global iteration.

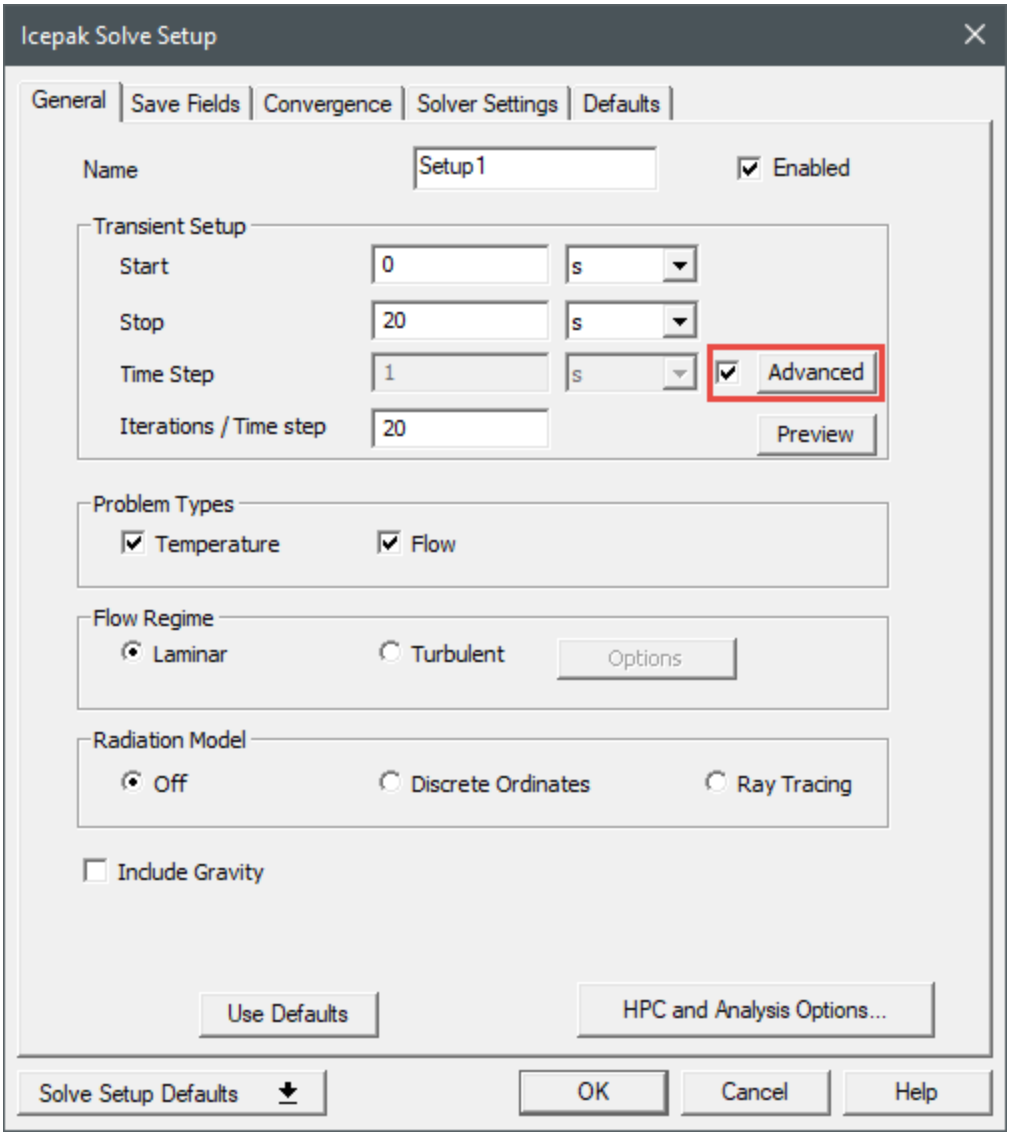
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angular Discretization (Discrete Ordinates only)	
Theta and **Phi Divisions**	**Theta Divisions** ($N\theta$) and **Phi Divisions** ($N\phi$) will define the number of control angles used to discretize each octant of the angular space. A finer angular discretization can be specified to better resolve the influence of small geometric features or strong spatial variations in temperature, but larger numbers of **Theta Divisions** and **Phi Divisions** will add to the cost of the computation.
Theta and **Phi Pixels**	**Theta Pixels** and **Phi Pixels** are used to control the pixelation that accounts for any control volume overhang. For problems involving gray-diffuse radiation, the default pixelation of 1 × 1 is usually sufficient. The computational effort, as a result of increasing the pixelation, is less than the computational effort caused by increasing the divisions. However, increasing the pixelation does add to the cost of computation.
Cluster Parameters (Ray Tracing only)	
Faces per Surface Cluster	**Faces per Surface Cluster** controls the number of radiating surfaces. By default, each is set to 20, so the number of surface clusters (radiating surfaces) will be equal to the total number of surface mesh elements divided by 20. For larger problems, you may want to reduce the number of surface clusters by increasing the faces per surface cluster to reduce both the size of the view factor file and the memory requirement. Such a reduction in the number of clusters, however, comes at the cost of some accuracy. The surfaces that are not adjacent to fluid regions will not participate in radiation.
View Factor Parameters (Ray Tracing only)	
Resolution	Increasing **Resolution** helps reduce the numerical errors caused by the finite-resolution approach used to compute the projected areas of the surface clusters and the resulting view factors. By default, it is set to 5. In most cases, however, the default settings will be sufficient.

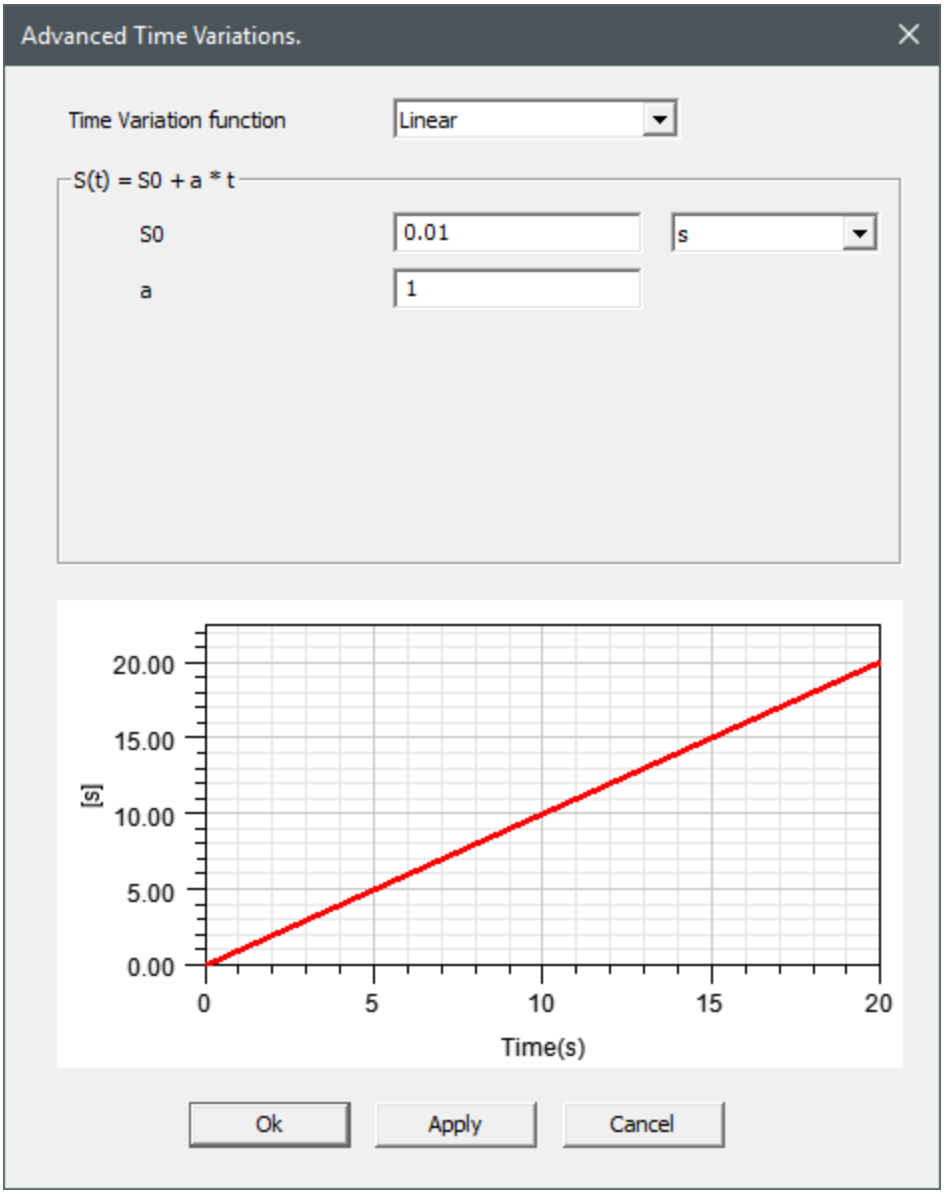
Specifying Varying Time Steps for Transient Solution Setups

For transient solution setups, you can specify varying time steps using the **Advanced Time Variations** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

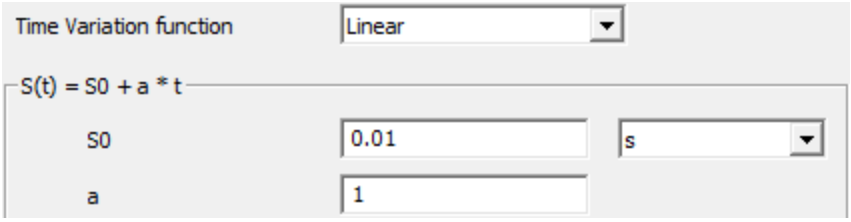


Linear

Linear specifies a linear variation of the time step with time.

$s_t = s_0 + at$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

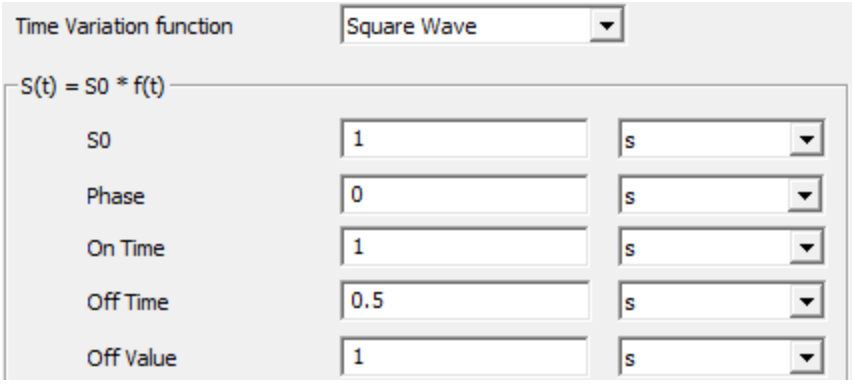


where t is the time, s_t is the value of the variable at time t, s_0 is the value of the variable at $t = 0$, and a is a constant.

Square Wave

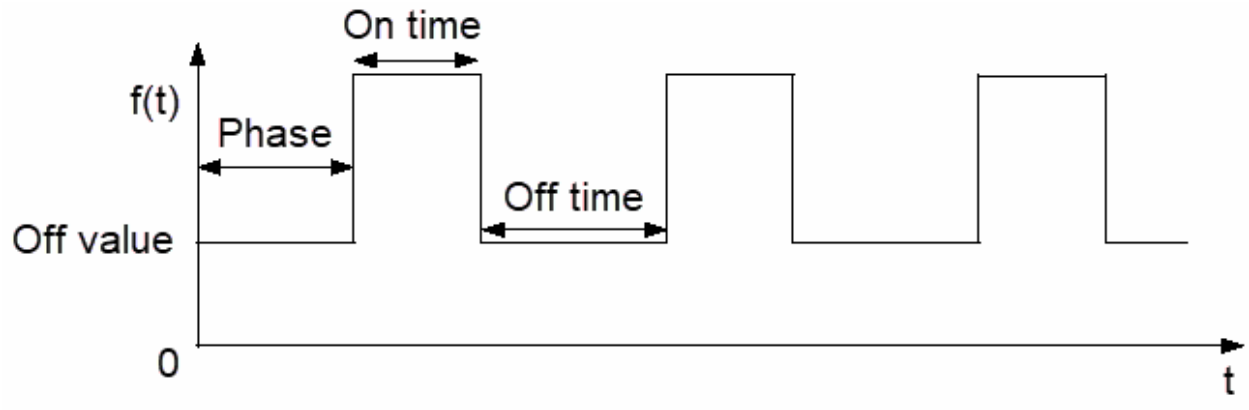
Square Wave specifies a square wave profile for the time step variation. If the variation of time step required is regular/periodic with time, then this option can be used instead of the **Piecewise Constant** option.

$s_t = s_0 f(t)$



where s_t is the value of the variable at time t and s_0 is the value of the variable at $t = 0$. $f(t)$ is a function of time as shown in the following figure.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

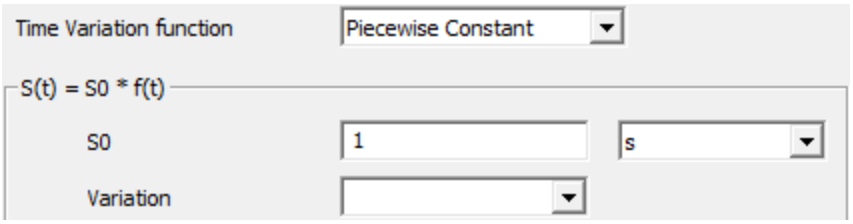


The **Phase** is the time between t = 0 and the first peak of the square wave. The **On Time** is the time that the square wave is at its peak value. The **Off Time** is the time between peak values of the square wave. The **Off Value** is the value of the square waves between the peaks of the wave. The peak value is the specified value of the transient quantity

Piecewise Constant

Piecewise constant specifies a piecewise constant variation of the time step, which uses a dataset of time/time-step pairs.

$s_t = s_0 f(t)$

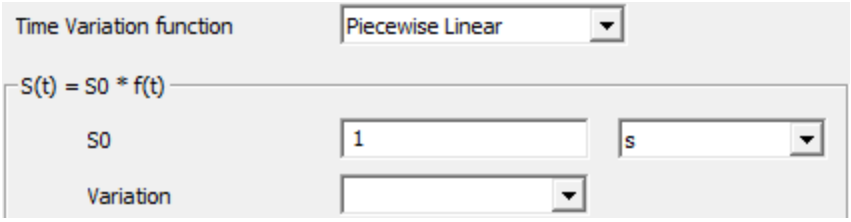


where $f(t)$ is a function of time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0. See Using Datasets to Define Transient Parameters for information on defining $f(t)$ using the **Add Dataset** dialog box. See Using a Piecewise Constant (pwc) Function for more information.

Piecewise Linear

$s_t = s_0 f(t)$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



where $f(t)$ is a function of time, s_t is the value of the variable at time t, s_0 is the value of the variable at t = 0. See Using Datasets to Define Transient Parameters for information on defining $f(t)$ using the **Add Dataset** dialog box.

Automatic

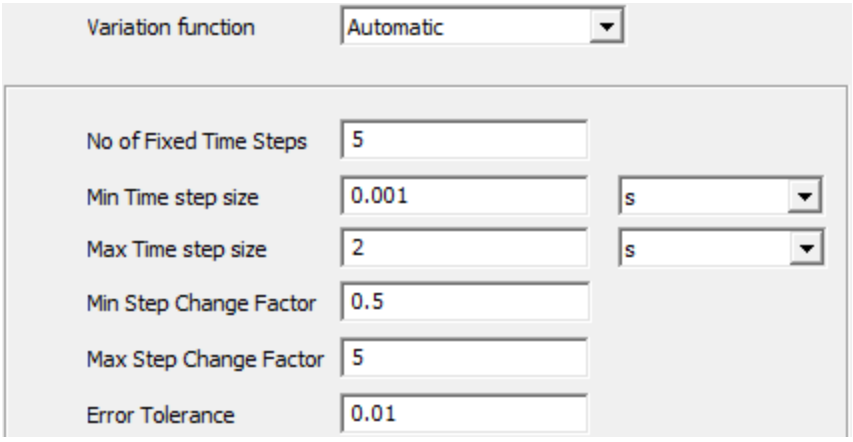
Automatic time step function is determined based on the estimation of the truncation error associated with the time integration scheme. If the truncation error is smaller than a specified tolerance, the size of the time step is increased; if the truncation error is greater, the time step size is decreased.

An estimation of the truncation error can be obtained by using a predictor-corrector type of algorithm in association with the time integration scheme. At each time step, a predicted solution can be obtained using a computationally inexpensive explicit method (forward Euler for the first-order unsteady formulation, Adams-Bashford for the second-order unsteady formulation). This predicted solution is used as an initial condition for the time step, and the correction is computed using the nonlinear iterations associated with the implicit formulation. The norm of the difference between the predicted and corrected solutions is used as a measure of the truncation error. By comparing the truncation error with the desired level of accuracy (that is, the truncation error tolerance), Ansys Icepak is able to adjust the time step size by increasing it or decreasing it.

In cases where the truncation error remains above the specified tolerance, Ansys Icepak will try to meet the tolerance within 5 attempts. If this tolerance is met, then the iteration moves on to the next time step. An explicit scheme is used to predict the solution at each time step, then the explicit prediction is corrected with an implicit scheme. The truncation error, which is a function of the difference between the predicted and corrected solutions at a specific time is used to calculate the next time step. However, if the calculated truncation error is greater than the tolerance limit, we revert from the currently performed iteration, which is moving from the nth step to n+1th step by performing the iteration with a smaller time step. Since the truncation error is proportional to the time step, decreasing the time step reduces the truncation.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying Parameters for Automatic Time Stepping



No of Fixed Time Steps specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for time step. It is a good idea to perform a few fixed-size time steps before switching to automatic time stepping. Sometimes spurious discretization errors can be associated with an impulsive start in time. These errors are dissipated during the first few time steps, but they can adversely affect the adaptive time stepping and result in extremely small time steps at the beginning of the calculation.

> **Important:**
>
> When the solution tends to exhibit incomplete convergence, rather than increasing the time step size or keeping the same time step size in the next step, Ansys Icepak reduces the time step size by at least half for the next time step (making sure that the time step size does not go below the specified minimum time step size.

Min/Max Time step size specify the upper and lower limits for the size of the time step. If the time step becomes very small, the computational expense may be too high; if the time step becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Min/Max Step Change Factor limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step change factor, , is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step is computed as follows:

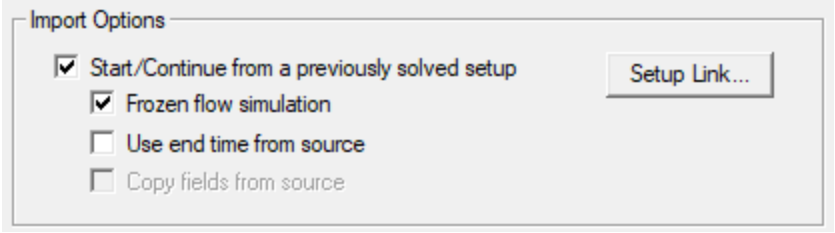
- If $1 < f < f_{max}$, Δt_n is increased to meet the desired tolerance.
- If $1 < f_{max} < f$, Δt_n is increased, but its maximum possible value is $f_{max} \Delta t_{n-1}$.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If $f_{min} < f < 1$, Δt_n is unchanged.
- If $f < f_{min} < 1$, Δt_n is decreased.

Error Tolerance specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

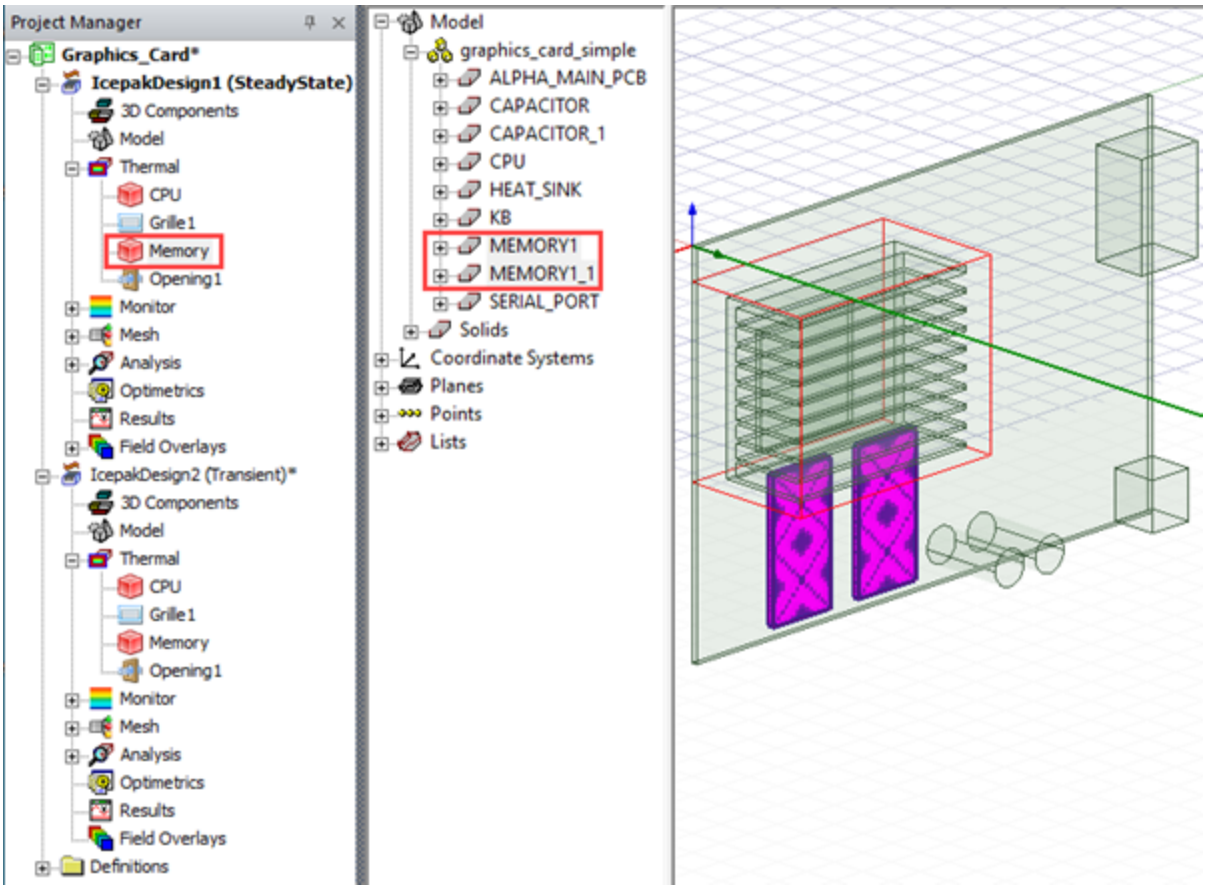
Transient Import Options Requirements

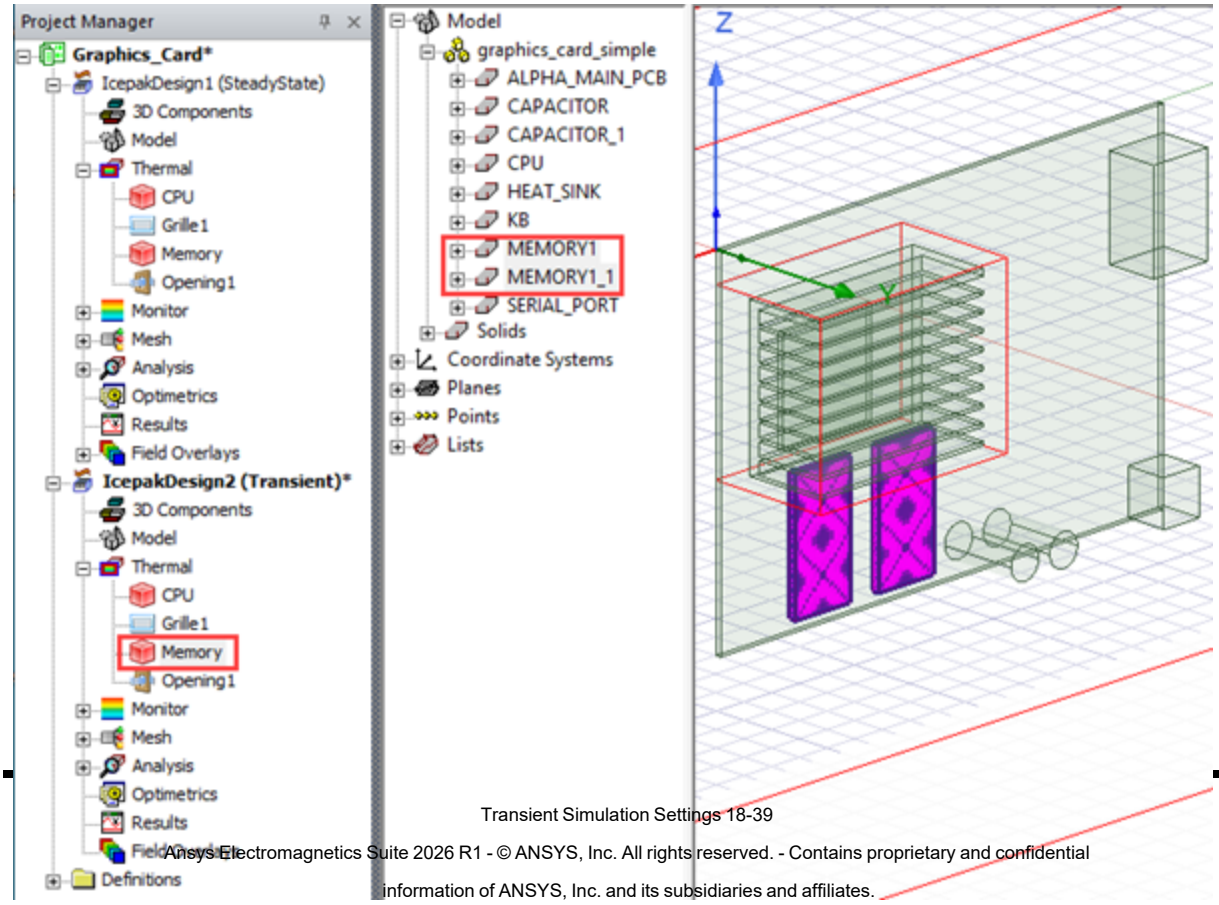


To use the **Start/Continue from a previously solved setup** options, both the source steady state and target transient designs must contain identical model geometry and boundary condition assignments. See the following images. In this example, note the identical geometry and that Memory boundary condition consists of the identical assigned geometry.

> **Note:** The following changes to boundary condition or native 3D component parameters in your design are not supported when using **Start/Continue from a previously solved setup**:
>
> - In Openings, changing the pressure specification to velocity (or vice versa).
> - In Fans, changing the direction of the fan flow.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

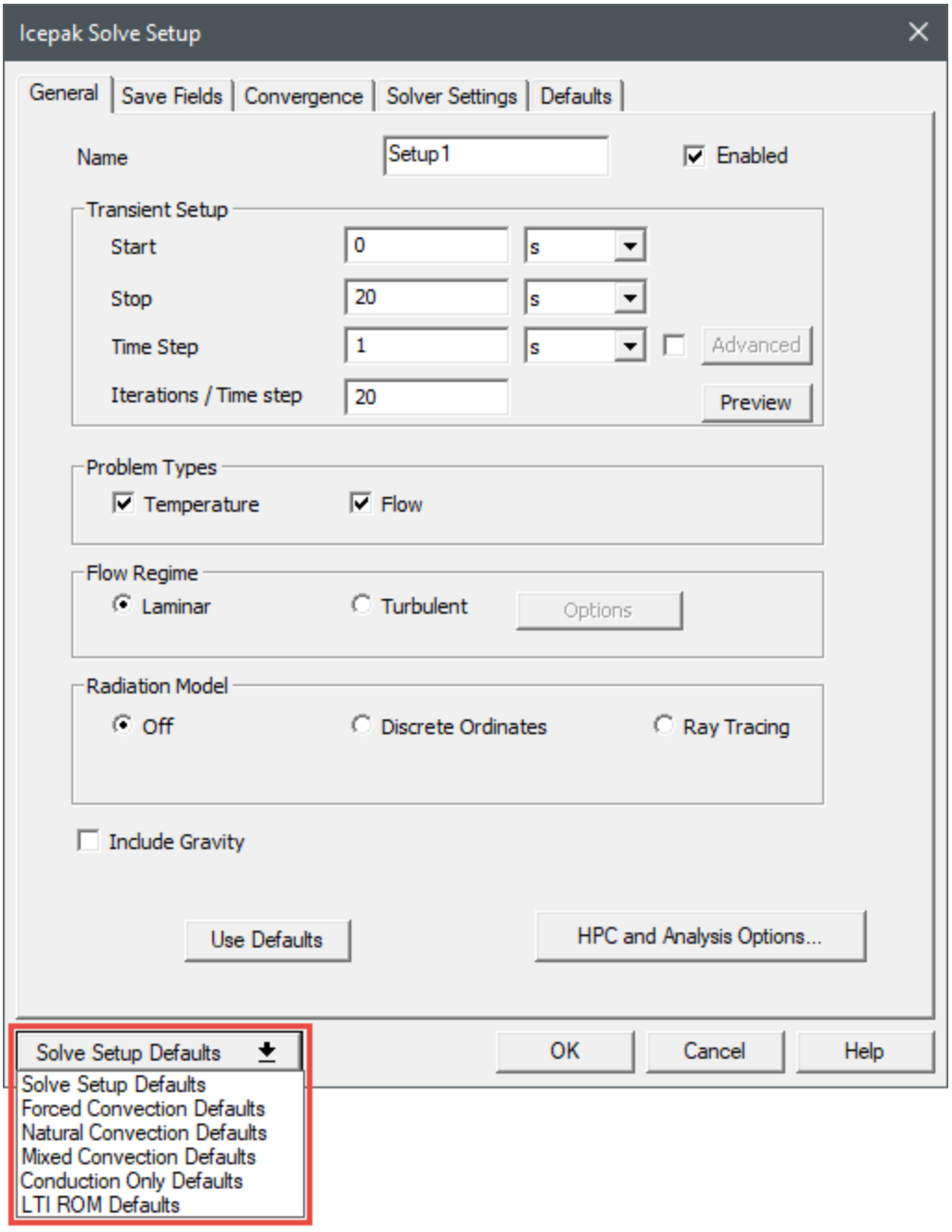




Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Icepak Solve Setup Defaults

The **Icepak Solve Setup Defaults** drop-down list contains options that automatically define solution settings based on the selected simulation type.



The following sections detail which settings are defined for each simulation type.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The **Solve Setup Defaults** options restores all settings to the application defaults.

Forced Convection

Select **Forced Convection Defaults** if your simulation includes forced convection, in which fluid moves within the computational domain by a source (for example, a fan). The following solution settings are defined as forced convection defaults.

Note:

Forced Convection Defaults does not appear as an option if the **Solution Type** is set to **Temperature Only**. The settings automatically defined depend on the **Solution Type**.

Solution Type = Flow Only	
Maximum Number of Iterations	100
Problem Types	**Temperature** and **Flow** enabled
Radiation Model	**Off**
Under-relaxation	**Pressure** = 0.3 and **Momentum** = 0.7
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Solution Type = Temperature and Flow	
Maximum Number of Iterations	100
Problem Types	**Temperature** and **Flow** enabled
Radiation Model	**Off**
Gravity	Disabled
Solve Flow and Energy Equations Separately	Enabled
Under-relaxation	**Pressure** = 0.3 and **Momentum** = 0.7
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Convergence	**Energy** = 1e-12
Linear Solver Options: Temperature	**Termination Criterion** = 1e-06 and **Residual Reduction Tolerance** = 1e-06

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Natural Convection

Select **Natural Convection Defaults** if your simulation includes natural convection, in which fluid moves within the computational domain by changes in the density of the fluid due to variations in temperature. The following solution settings are defined as natural convection defaults.

> **Note:**
>
> **Natural Convection Defaults** does not appear as an option if the **Solution Type** is set to **Temperature Only** or **Flow Only**.

Maximum Number of Iterations	100
Problem Types	**Temperature** and **Flow** enabled
Include Gravity	Enabled
Under-relaxation	**Pressure** = 0.7 and **Momentum** = 0.3
Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Initial Conditions: X, Y, or Z Velocity	-/+ 1e-4 depending on the direction of the **Gravity Vector** defined in the **Design Settings**
Radiation Model	**Discrete Ordinates** enabled
Flow Iterations per Radiation Iteration	10
Angular Discretization Settings	2

Mixed Convection

Select **Mixed Convection Defaults** if your simulation includes forced convention and natural convection. The following solution settings are defined as mixed convection defaults.

> **Note:**
>
> **Mixed Convection Defaults** does not appear as an option if the **Solution Type** is set to **Temperature Only** or **Flow Only**.

Maximum Number of Iterations	100
Problem Types	**Temperature** and **Flow** enabled
Include Gravity	Enabled
Solve Flow and Energy Equations Separately	Disabled
Under-relaxation	**Pressure** = 0.7 and **Momentum** = 0.3

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discretization Scheme	**Pressure** = **Standard** and **Momentum** = **First**
Initial Conditions: X, Y, or Z Velocity	-/+ 1e-4 depending on the direction of the **Gravity Vector** defined in the **Design Settings**
Radiation Model	**Discrete Ordinates** enabled
Flow Iterations per Radiation Iteration	10
Angular Discretization Settings	2

Conduction Only

Select **Conduction Only Defaults** if your simulation includes only temperature analysis. The following solution settings are defined as conduction only defaults.

> **Note:**
>
> **Conduction Only Defaults** does not appear as an option if the **Solution Type** is set to **Flow Only**.

Problem Types	**Temperature** enabled and **Flow** disabled
Maximum Number of Iterations	40
Convergence	**Energy** = 1e-12
Linear Solver Options: Temperature	**Termination Criterion** = 1e-06 and **Residual Reduction Tolerance** = 1e-06

LTI ROM

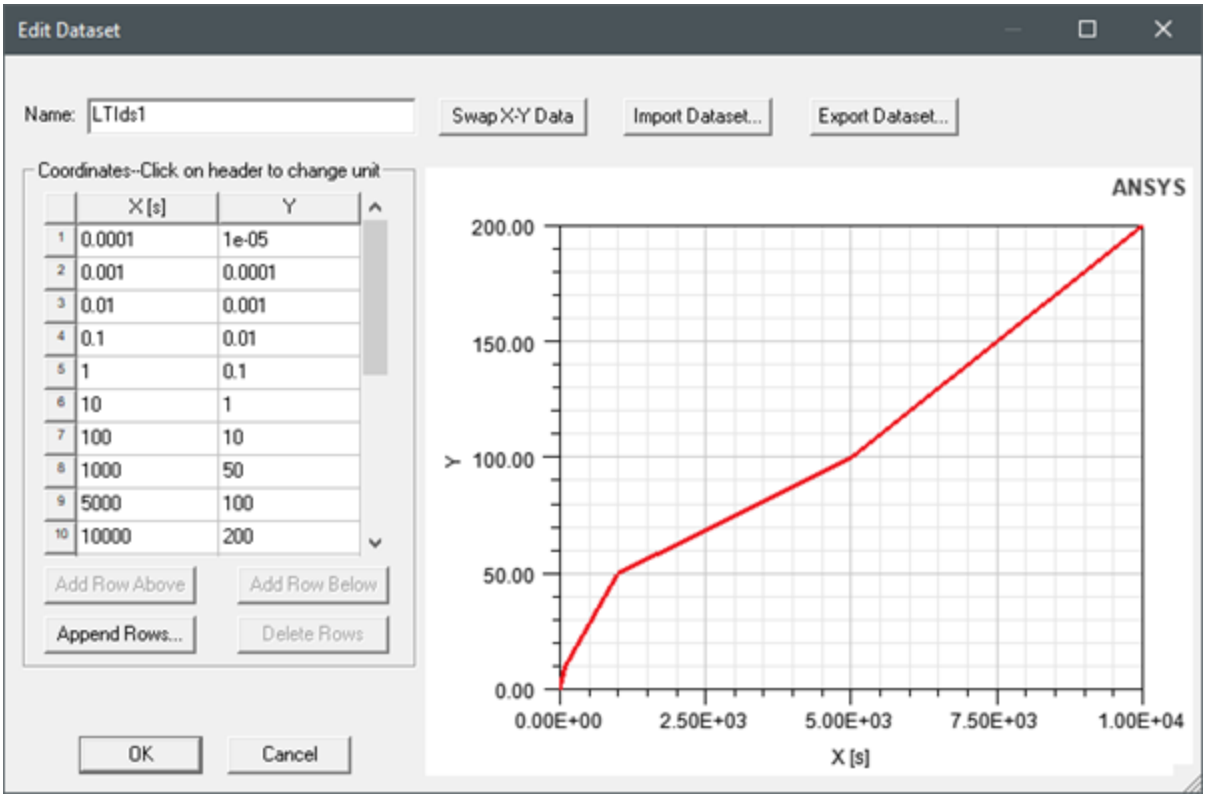
Select **LTI ROM Defaults** if your transient simulation includes a parametric analysis intended for LTI ROM export. The following solution settings are defined as LTI ROM defaults.

General Tab	
Transient Setup	**Start** = 0, **Stop** = 10000, **Time Step** = Advanced (see **Transient Variation** below), and **Iterations/Time Step** = 20
Transient Variation	**Variation function** = Piecewise Constant, **S0** =1, and **Variation** = dataset (see dataset values below)
Save Fields	
Field save times	Every 1000 steps

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When you select **LTI ROM Defaults**, a dataset with preset defaults is created and assigned to the piecewise constant variation function.



Using Solved Simulation Data

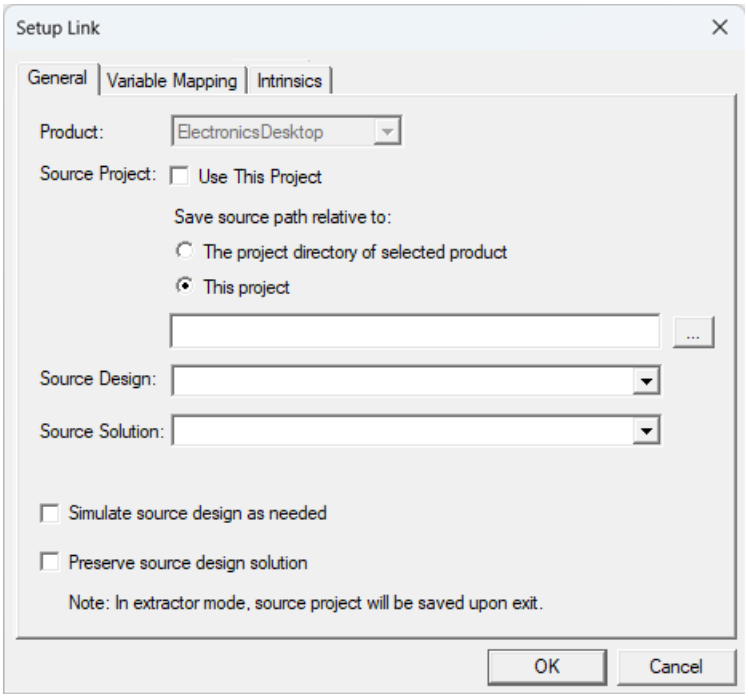
An Icepak analysis can be started or continued based on a previously solved setup in an Icepak design. Mesh from the selected design is automatically imported and used in the simulation.

Note: You can import mesh without starting or continuing a simulation from a previously solved setup. See Add Mesh Link for more information.

1. On the **Icepak Solve Setup Dialog Solver Settings** tab, select **Start/Continue from a previously solved setup** and then click **Setup Link**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

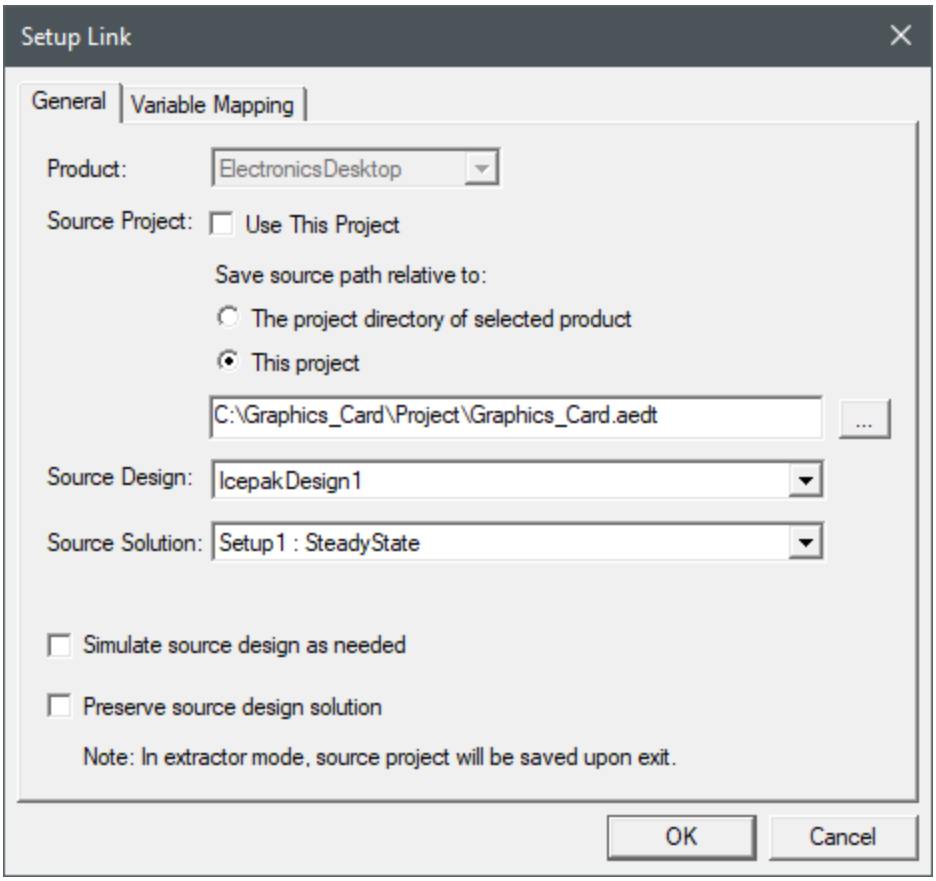
The **Setup Link** dialog box is displayed.



2. Specify the project that has the source design in one of the following ways:
    a. Check **Use This Project** to pick a design from a current working project.
    b. Click the ellipsis and browse your computer to pick the design.

> **Note:**
>
> The option **Use this project** points to the directory relative to where your current working project is located.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

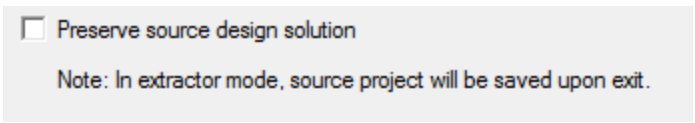


3. Pick the source design from the **Source Design** drop-down menu which lists all the designs in the specified project.
4. Pick the setup type from the **Source Solution** pull-down menu (SteadyState or Transient).
5. Select the last two check boxes as needed and click **OK**.

> **Note:**
>
> Go to Simulate source design as needed, for more information about that option.

Preserve source design solution

This is the last option on the **Setup Link** dialog box. By default, the source design results are not saved. Select this option to save the source design results.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simulate source design as needed

Select this option if you modify the source design and you want an updated source design solution to be linked to the target design. If you select this option, make sure that you also invalidate the cached link data in the target. To do this, right-click **Analysis** and select the option **Clear Linked Data** from the short-cut menu.



6. Click the **Variable Mapping** tab and ensure the **Variable Mapped to Source Design** values are as desired for each variable.

> **Note:**
>
> You can click **Map Variable by Name** to compare the source design variables to the Icepak design variables. Any variables with the same name are mapped so that the source design conforms to the Icepak design.

7. Click **OK** to close the **Setup Link** dialog.

Monitoring a Transient Solution

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 a profile of the simulation as well as convergence data during or after the solution process.

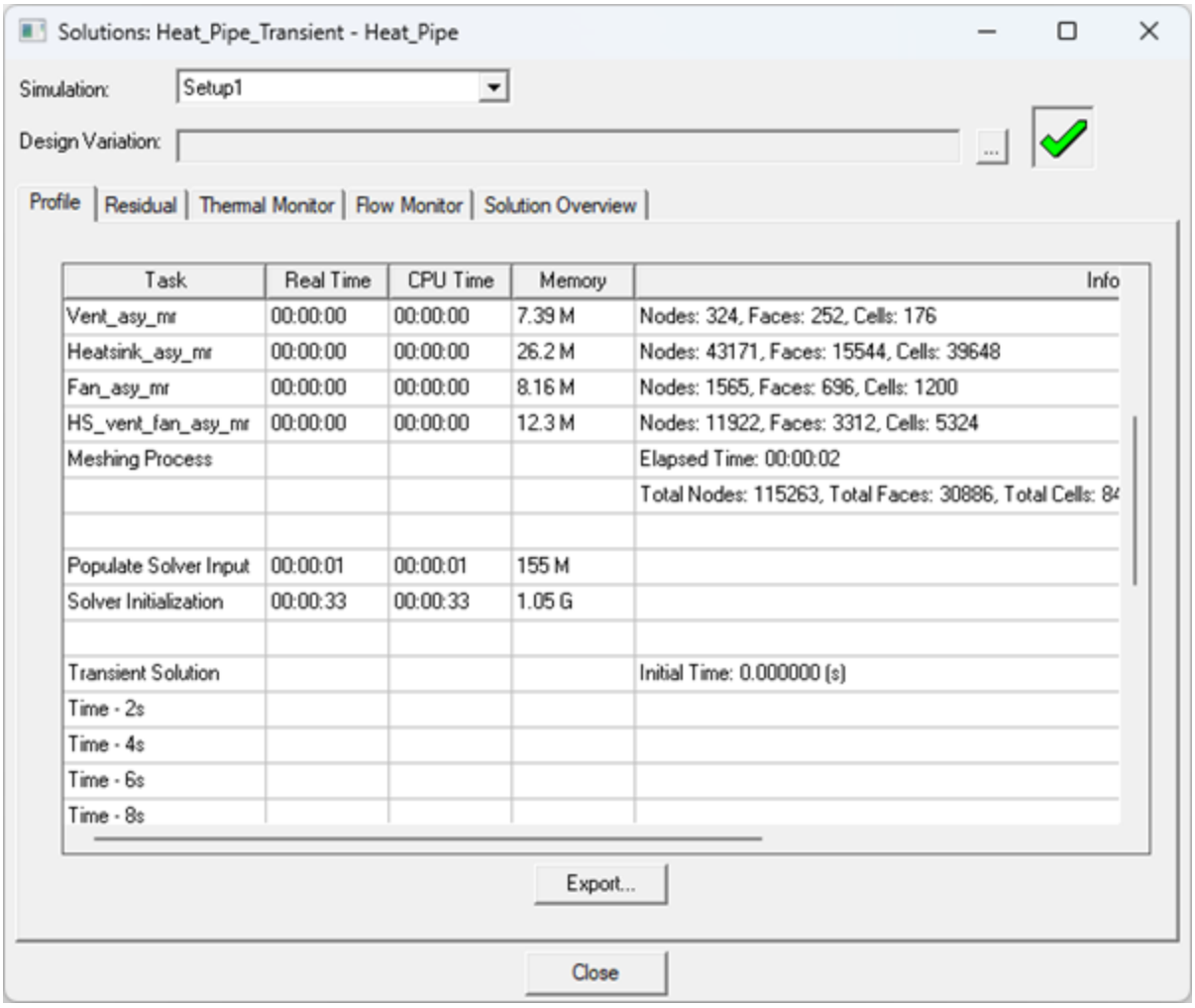
- The solution profile includes meshing information (memory, times, and mesh counts per region), solver times (input, initialization, and solve), surface and volume EM loss data, exported AC loss data from HFSS 3D Layout, and total solution process.
- The solution residuals include those for continuity, velocity, energy, and joule heating (if enabled).
- The solution monitor includes data plotted for any defined thermal and flow monitor points.
- The solution overview includes thermal and flow data for boundary conditions and fan components.

> **Note:**
>
> For solution overviews, Heatsink, PCB, CTM, and 3D-IC components are not currently supported.

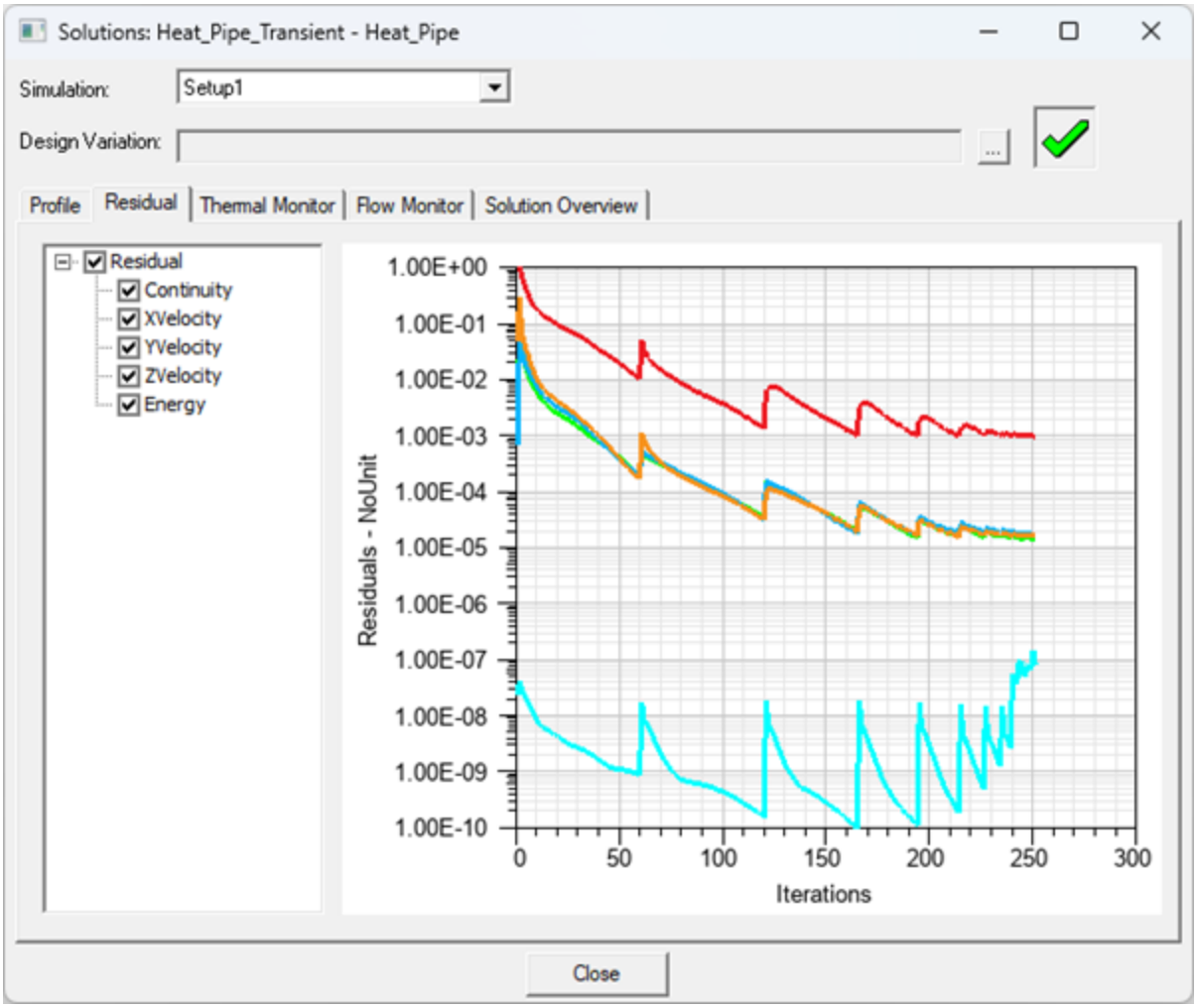
To view the solution's profile information or convergence data:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

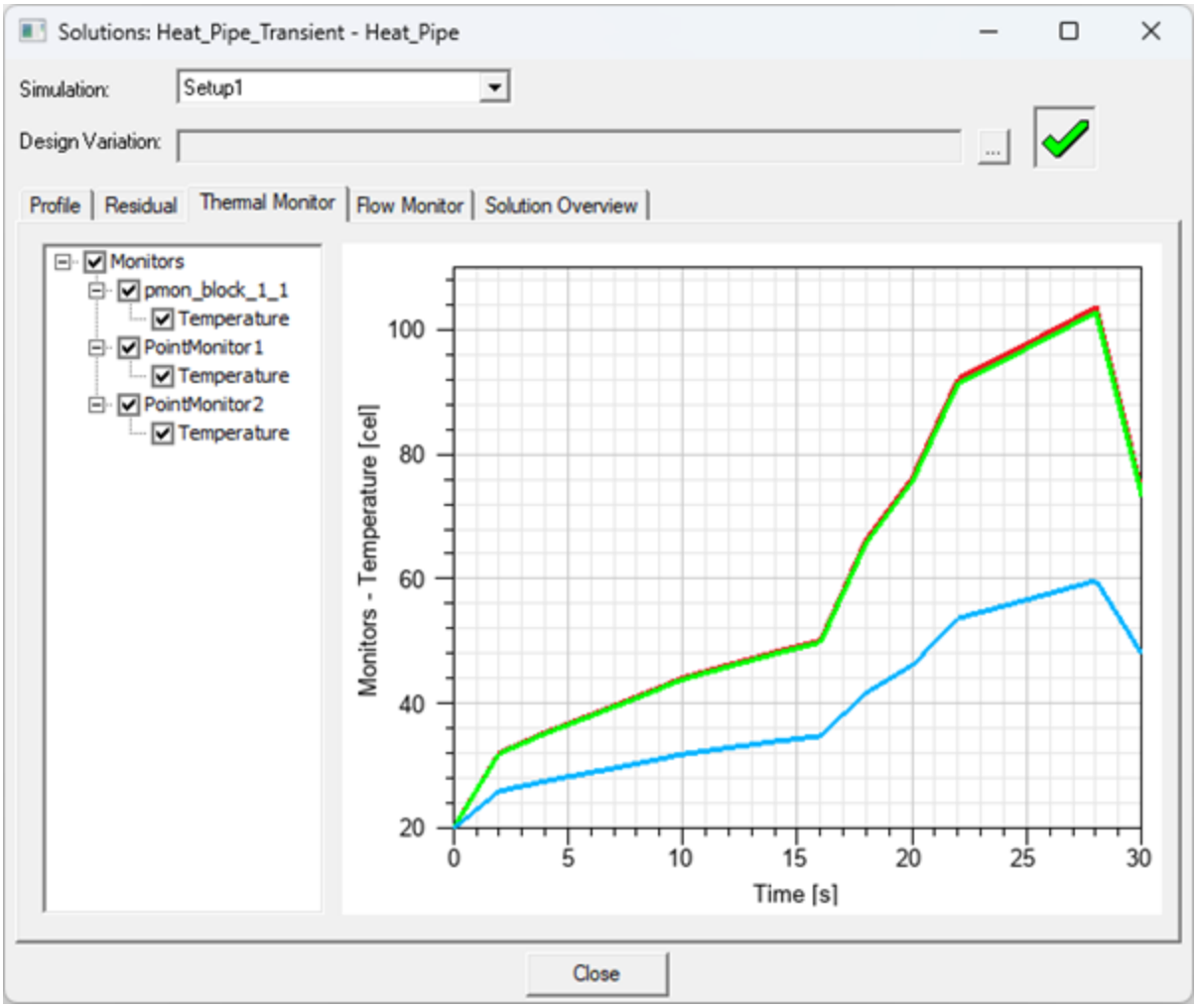
1. In the **Project Manager**, expand **Analysis**.
2. Right-click the solution setup and select **Profile**, **Residual**, **Thermal Monitor** or **Flow Monitor**.
3. In the **Solutions** dialog box, toggle between the **Profile**, **Residual**, **Thermal Monitor**, **Flow Monitor**, and **Solution Overview** tabs to view the information. See "Solution Overview" on page 18-52 for details on the data displayed on the **Thermal**, **Flow**, and **Fan** tabs.
4. For Parametric trials, click [**...**] next to **Design Variation** to open the **Set Design Variation** dialog box and choose the variation for which to display data.



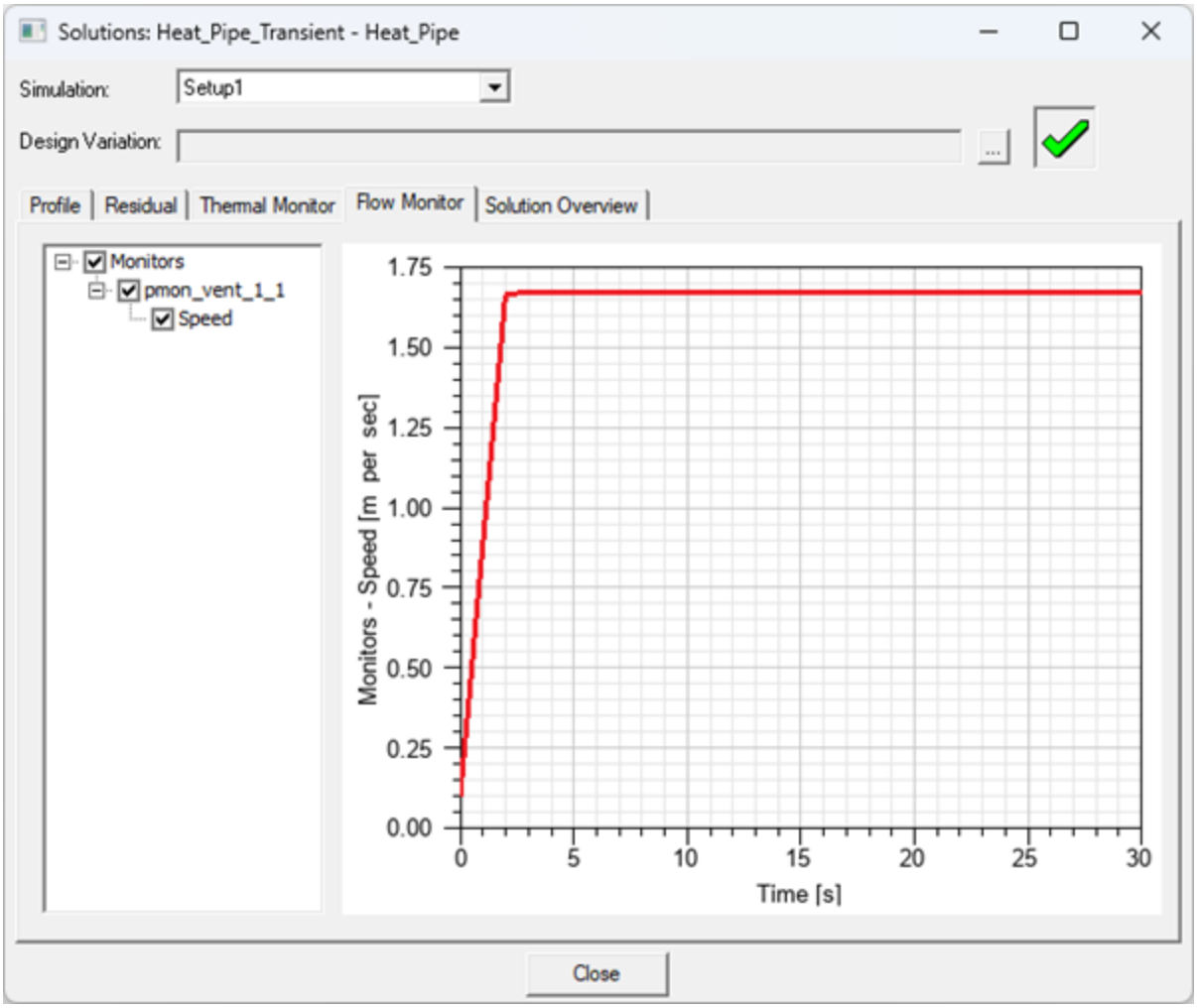
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



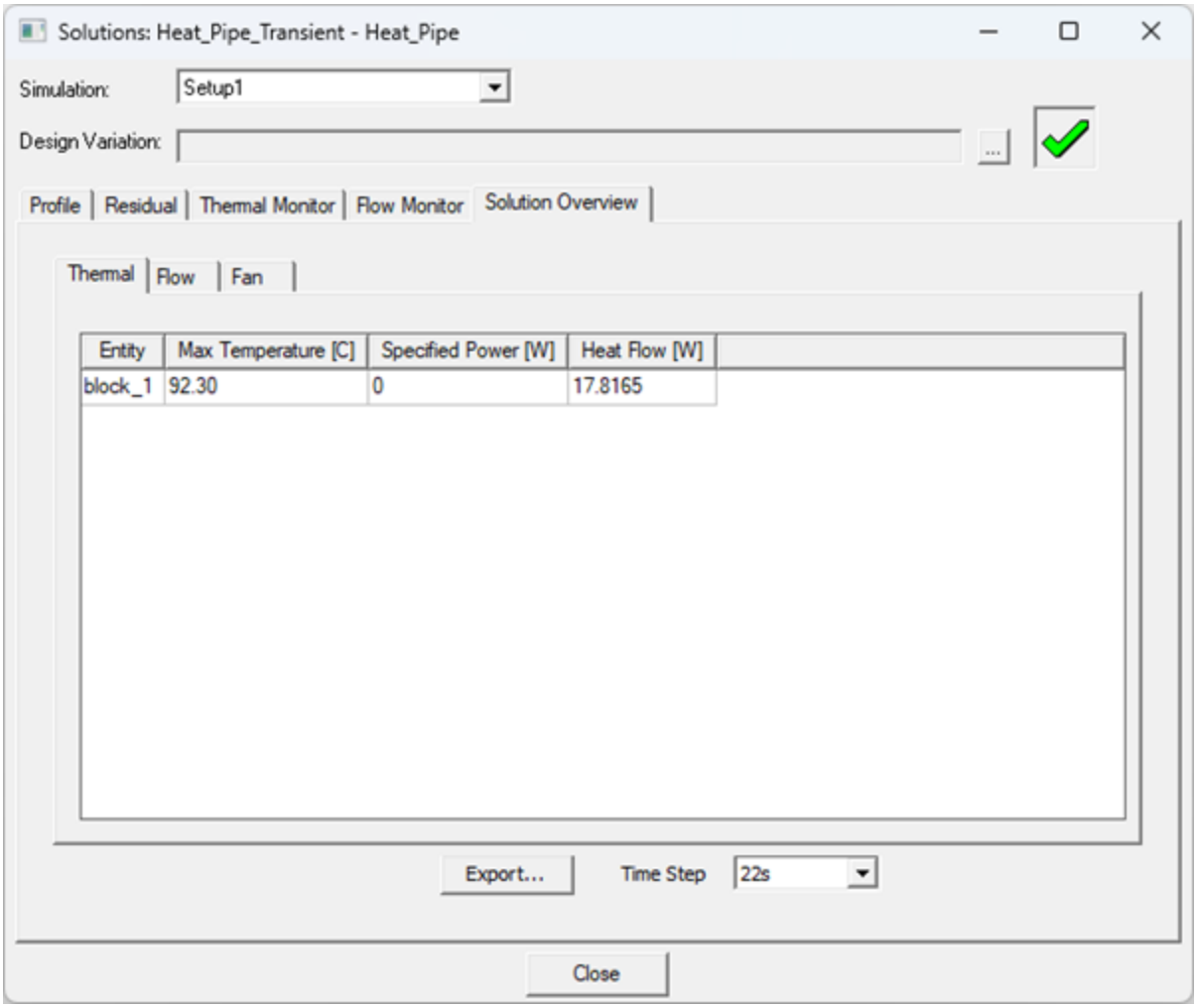
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> For transient simulations, the **Thermal Monitor** and **Flow Monitor** tabs plots the thermal and flow curves for each defined time step.

Solution Overview

The solution overview has three tabs that display thermal, flow, and fan data. Select a **Time Step** from the drop-down list to view data from different time steps in the transient solution. Toggle between the **Thermal**, **Flow**, and **Fan** tabs to view the information. Click **Export** to generate a plain text (.txt) file containing the data. For parametric trials, click **[...]** next to **Design Variation** to open the **Set Design Variation** dialog box and choose the variation for which to display data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Data in the solution overview is cleared if you delete a boundary condition or component in the report or if you clean up solution data.

Thermal Tab

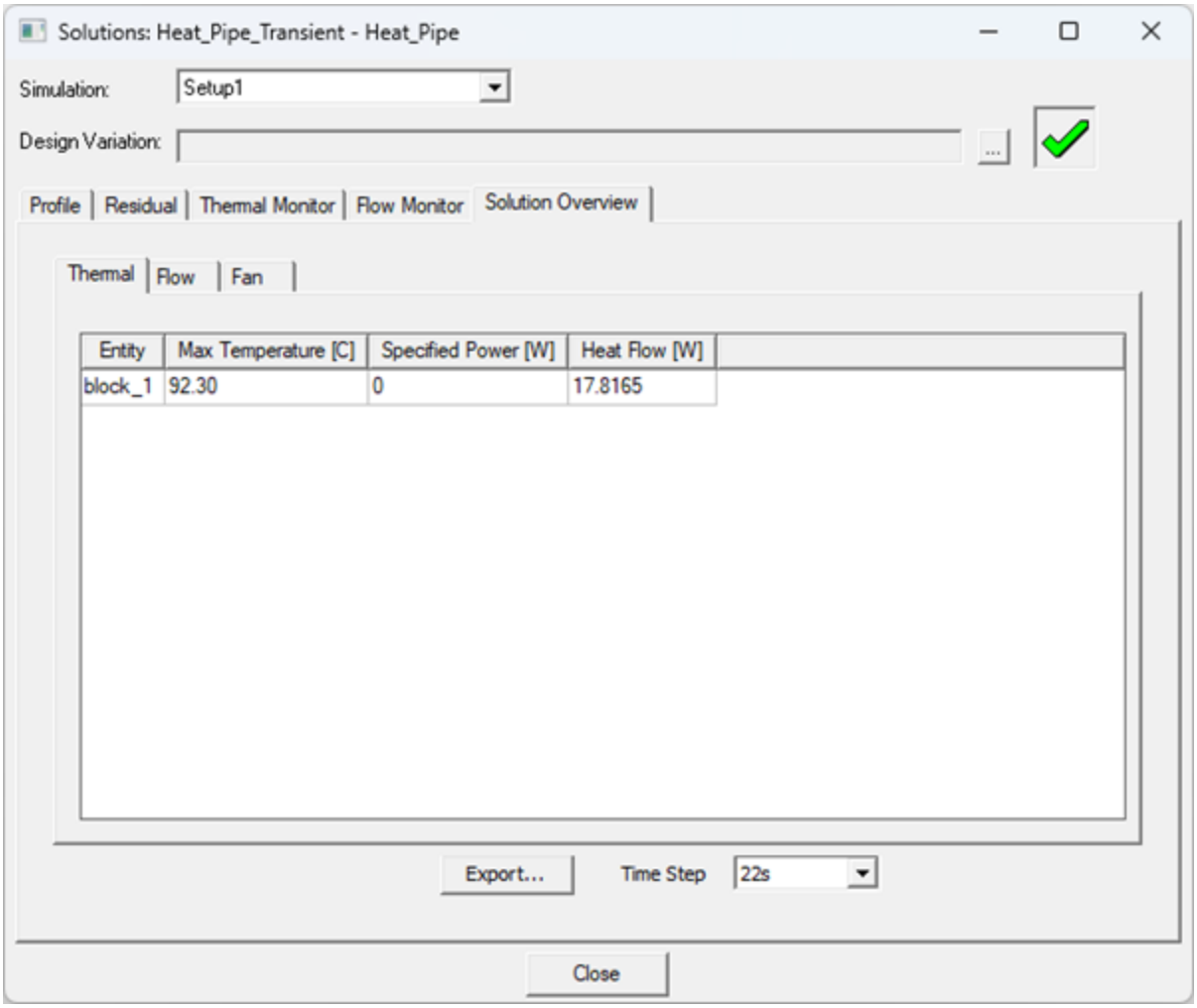
The **Thermal** tab displays data for boundary conditions, including Blocks, Networks, Plates, Sources, Em Losses, Resistances, and Walls. Data is also displayed for external 3D components with supported thermal boundary conditions.

- Max Temperature
- Specified Power

  > **Note:**
  >
  > If a dataset is used to specify power, a default value of zero is displayed.

- Heat Flow

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



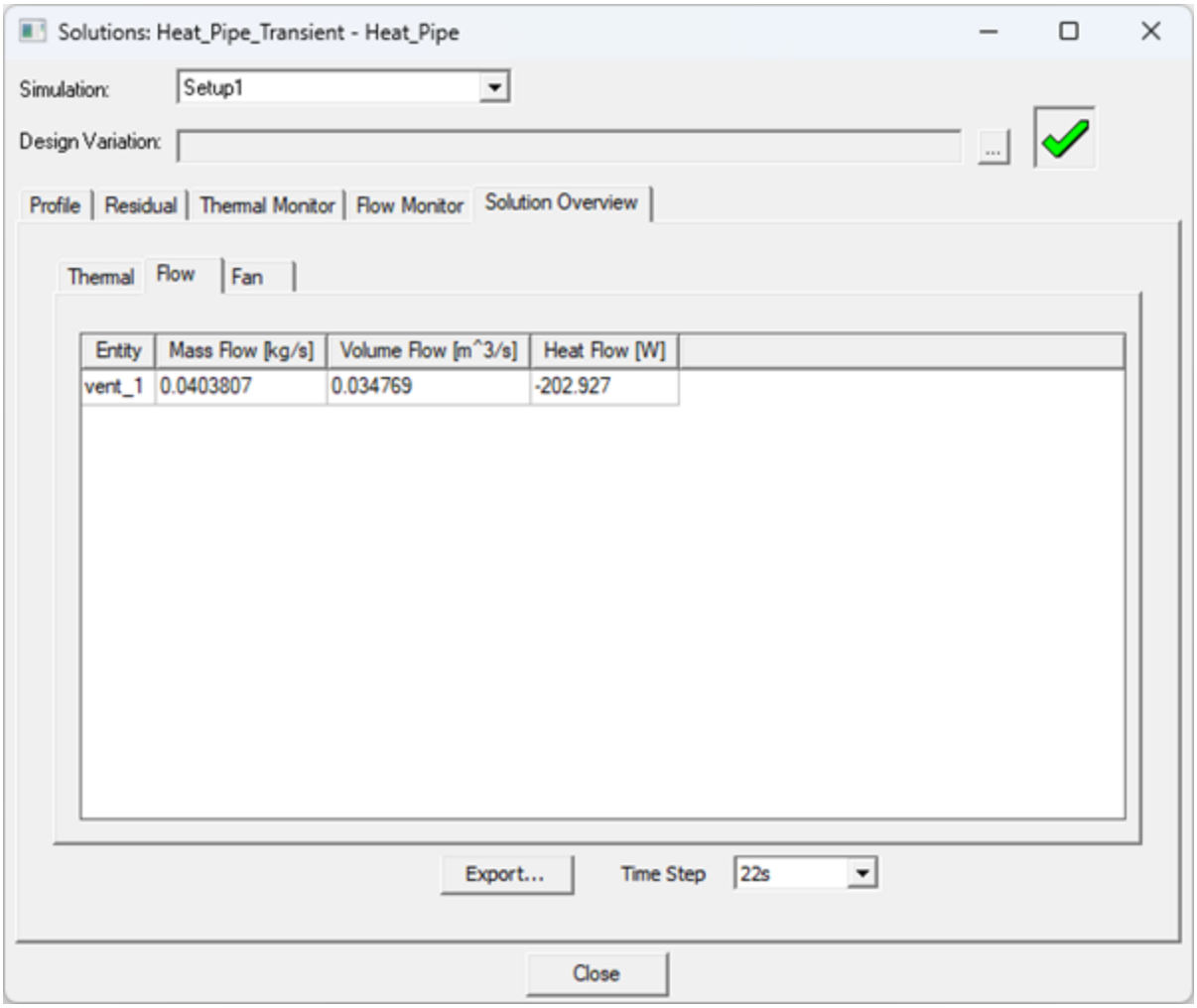
Flow Tab

The **Flow** tab displays data for boundary conditions, including Openings, Grilles, and Blowers. Data is also displayed for external 3D components with supported flow boundary conditions.

- Mass Flow
- Volume Flow
- Heat Flow

> **Note:**
>
> For openings on the external boundary of the computational domain, the heat flow rates are reported at reference temperature in Fluent (25°C).

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

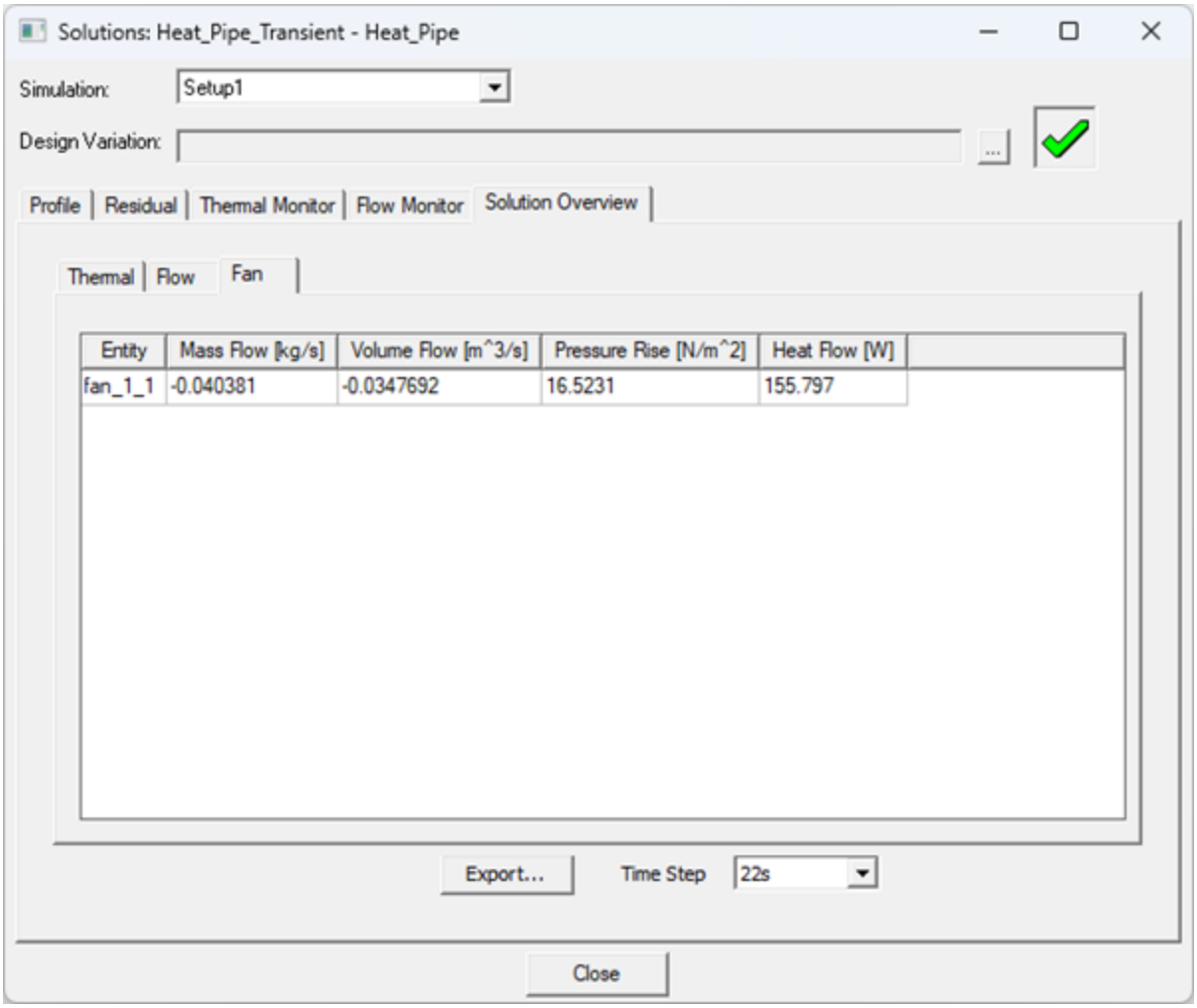


Fan Tab

The **Fan** tab displays data for fan components in the design (if any). Data is also displayed for external 3D components with fan components.

- Mass Flow
- Volume Flow
- Pressure Rise
- Heat Flow

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Postprocessing Transient Solutions

The time steps defined by the Icepak Solve Setup Dialog are available for creating postprocessing reports, plots, animations, and calculations.

Create a Fields Summary

The following example assigns a block boundary condition to box geometry and uses the function to define the **Total Power** setting.

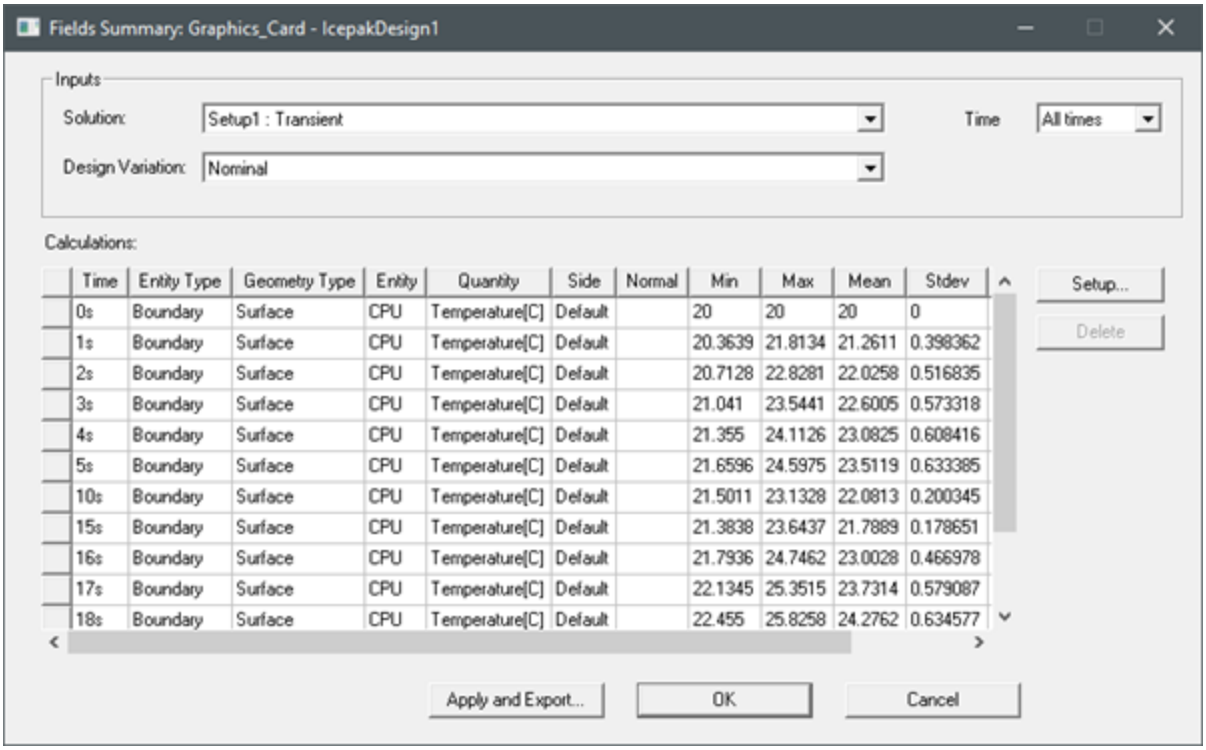
1. From the **Icepak** menu, select **Fields** > **Create Fields Summary**.
2. In the **Fields Summary** dialog box for **Solution**, select the transient solution setup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Time** drop-down list, select the time step at which you want to add calculations.

> **Note:**
>
> Select **All times** to add the calculations at each time step.

4. In the **Setup Calculation** dialog box, select the following:
   - **Entity Type**
   - **Geometry Type**
   - **Entity**
   - **Quantity**
5. Click **Add** and select **Add as Single Calculation** or **Add as Multiple Calculations**.
6. Repeat steps 3 through 5 for each desired time step. The following example displays the surface temperature of the CPU at each defined time step.



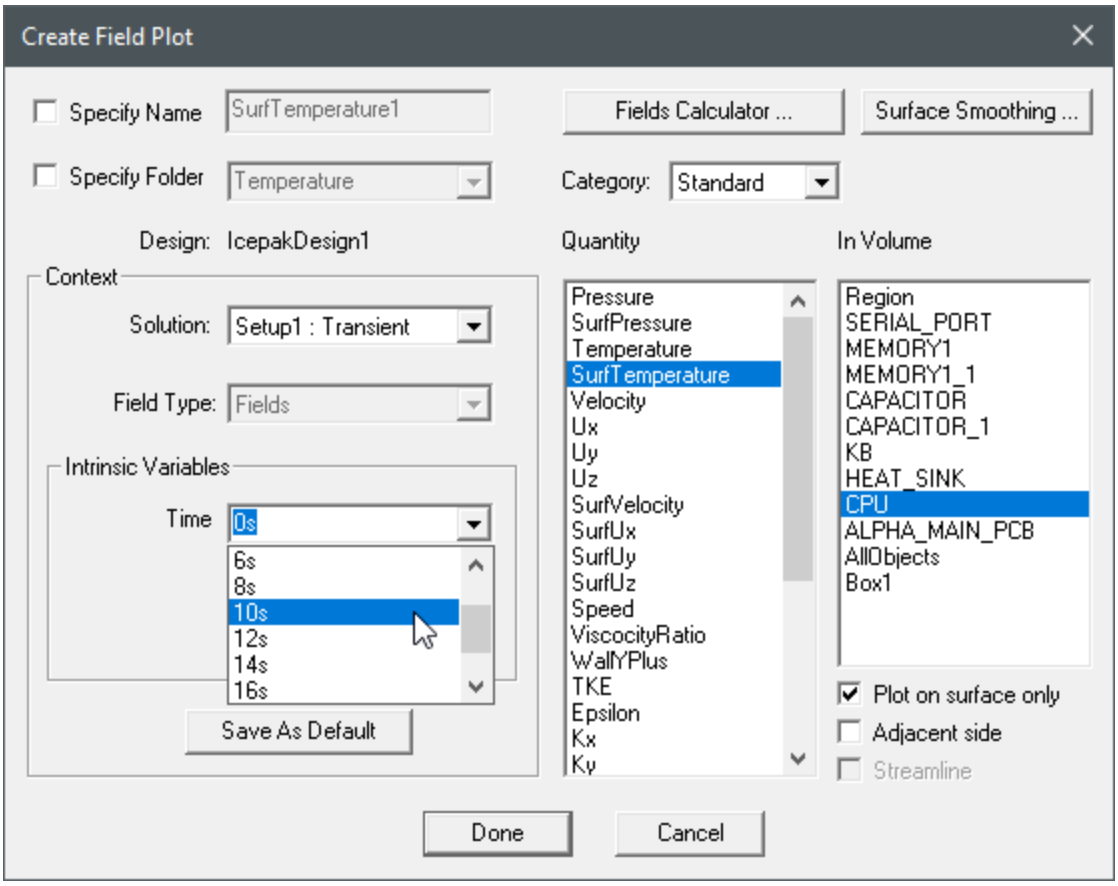
Time	Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev
0s	Boundary	Surface	CPU	Temperature[C]	Default		20	20	20	0
1s	Boundary	Surface	CPU	Temperature[C]	Default		20.3639	21.8134	21.2611	0.398362
2s	Boundary	Surface	CPU	Temperature[C]	Default		20.7128	22.8281	22.0258	0.516835
3s	Boundary	Surface	CPU	Temperature[C]	Default		21.041	23.5441	22.6005	0.573318
4s	Boundary	Surface	CPU	Temperature[C]	Default		21.355	24.1126	23.0825	0.608416
5s	Boundary	Surface	CPU	Temperature[C]	Default		21.6596	24.5975	23.5119	0.633385
10s	Boundary	Surface	CPU	Temperature[C]	Default		21.5011	23.1328	22.0813	0.200345
15s	Boundary	Surface	CPU	Temperature[C]	Default		21.3838	23.6437	21.7889	0.178651
16s	Boundary	Surface	CPU	Temperature[C]	Default		21.7936	24.7462	23.0028	0.466978
17s	Boundary	Surface	CPU	Temperature[C]	Default		22.1345	25.3515	23.7314	0.579087
18s	Boundary	Surface	CPU	Temperature[C]	Default		22.455	25.8258	24.2762	0.634577

Create a Field Plot

The following example plots the surface temperature of a CPU at the 10 second time step. Other quantities are available for plotting fields in the **Plot Fields** sub-menus for a transient solution. For more information about creating animations, see Plotting Field Overlays.

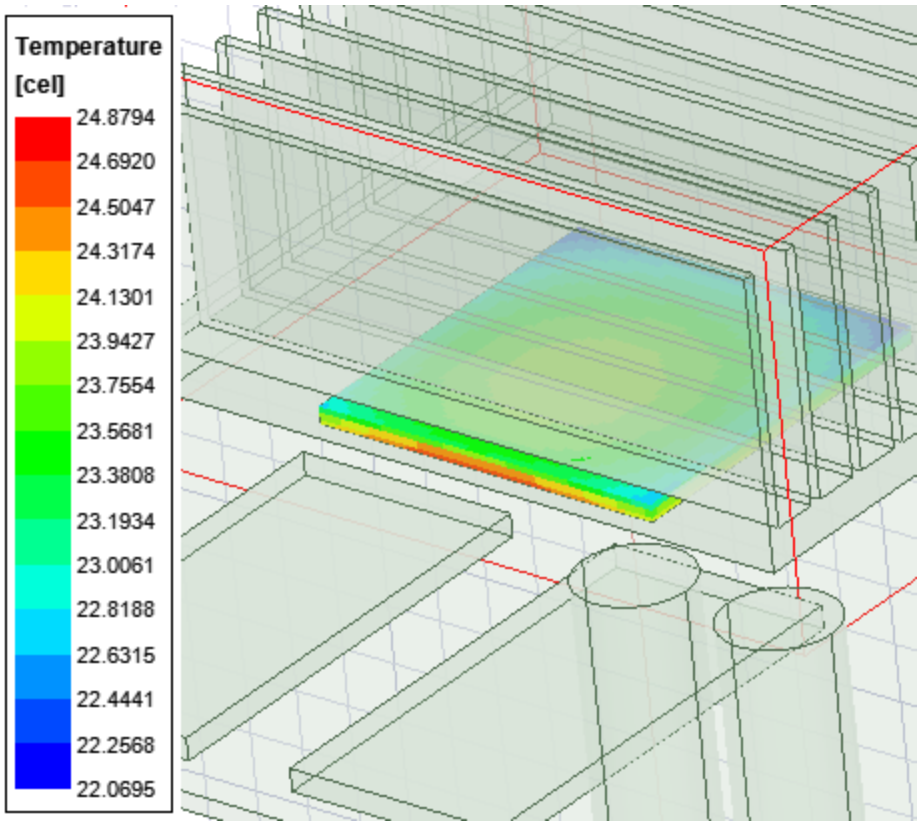
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In object-selection mode, right-click the geometry in the history tree or **3D Modeler** window.
2. From the right-click menu, select **Plot Fields** > **Temperature** > **SurfTemperature**.
3. In the **Create Field Plot** dialog box under **Context**, ensure the **Solution** is set to the transient solution.
4. Under **Intrinsic Variables**, select **10s** for **Time**.
5. Under Quantity, retain the selection of **SurfTemperature**.
6. Under **In Volume**, select **CPU** and select **Plot on surface only**.



7. Click **Done**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

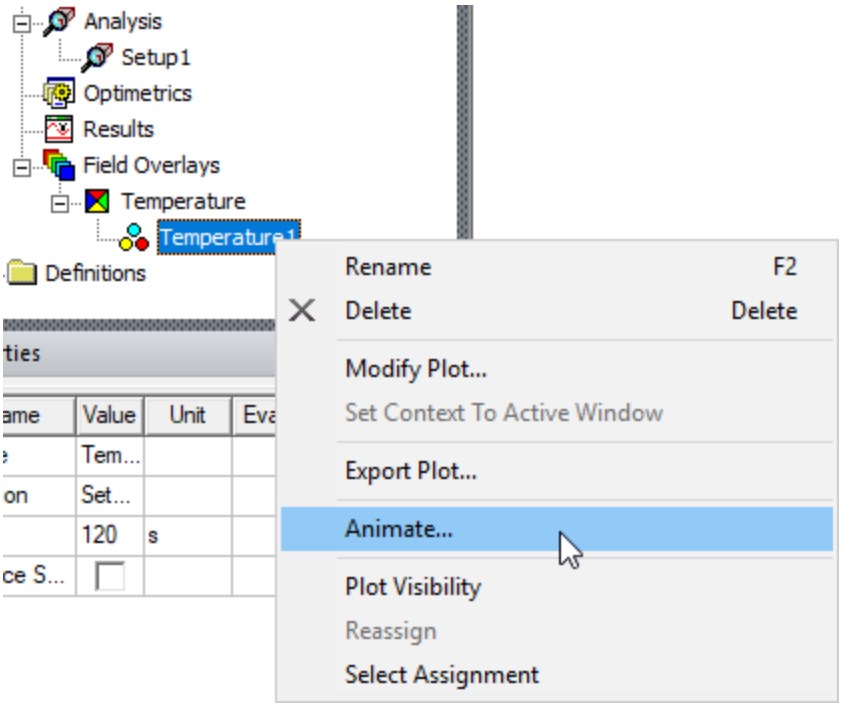


Animate a Field Plot

The following example animates the temperature of a heatsink at all time steps. For more information about creating animations, see Creating Animations.

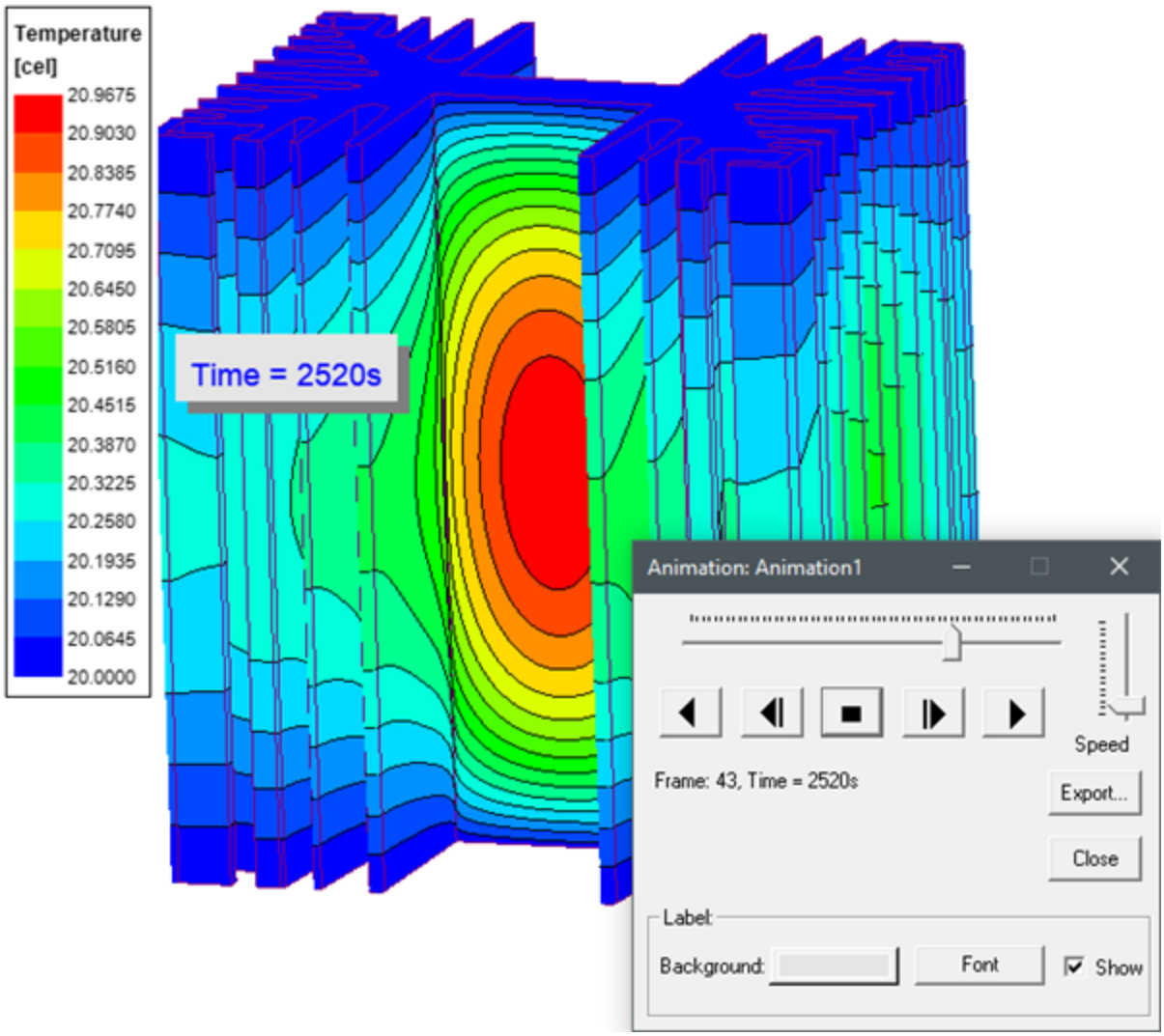
1. Create a field plot (for example, as in "Create a Field Plot" on page 18-57 above).
2. In the **Project Manager**, right-click on the field plot and select **Animate**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the **Create Animation Setup** dialog box, retain the default setting and click **OK**. The animation is displayed in the **3D Modeler** window.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create a Report

When creating data tables and reports for a transient simulation, data for each defined time step is displayed. See the following examples of monitor data table and a calculator expressions plot. For more information about reports, see Creating Reports.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

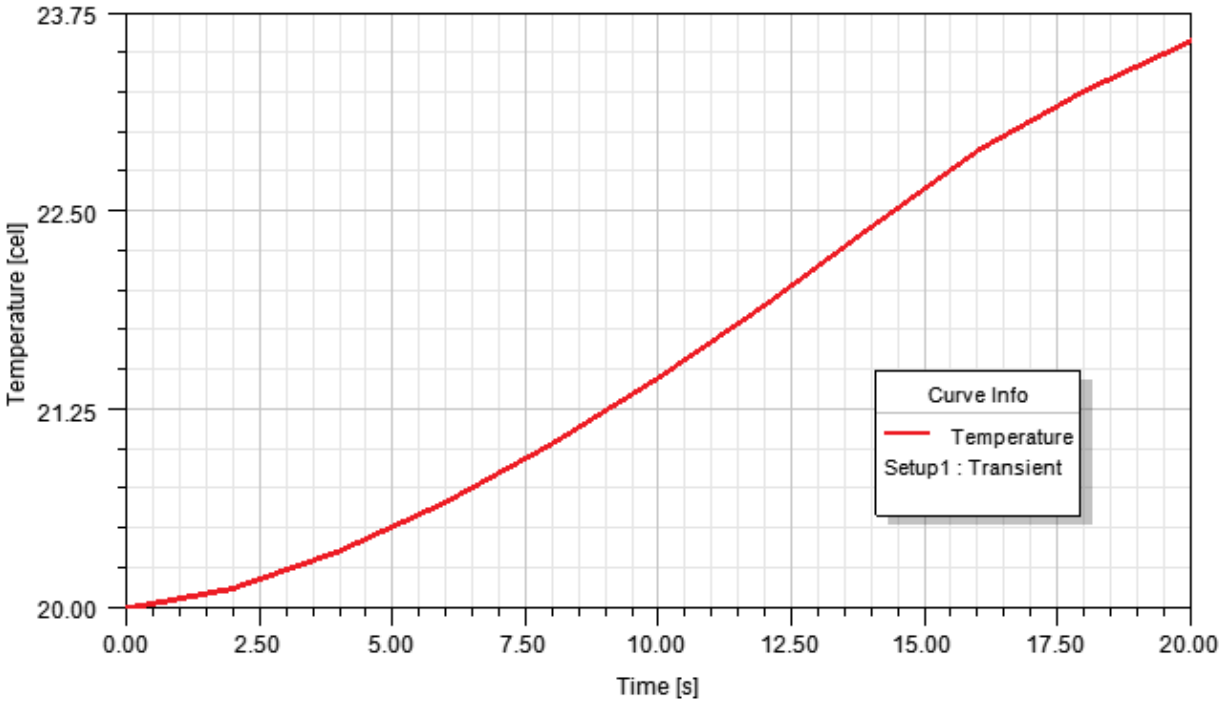
Monitor Table 1

IcepakDesign1 ANSYS

	Time [s]	PointMonitor2.Temperature [cel] Setup1 : Transient	PointMonitor3.Speed [m_per_sec] Setup1 : Transient
1	0.000000	32.781122	13.500411
2	2.000000	32.781122	13.500411
3	4.000000	32.781122	13.500411
4	6.000000	32.781122	13.500411
5	8.000000	32.781122	13.500411
6	10.000000	32.781122	13.500411
7	12.000000	32.781122	13.500411
8	14.000000	32.781122	13.500411
9	16.000000	32.781122	13.500411
10	18.000000	32.781122	13.500411
11	20.000000	32.781122	13.500411

Calculator Expressions Plot 1

IcepakDesign1 ANSYS



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

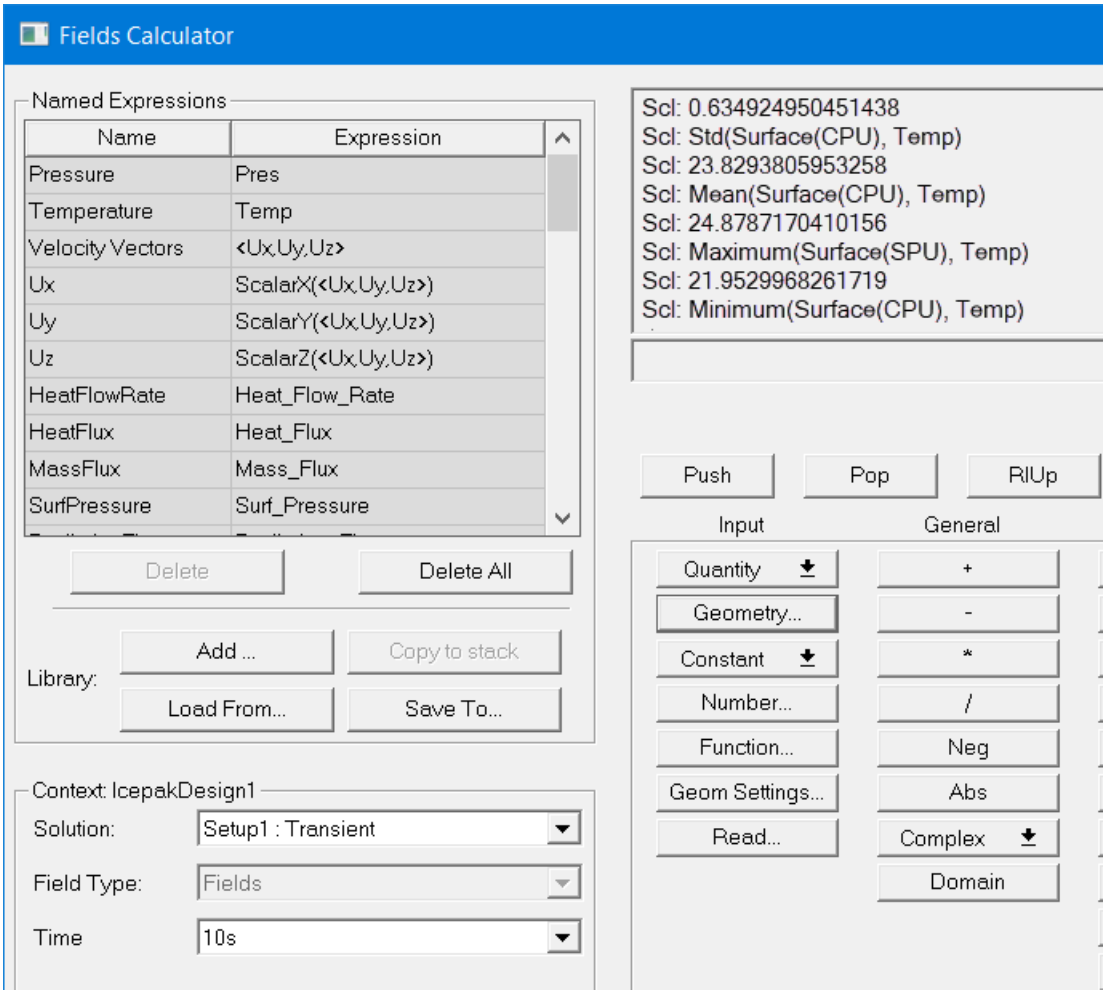
Use the Fields Calculator

The following example calculates the surface temperature (min, max, mean, and standard deviation) of a CPU at the 10 second time step. Other quantities are available for calculations using the **Fields Calculator** for a transient solution. For more information about the Fields Calculator, see Using the Fields Calculator.

1. From the **Icepak** menu, select **Fields** > **Calculator**.
2. In the **Fields Calculator** dialog box under **Context**, ensure the **Solution** is set to the transient solution.
3. From the **Time** drop-down list, select **10s**.
4. Under **Input**, click **Quantity** and select **Temp**.
5. Under **Input**, click **Geometry** and select **Surface** and **CPU** in the **Geometry** dialog box.
6. Under Scalar, click **Min** and select **Value**.
7. Under **Output**, click **Eval** to calculate the minimum surface temperature for the CPU.
8. Repeat steps 4 through 7 for **Max**, **Mean**, and **Std** under **Scalar**. The results of the calculations are displayed in the stack area.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Mean and **Std** do not require the selection of **Value**.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19 - 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
- IcepakFEA
- Q3D Extractor
- 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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 icon, search for and then select **Modify Integration with Ansys 20*yy* R*n***.

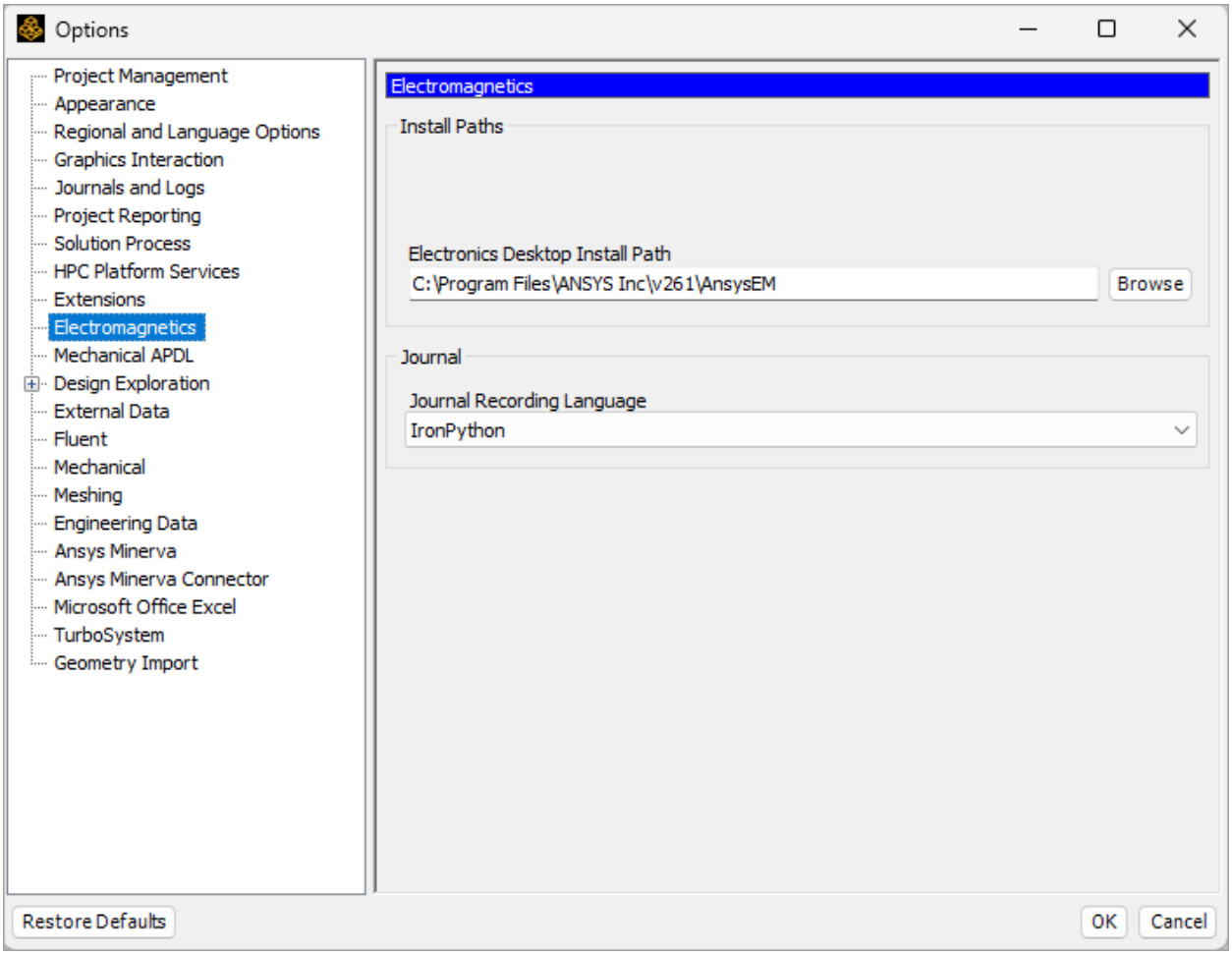
   A *Modify Integration with Ansys 20yy Rn* command window appears.

   - 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 performed, 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 *Tools* > *Options* window in Workbench. The path to the integrated Electronics Desktop application is listed in the *Electromagnetics* group of settings.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

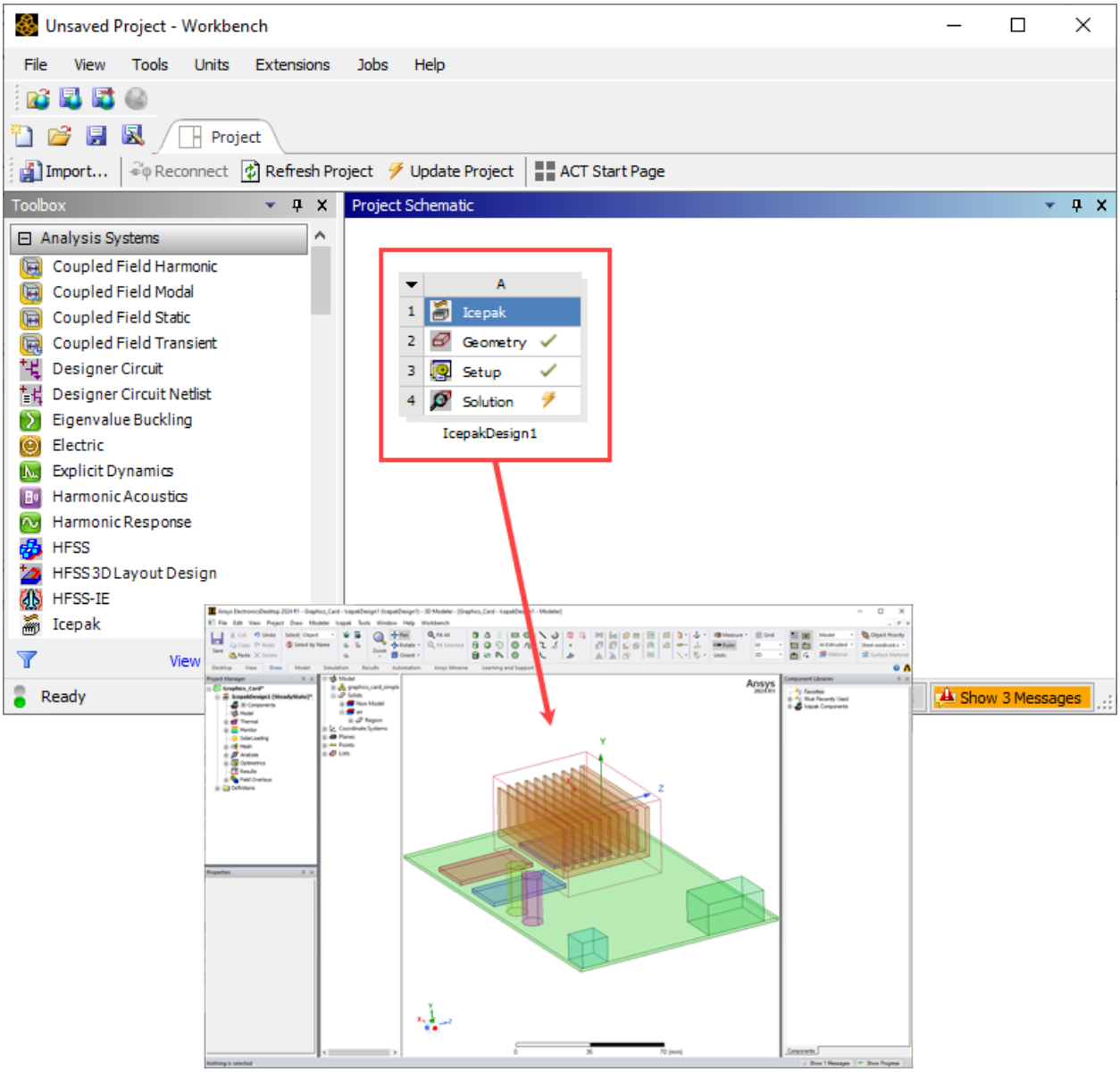


In this group of options, you can also specify Workbench's default **Journal Recording Language** for use when interoperating with the Ansys Electromagnetics products. **Iron Python** is the default language, and the alternative choice is **Visual Basic** (VB). Changes applied here affect any new Workbench 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

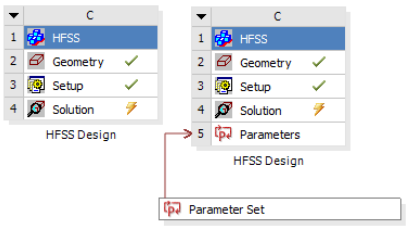


Objects placed on a Workbench Project Schematic (such as instances of Ansys Electromagnetics projects) are referred to as *systems*. The number of cells in the system depend on the design type, as follows:

- **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 **IcepakFEA:** Include an additional **Geometry** cell along with the **<*System_Type_Name*>, Setup,** and **Solution** cells, for a total of four cells.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

> **Note:**
>
> Look in the *Ansys Workbench Help* to find detailed information about performing the above-listed operations.

Electronics Desktop Icepak Systems in Workbench

The following details are specific to Ansys Electronics Desktop (AEDT) **Icepak** systems within a Workbench schematic. These details apply Whether the system is newly created or is an existing AEDT Icepak design that has been imported into the Workbench schematic:

- Only *Steady-State* and *Transient* solution types are currently supported for *Electronics Desktop Icepak* systems in Workbench.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additionally, you can only link the temperature results from these solutions to *Static Structural* or *Transient Structural* analysis systems in the Workbench schematic.

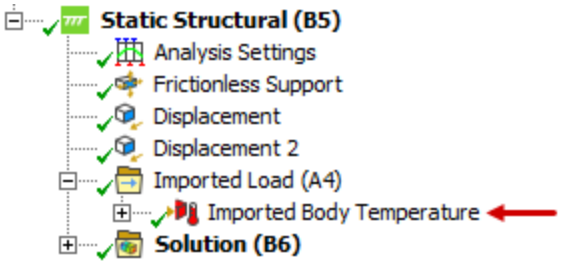
- The referenced AEDT project can only contain a single design.

  Instead of attempting to link to a multi-design AEDT project, create multiple *Electronics Desktop Icepak* systems in the WB schematic, each referencing a single-design AEDT model.

- The linked AEDT project's design ***cannot*** include any type of coupling or linking with other AEDT projects or designs, such as:
  - Mesh linking
  - Imported loads (for example, EM Losses from an HFSS source design or heat transfer coefficients from an Icepak source design)
  - Layout Components (for trace mapping based on an HFSS 3D layout source design of a printedcircuit board)
- When you create a new *Electronics Desktop Icepak* 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 Icepak design already added to it.

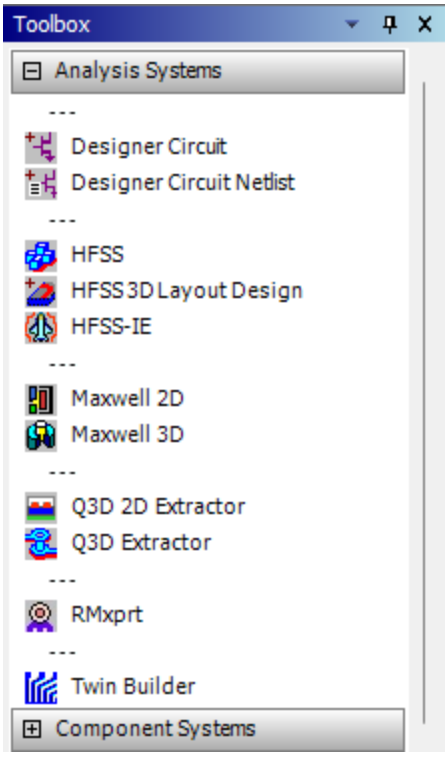
- When linking temperature data between Electronics Desktop Icepak and Electronics Desktop IcepakFEA systems, the Icepak simulation must be run within Workbench.
- When you import an existing AEDT Thermal project, a copy of the design is created in a subfolder of the Workbench project location, preserving the original source design.
- After setting up and solving the Electronics Desktop Icepak 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

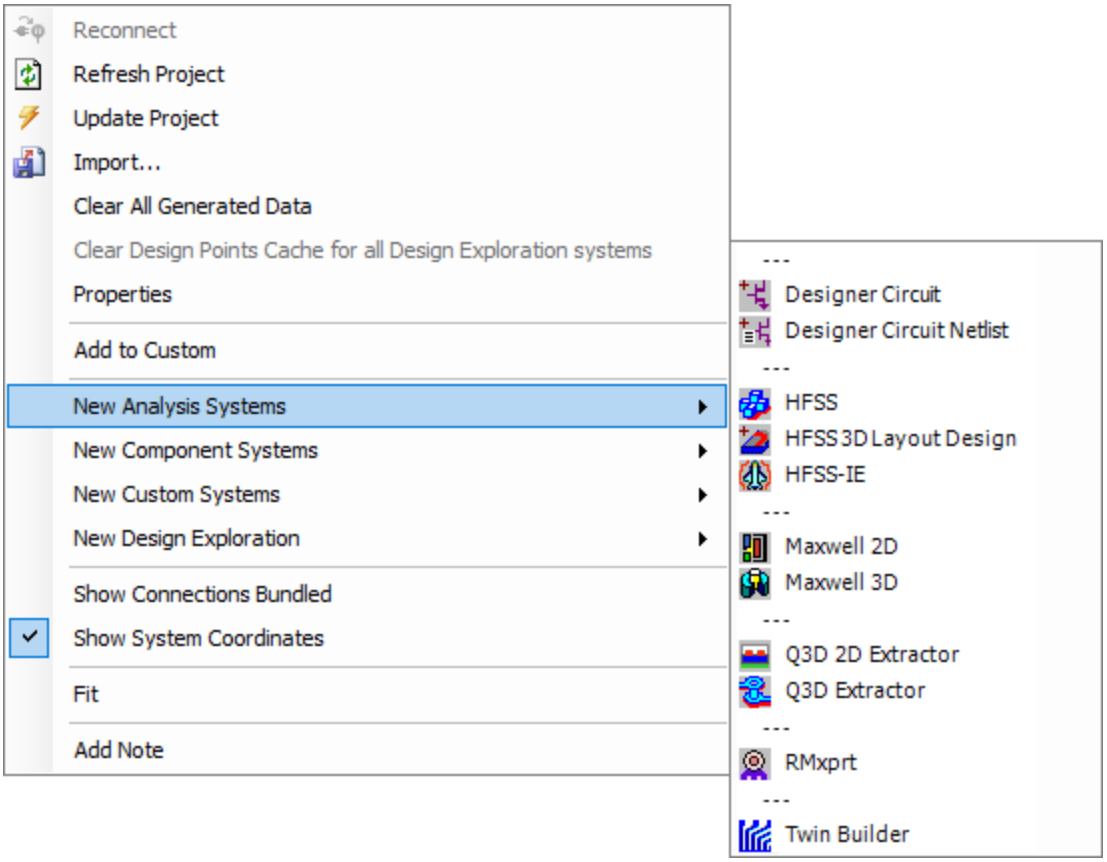
A new Ansys Electromagnetics Analysis System can be added to a Workbench Project Schematic by dragging and dropping it from the Toolbox:

Ansys Electromagnetics Suite 2026 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]**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

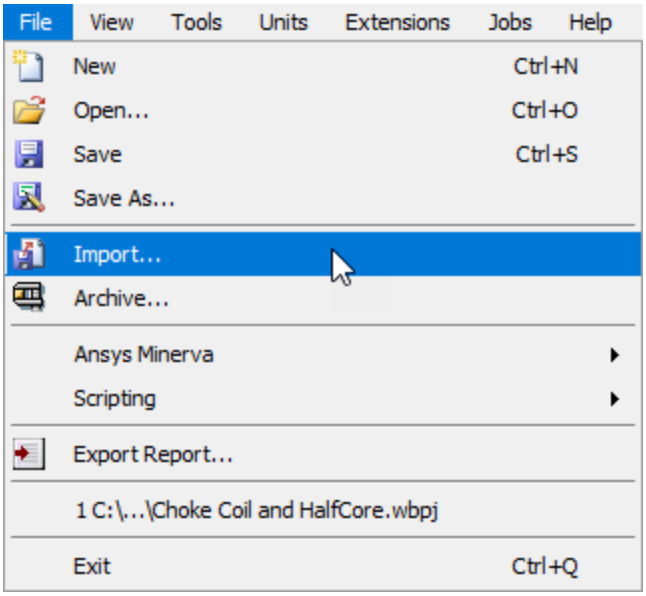


Importing Existing Ansys EM Projects into Workbench

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

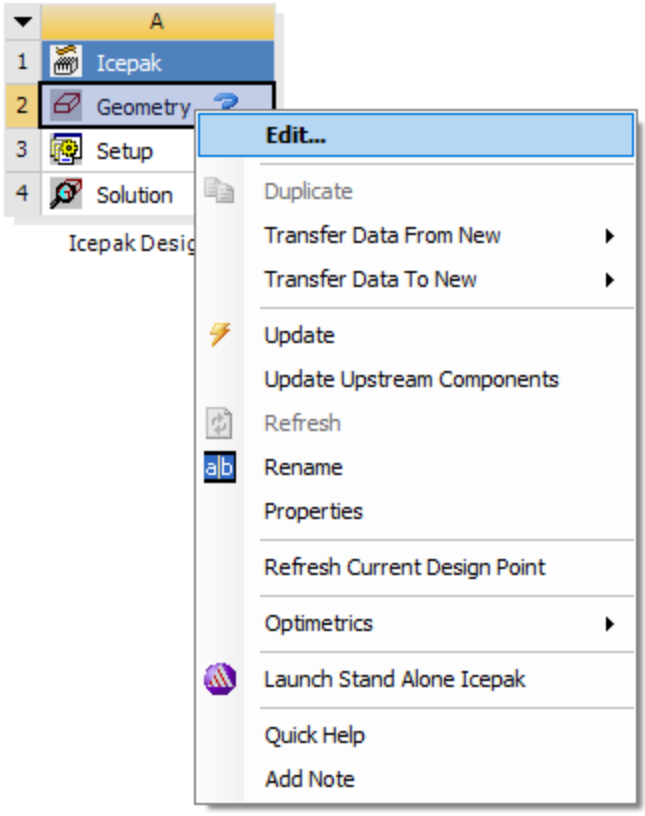


Editing Ansys EM Models in Workbench

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.

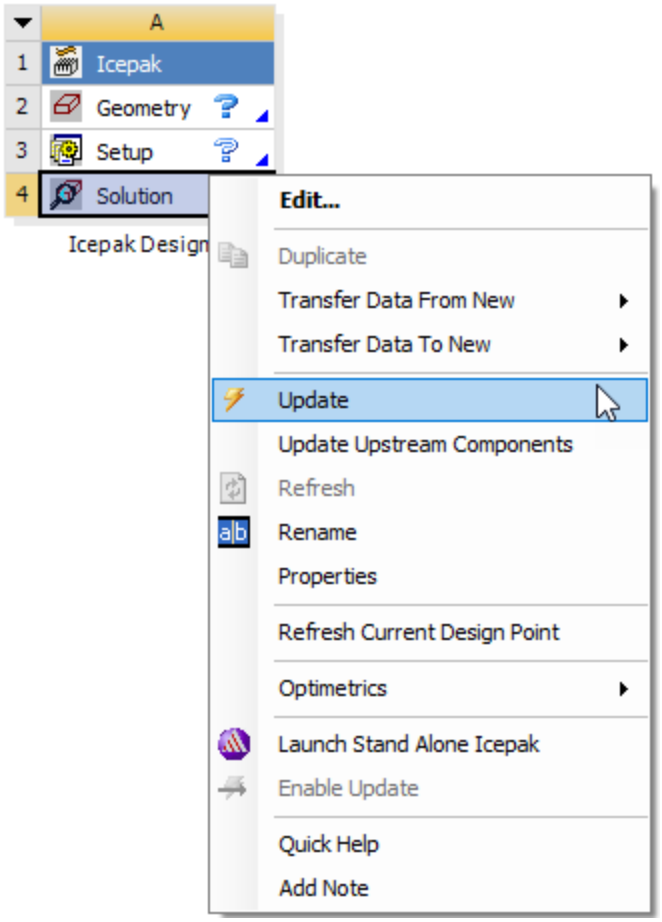
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



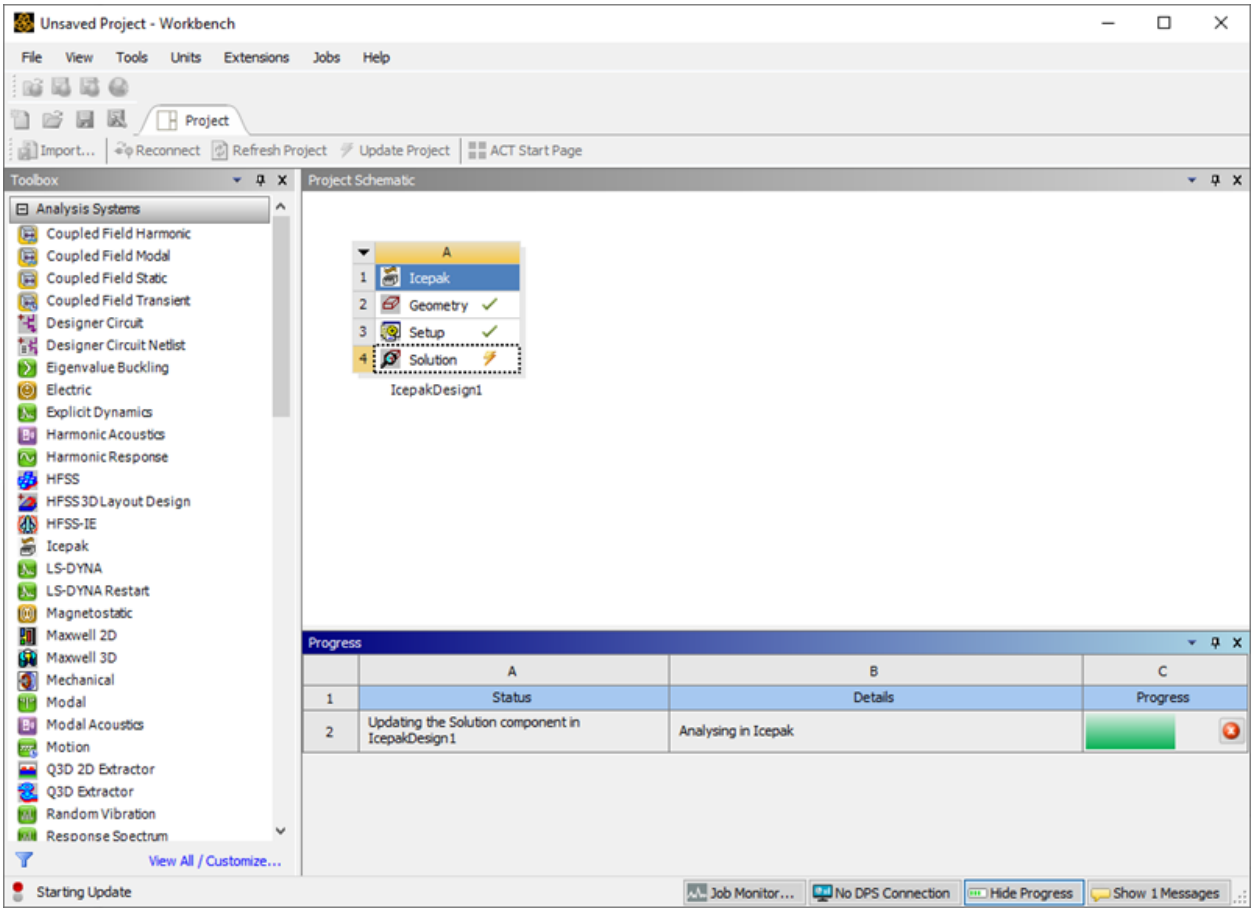
Analyzing Ansys EM Models in Workbench

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



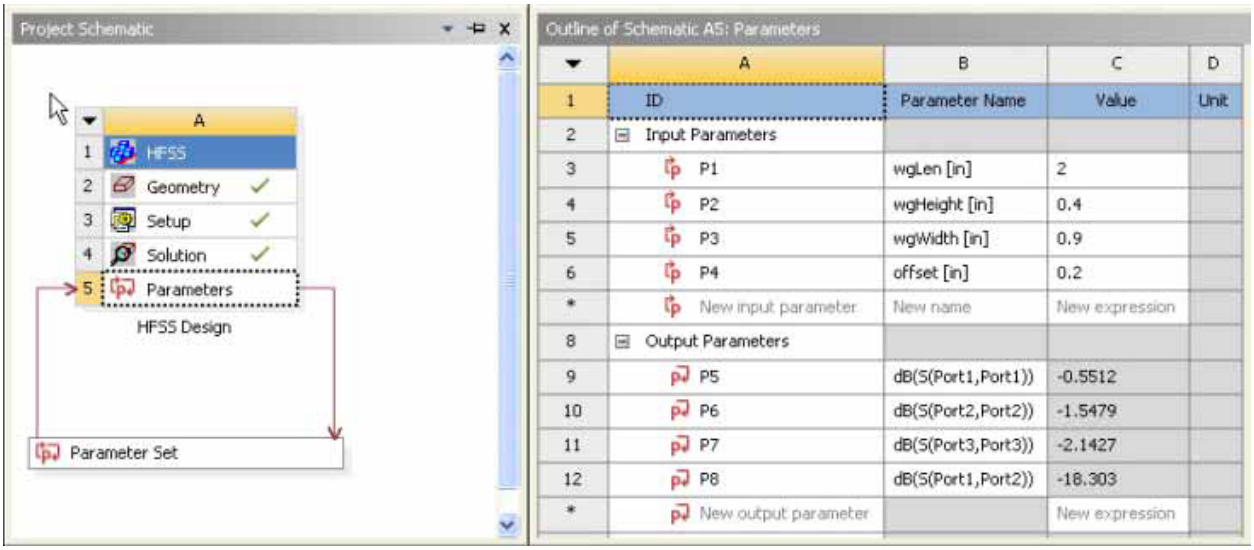
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

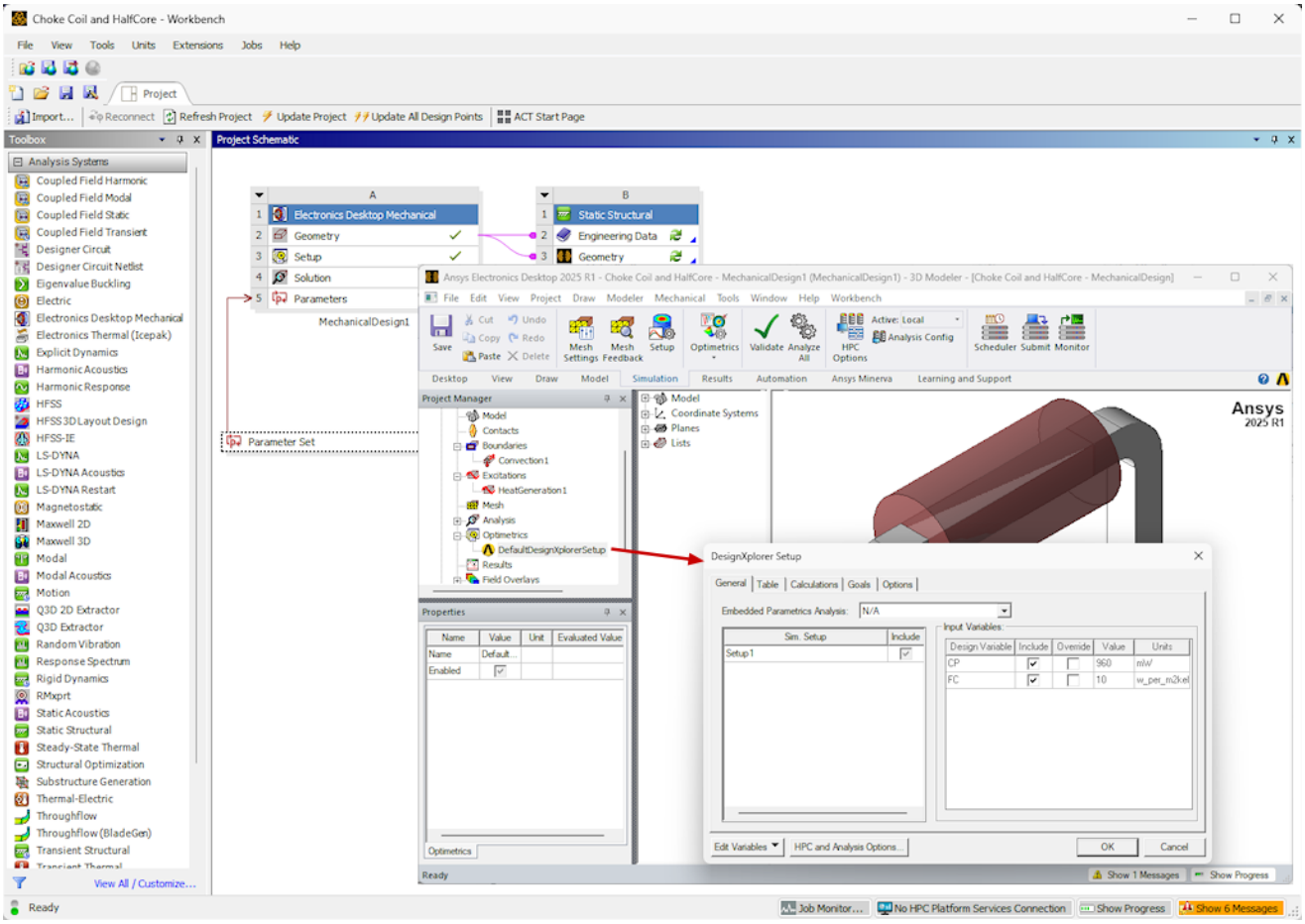


Note:

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 an IcepakFEA, 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:

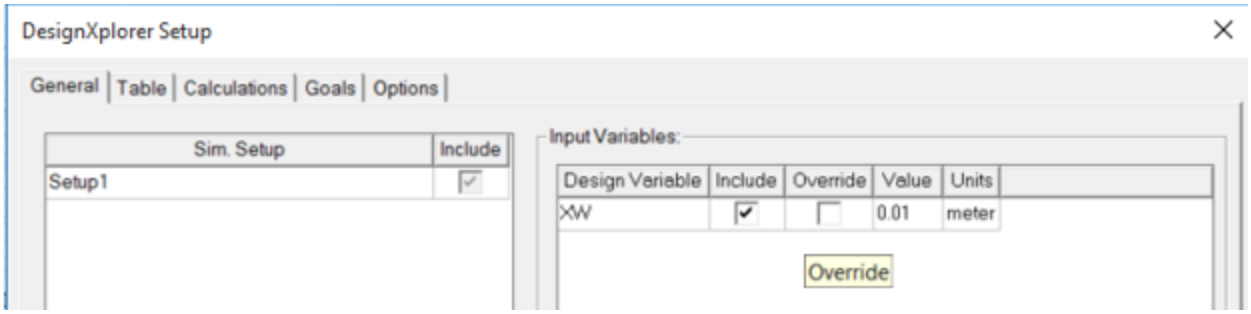
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



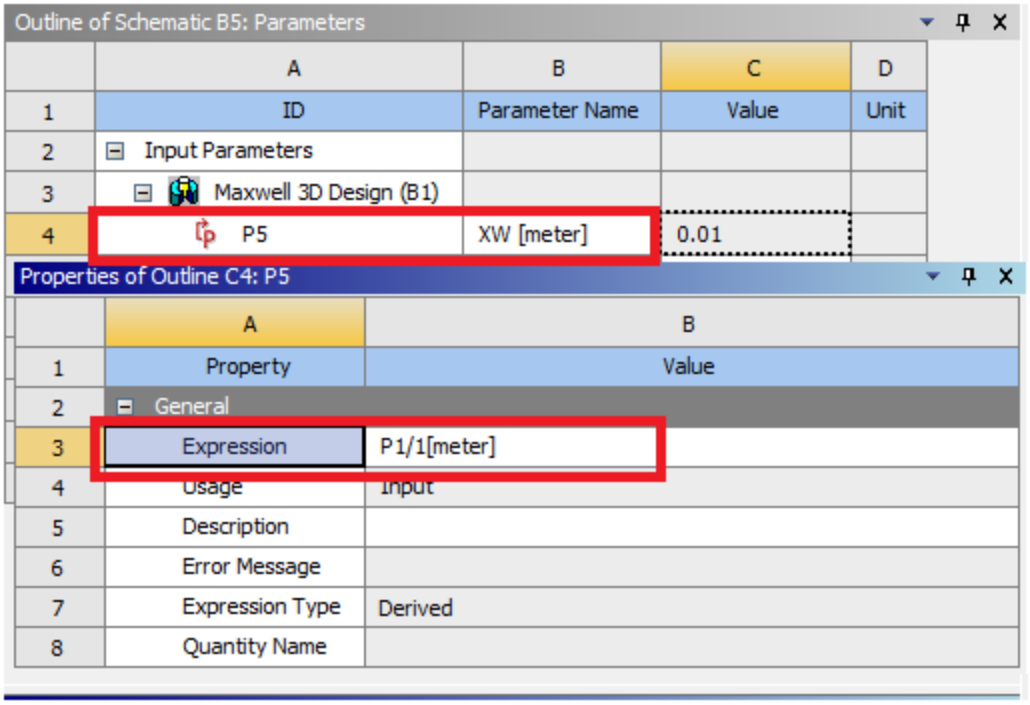
The workflow for using Electronics Desktop systems with the Workbench to take advantage of Distributed Analysis is as follows:

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

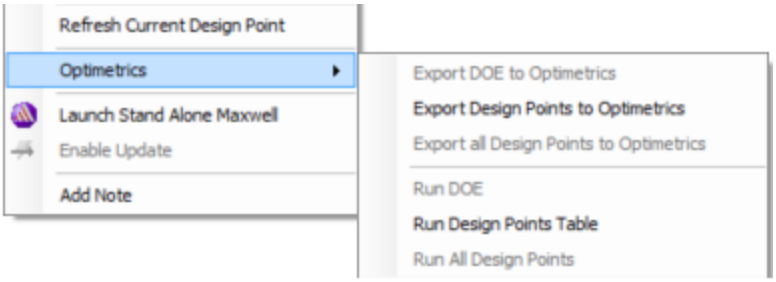


2. In the Workbench Design, change the P5 expression to a new value (in this example figure, the P5 parameter is assigned an expression of P1/1[meter] to tie it to the geometry system's variable.



In Workbench, the **Export Design Points to Optimetrics** and **Run Design Points Table** commands are enabled.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

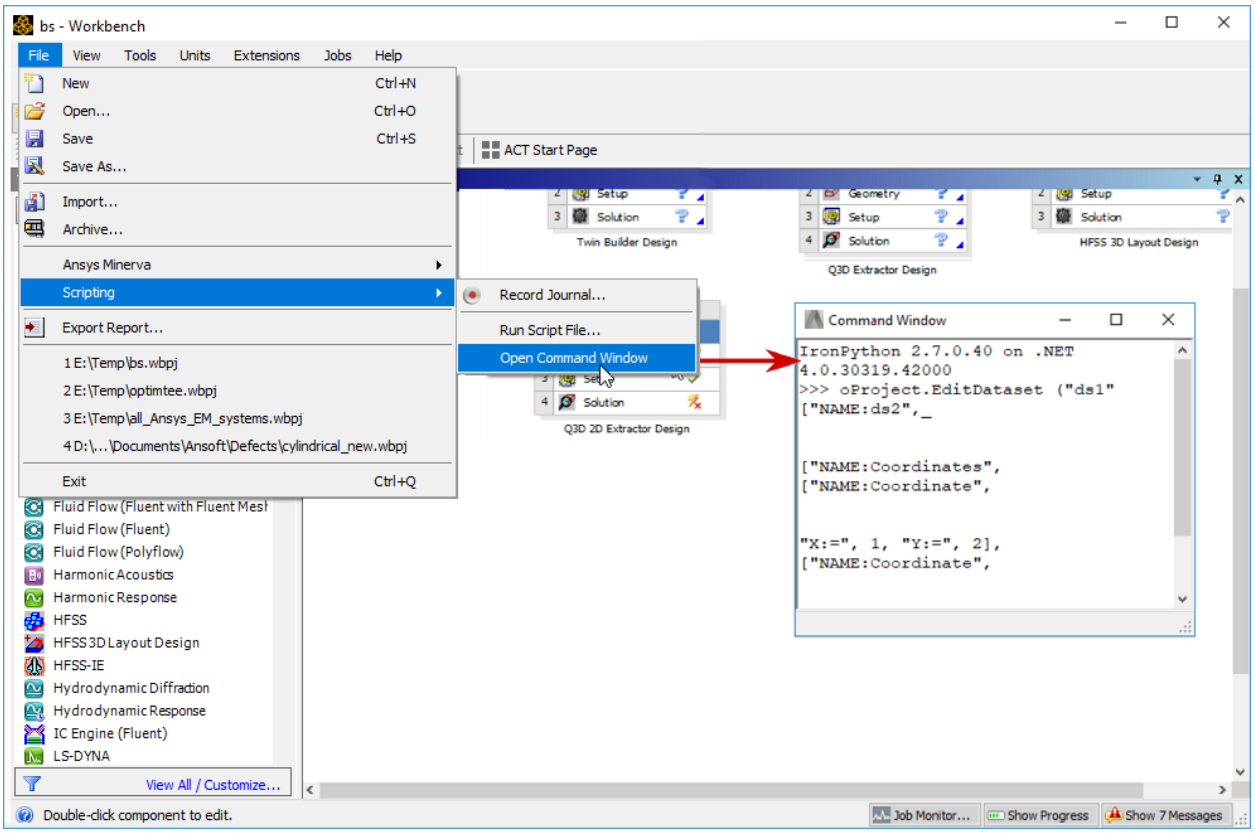


3. In Electronics Desktop, you can click on the default DesignXplorer Setup, and choose **Generate Variation Data**.

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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

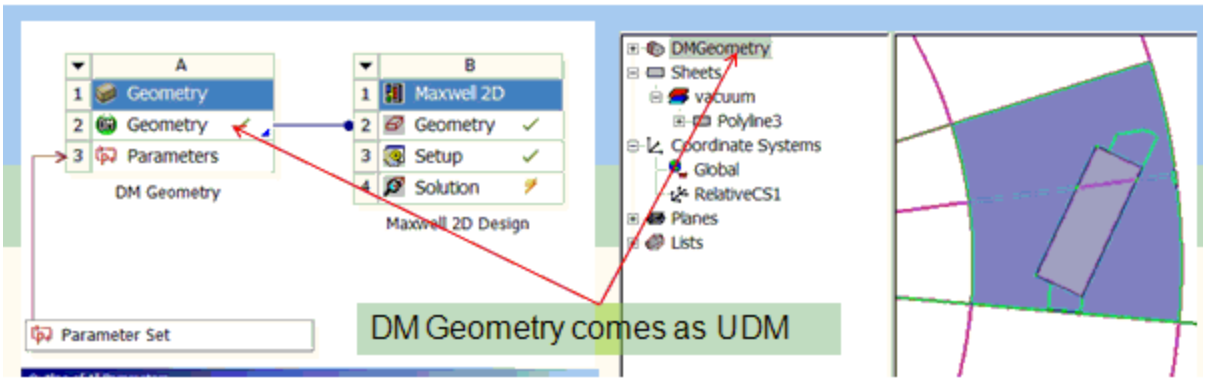
When a Workbench journal is run from the command line, Electronics Desktop allows the Ansys EDT portion to also run in batch mode (that is, when using RunWB2 -b).

Example: From a Windows Command prompt, enter the following command to replay the journal named "myReplayJournal.wbjn":

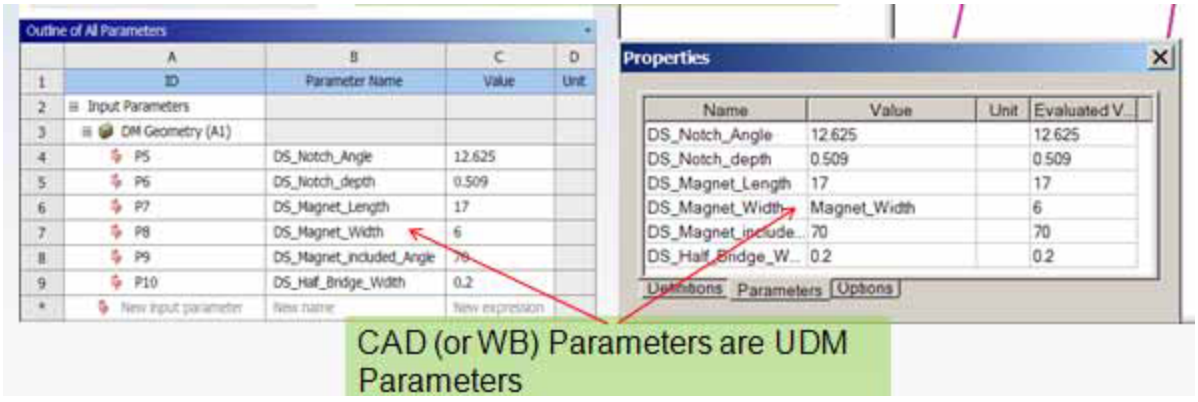
```
RunWB2 -b -r myReplayJournal.wbjn
```

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 IcepakFEA, as the Ansys Framework. The feature is available only through Workbench, and is not available via standalone Ansys Electromagnetics products.



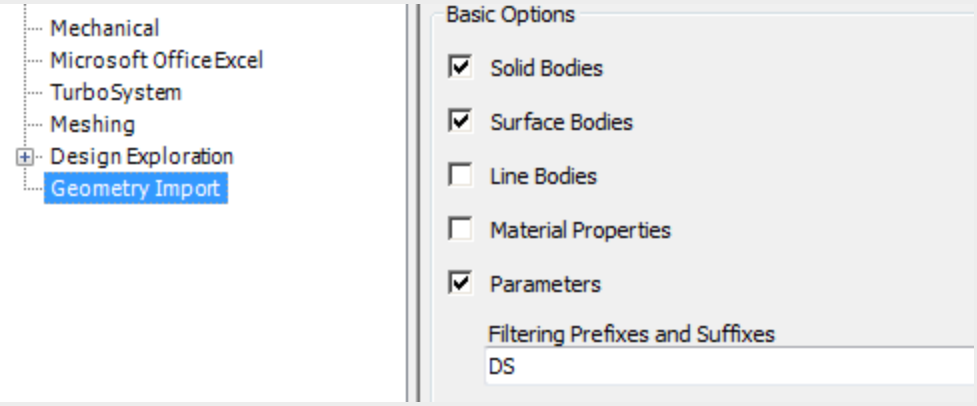
The UDM format makes it possible to exchange parameters. Ansys online help contains further description of the UDM feature and function.



Ansys Electromagnetics Suite 2026 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 2026 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

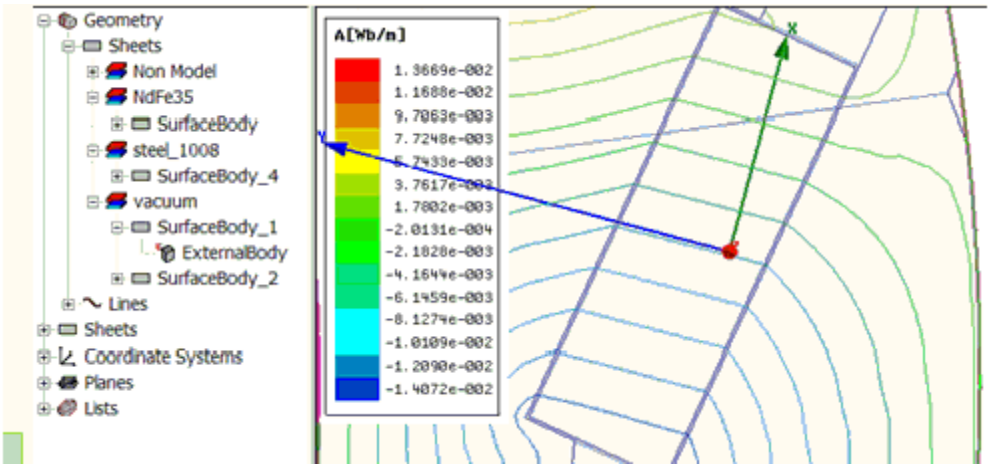
Healing with CAD Integration

Important Geometry Options for CAD Integration

CAD Integration Functionality

CAD Integration includes the following functionality:

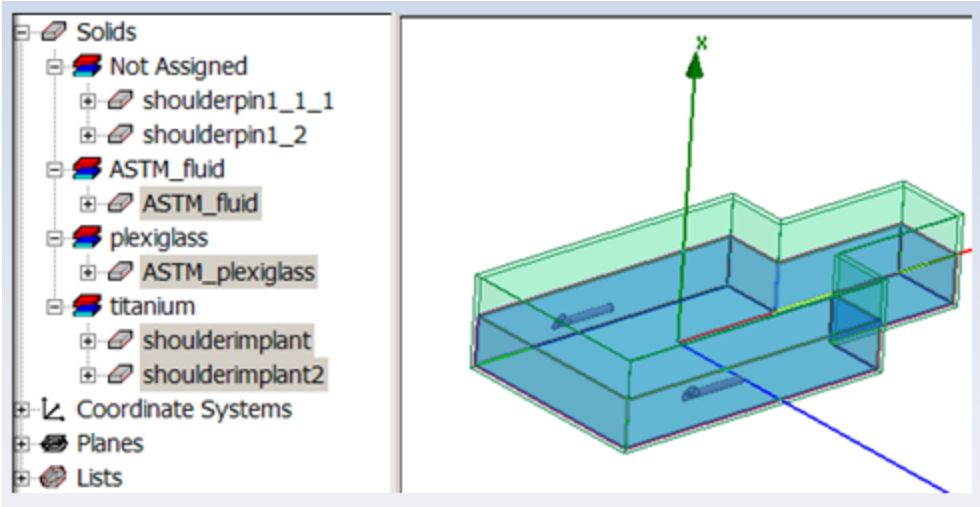
- DX analysis
- WB Update Project
- WB Update All Design Points
    - Parametric analysis with DSO
    - Animation
    - Geometry
    - Field plots



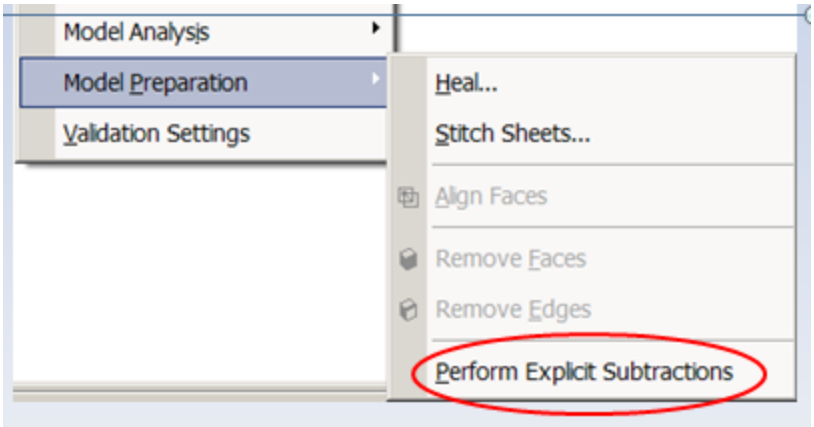
Ansys Mechanical does not do implicit subtraction. Ansys Electromagnetics products can do explicit subtractions before sending geometry to Ansys using **Modeler > Model Preparation > Perform Explicit Subtraction**.

For example, consider the following model.

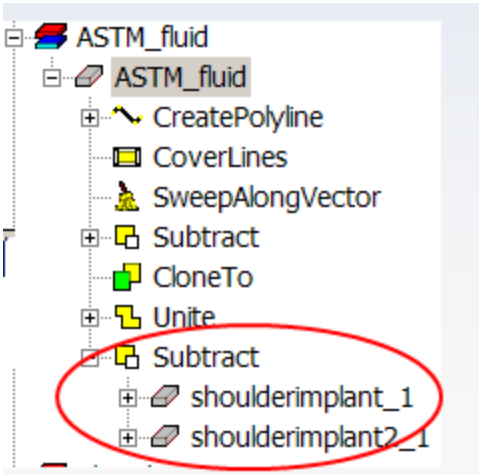
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Perform Explicit Subtractions can be performed.

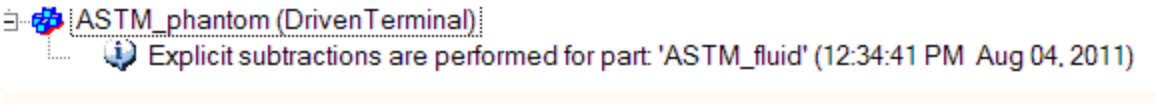


Results appear in the History tree, as shown:



Ansys Electromagnetics Suite 2026 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.



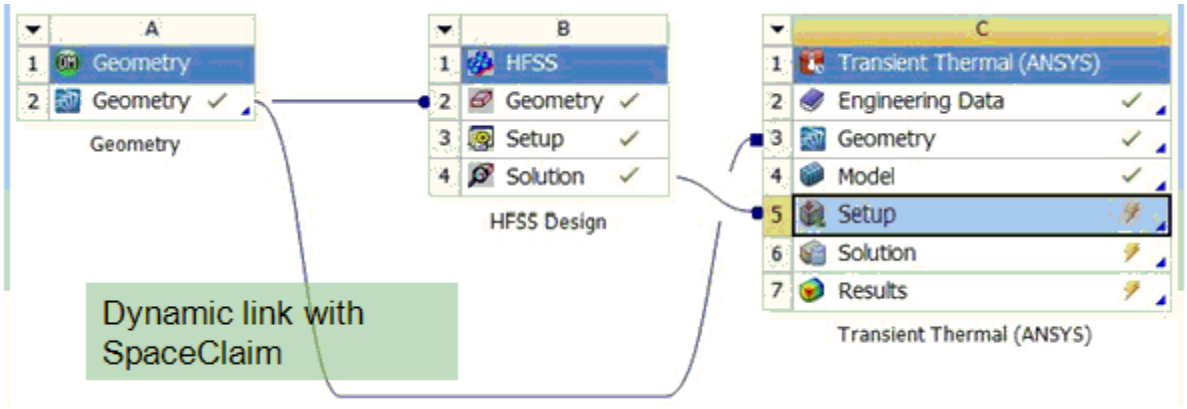
Me

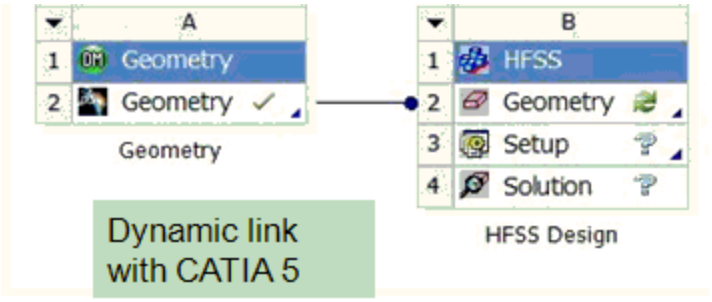
CAD Integration and Geometry Sharing

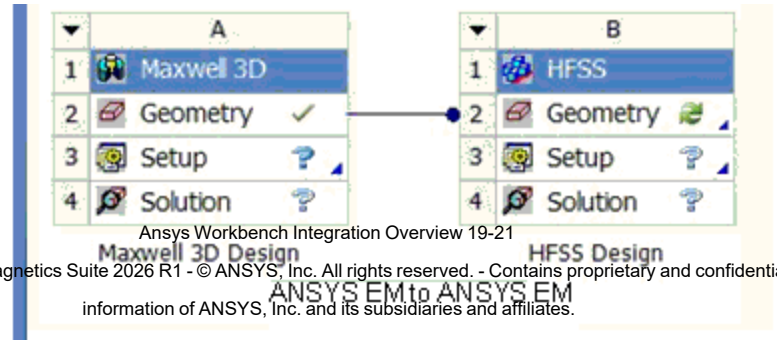
A CAD model comes into Ansys Electromagnetics as a User Defined Model (UDM).

The inputs to Ansys Electromagnetics from CAD are

- Geometry/Topology with persistent IDs
- CAD parameters
- Material assignment
- Attributes, such as name and color

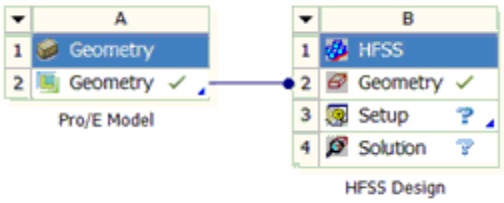




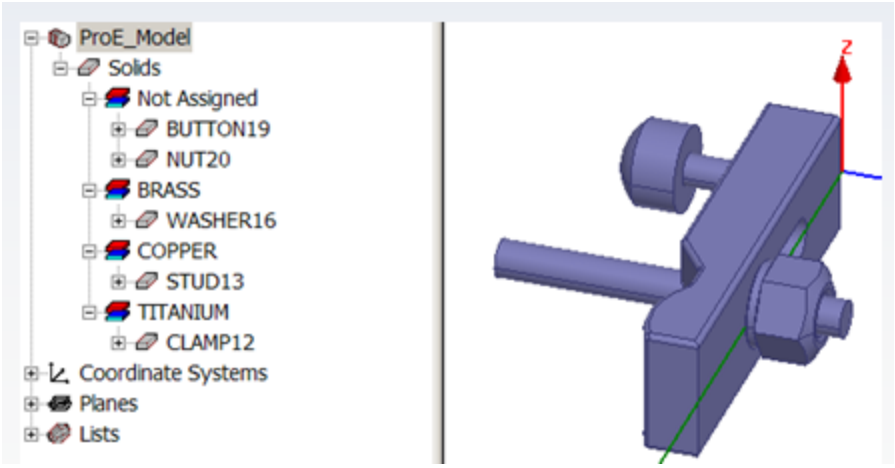


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in Workbench, a Pro/E Model can be linked to Ansys:



The geometry can then be viewed as a UDM.



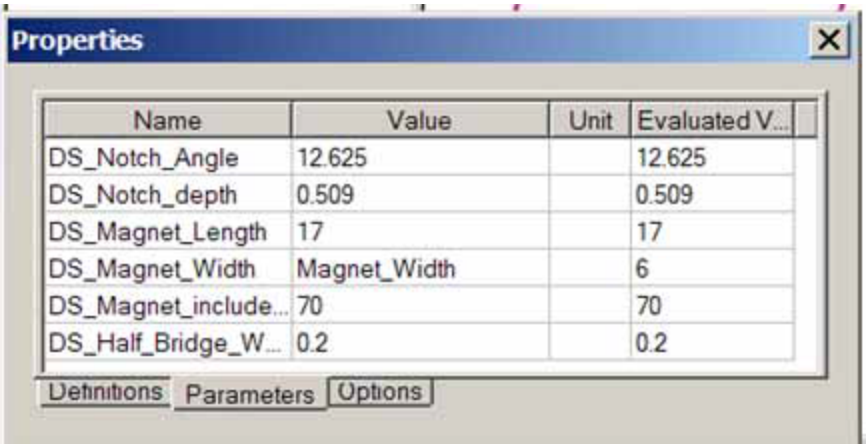
The CAD or WB model parameters appear in the Workbench:

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	

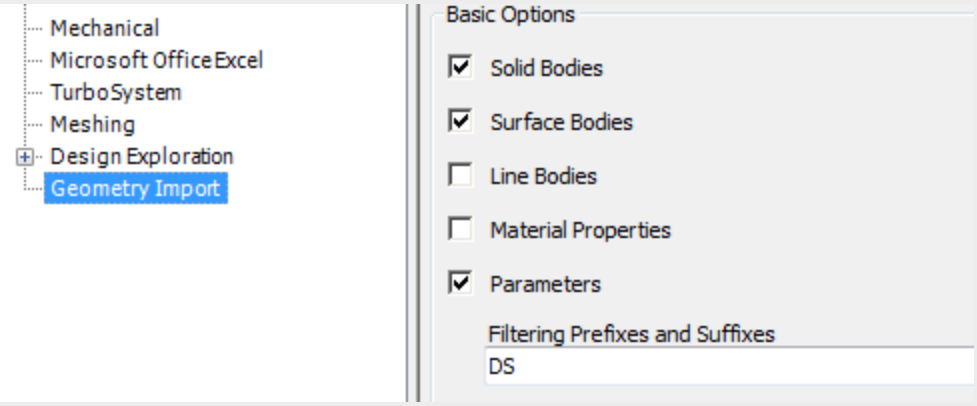
Through the Ansys Electromagnetics CAD integration, the linked UDM includes the same parameters.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



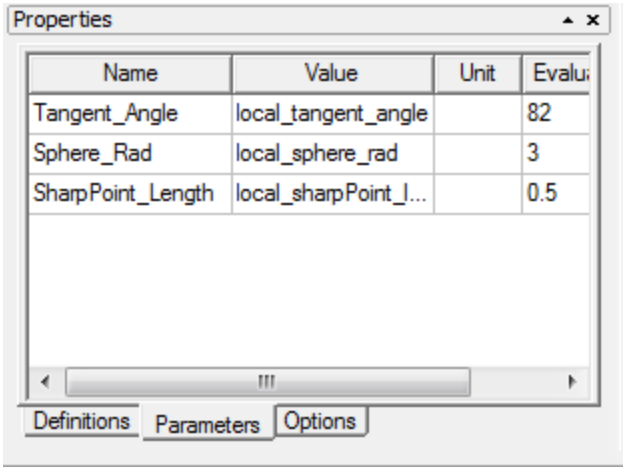
Note:

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 2026 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.

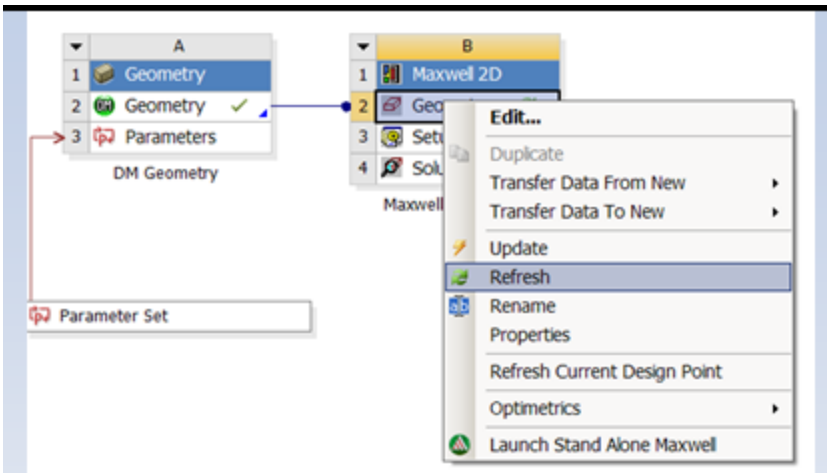
Table of Schematic D2: Design of Experiments (Custom)

	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.

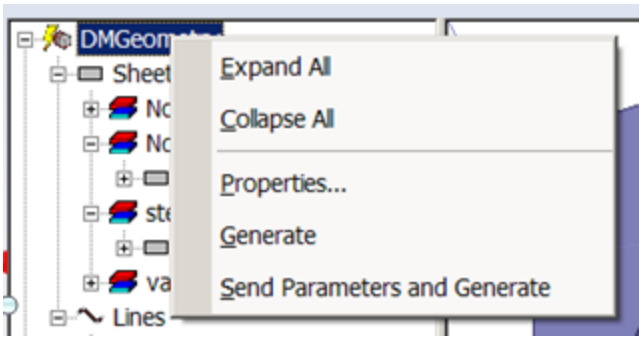


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

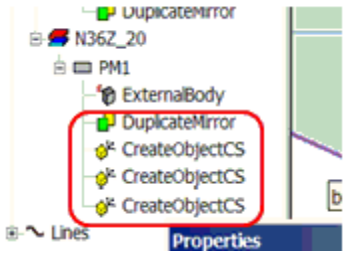
If you edit UDM (CAD) parameters in the Ansys Electromagnetics modeler window you can run the **Send Parameters and Generate** command.



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**.

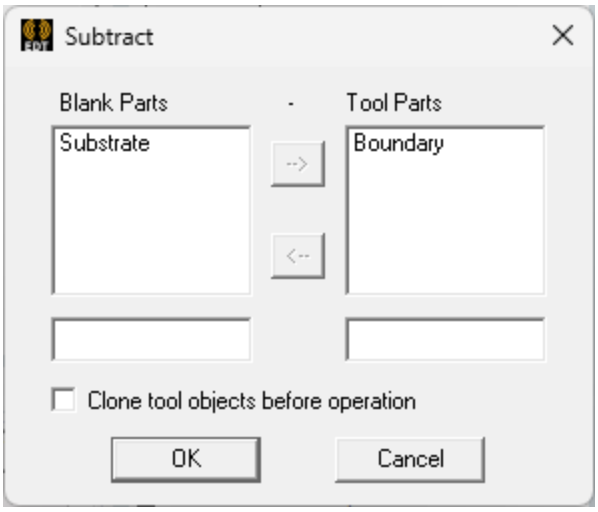


The following operations are not allowed:

- Non-history tree operations like Heal or defeature.
- Operations that use UDM parts as tools, such as Sweep

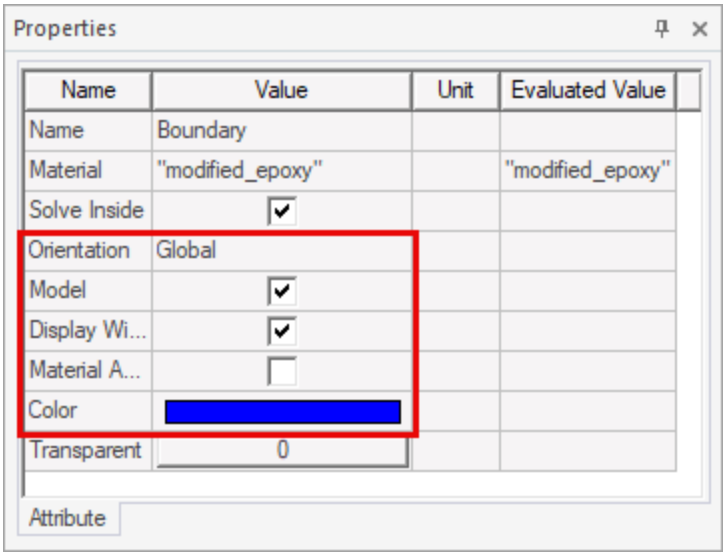
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 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



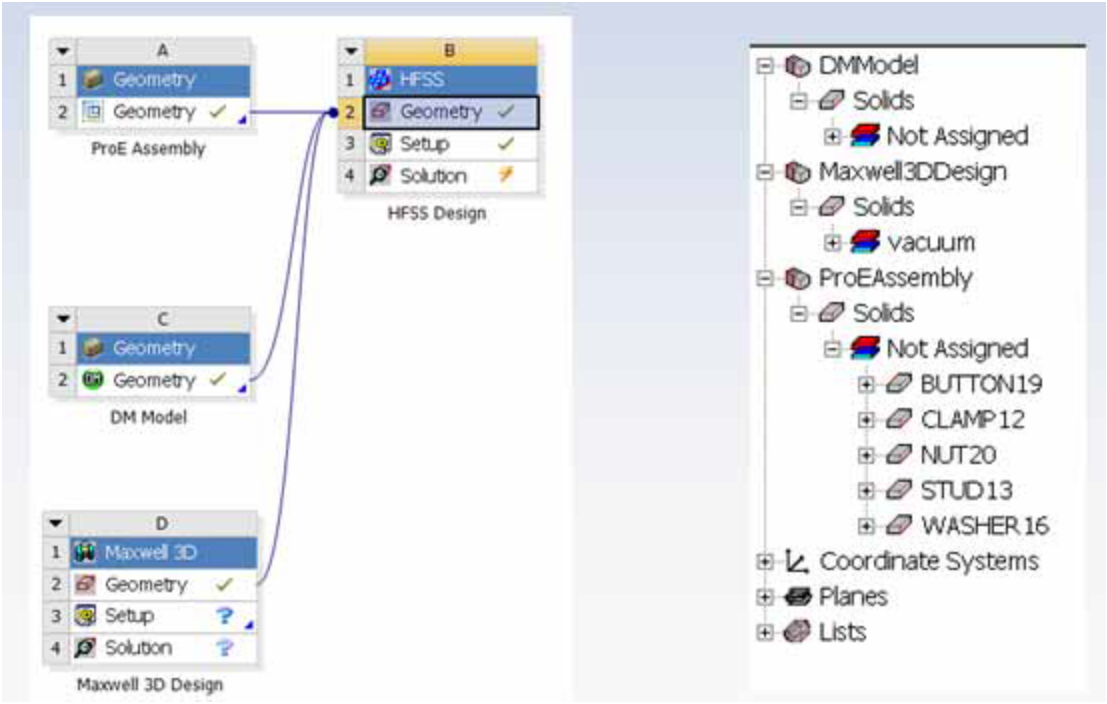
It is not possible to delete individual parts of UDM.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

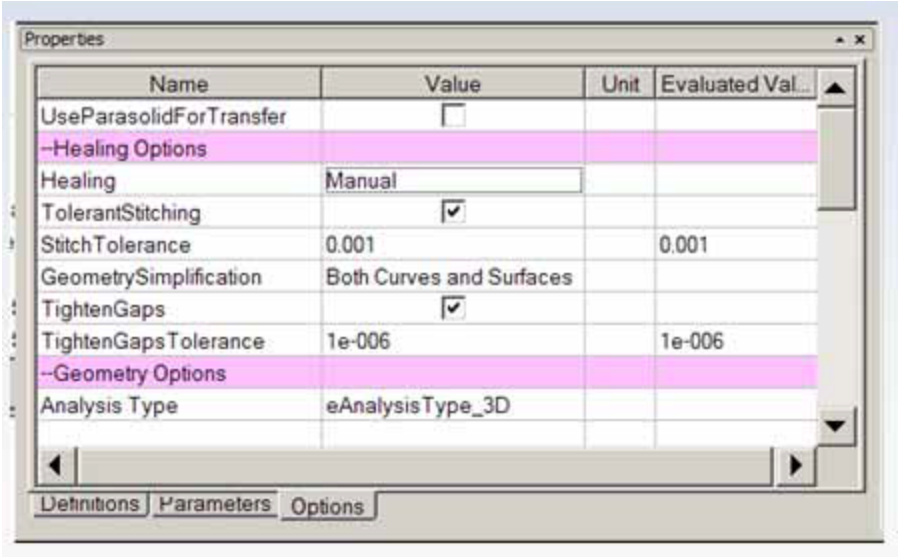
For example, the following figure shows a DM Model, a Maxwell model, and ProE model linked to HFSS in Workbench, and displayed in the HFSS History tree as three UDMs.



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.

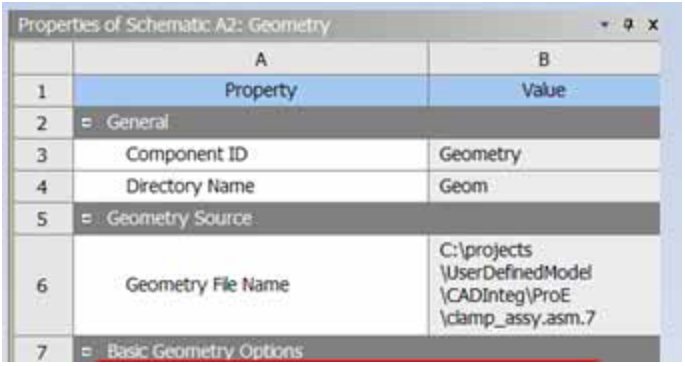
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



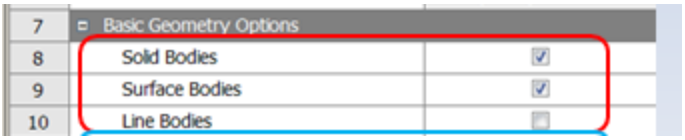
Healing options are None, Auto and Manual. By default, healing is off (None) and should be turned on only if required.

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

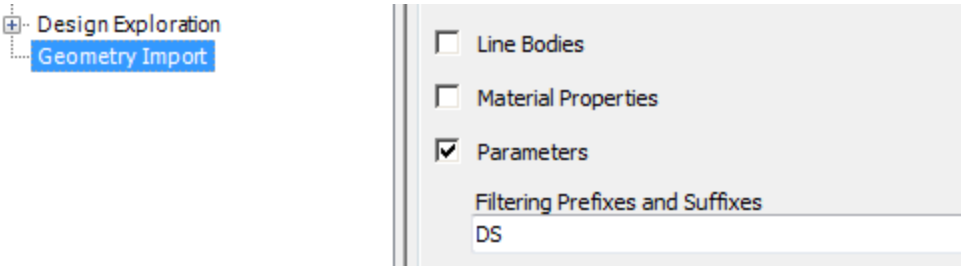


Make sure **Parameters** is checked and that the parameter key (filter) is appropriate for CAD parameters.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.



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)

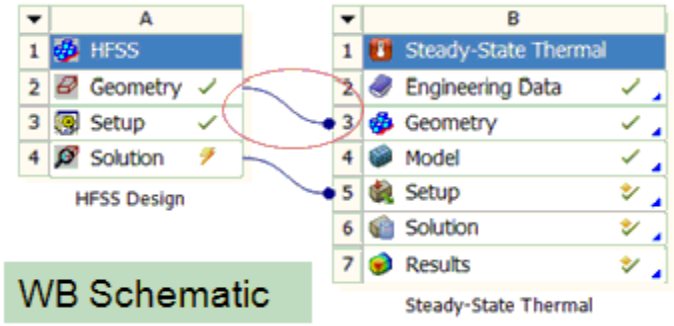


Refer to Ansys Help for details.

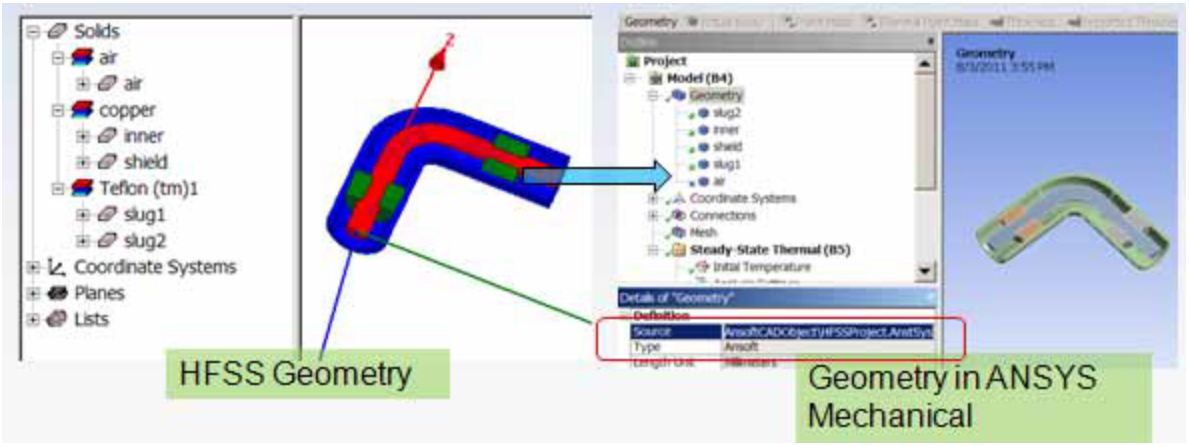
Ansys EM to Ansys Geometry Transfer

Ansys Electromagnetics CAD Integration transfers model information based on Workbench links.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

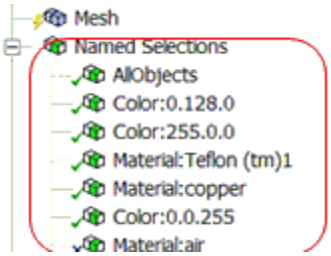


The following figure shows how the information is transferred between simulators.



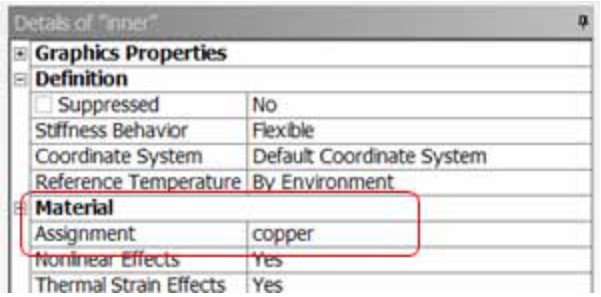
Information transferred includes

- Geometry
- Ansys Electromagnetics lists and material assignment as Named Selection



- Material assignment

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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.

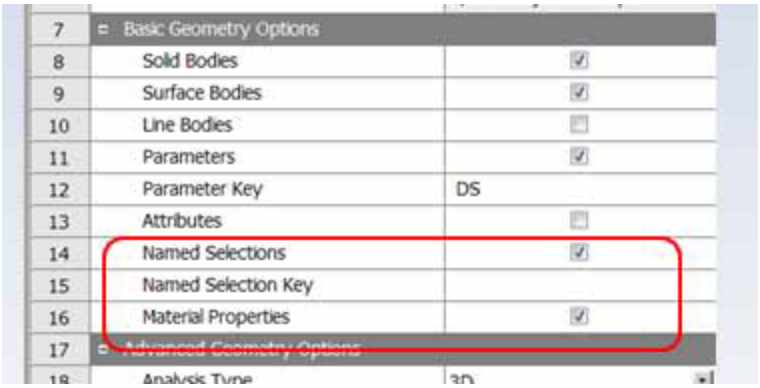
Material Assignment Transfer

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."

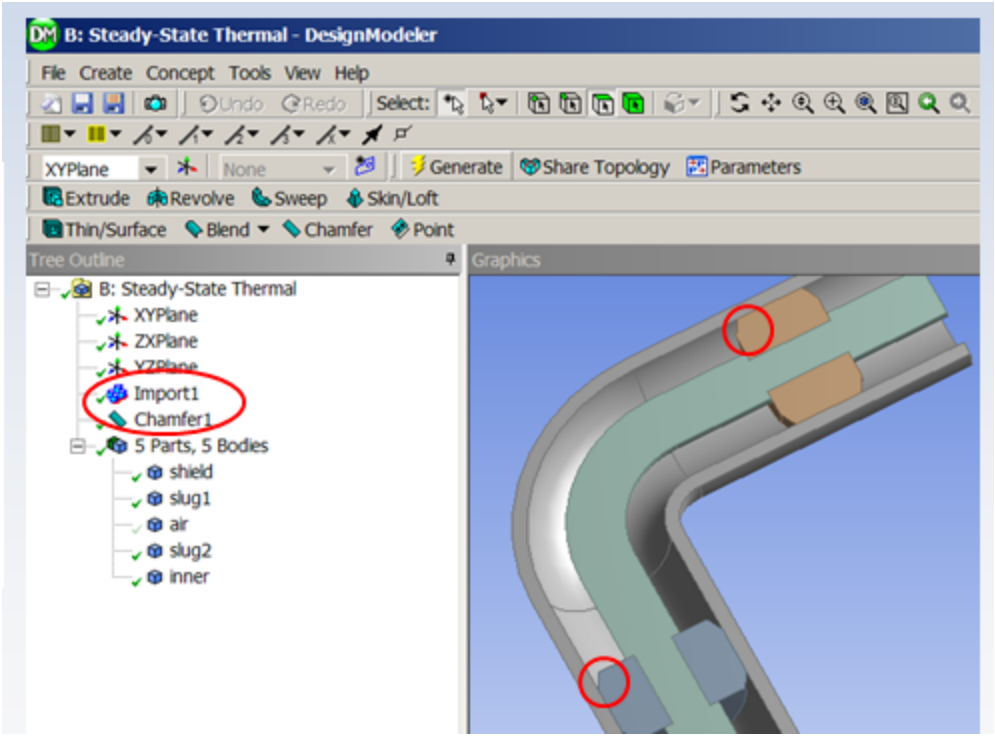


Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Transfer through Ansys DesignModeler (DM)

It is possible to edit geometry in Ansys DesignModeler (DM) before consuming it in Icepak. This is useful when geometry requires preprocessing 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.

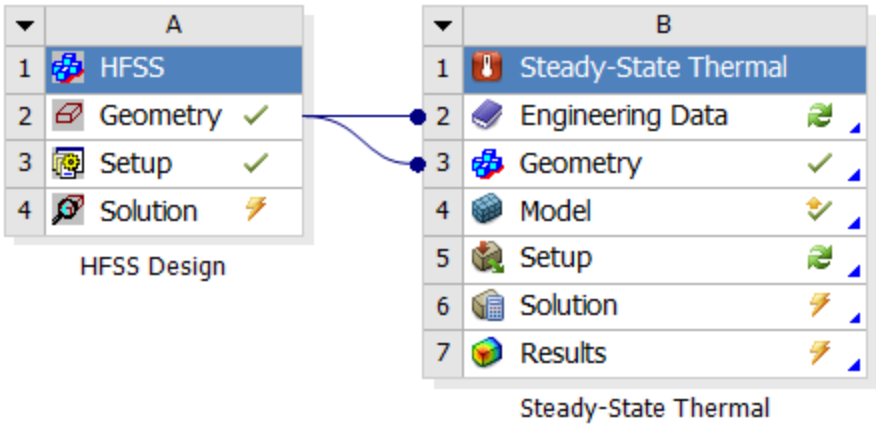


Workbench Material Data Transfer

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 2026 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20 - Icepak Technical Notes

The Icepak Technical Notes provide theoretical background for the theory and equations used by Icepak.

Information is included on the following:

- Governing Equations
- Turbulence
- Buoyancy-Driven Flows and Natural Convection
- Radiation
- Optimization

Governing Equations

Ansys Icepak solves the Navier-Stokes equations for transport of mass, momentum, and energy when it calculates laminar flow with heat transfer. Additional transport equations are solved when the flow is turbulent (see the section called Turbulence).

The Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

Equation 1

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

For an incompressible fluid, this reduces to

Equation 2

$$\nabla \cdot \vec{v} = 0$$

Momentum Equations

Transport of momentum in an inertial (non-accelerating) reference frame is described by [2]

Equation 3

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left(\overline{\overline{\tau}}\right) + \rho \vec{g} + \vec{F}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where p is the static pressure, $\overline{\overline{\tau}}$ is the stress tensor (described below), and $\rho\vec{g}$ is the gravitational body force. $\vec{F}$ contains other source terms that may arise from resistances, sources, and so on.

The stress tensor $\overline{\overline{\tau}}$ is given by

Equation 4

$$\overline{\overline{\tau}} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^T\right) - \frac{2}{3}\nabla\cdot\vec{v}I\right]$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right-hand side is the effect of volume dilation.

Energy Conservation Equation

The energy equation for a fluid region can be written in terms of sensible enthalpy h ($h = \int_{T_{ref}}^{T} c_p dT$, where T_{ref} is 298.15 K) as

Equation 5

$$\frac{\partial}{\partial t}(\rho h) + \nabla\cdot(\rho h\vec{v}) = \nabla\cdot[(k + k_t)\nabla T] + S_h$$

where k is the molecular conductivity, k_t is the conductivity due to turbulent transport ($k_t = c_p\mu_t/Pr_t$), and the source term S_h includes any volumetric heat sources you have defined.

In conducting solid regions, Icepak solves a simple conduction equation that includes the heat flux due to conduction and volumetric heat sources within the solid:

Equation 6

$$\frac{\partial}{\partial t}(\rho h) = \nabla\cdot(k\nabla T) + S_h$$

where ρ is density, k is conductivity, T is temperature, and S_h is the volumetric heat source.

> **Note:** "Equation 6" above is solved simultaneously with the energy transport equation, "Equation 5" above, in the flow regions to yield a fully coupled conduction/convection heat transfer prediction.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence

Eight turbulence models are available in Ansys Icepak: the zero-equation (mixing-length) model, the two-equation (standard $k-\varepsilon$) model, the RNG $k-\varepsilon$ model, the realizable $k-\varepsilon$ model, the enhanced two-equation (standard $k-\varepsilon$ with enhanced wall treatment) model, the enhanced RNG $k-\varepsilon$ model, the enhanced realizable $k-\varepsilon$ model, the Spalart-Allmaras model and the $k-\omega$ SST model.

Zero-Equation Turbulence Model

The mixing-length zero-equation turbulence model (also known as the algebraic model) uses the following relation to calculate turbulent viscosity, μ_t:

Equation 7

$$\mu_t = \rho \mathrm{l}^2 S$$

The mixing length, l, is defined as

Equation 8

$$\mathrm{l} = \min\left(\kappa d, 0.09 d_{max}\right)$$

where d is the distance from the wall and the von Kármán constant κ = 0.419.

S is the modulus of the mean rate-of-strain tensor, defined as

Equation 9

$$S \equiv \sqrt{2 S_{ij} S_{ij}}$$

with the mean strain rate S_{ij} given by

Equation 10

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)$$

Advanced Turbulence Models

In turbulence models that employ the Boussinesq approach, the central issue is how the eddy viscosity is computed. The model proposed by Spalart and Allmaras [26] solves a transport equation for a quantity that is a modified form of the turbulent kinematic viscosity.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The standard $k-\varepsilon$, RNG and realizable $k-\varepsilon$ models have similar forms, with transport equations for k and ε. The major differences in the models are as follows:

- the method of calculating turbulent viscosity
- the turbulent Prandtl numbers governing the turbulent diffusion of k and ε
- the generation and destruction terms in the ε equation

This section describes the Reynolds-averaging method for calculating turbulent effects and provides an overview of the issues related to choosing an advanced turbulence model in Icepak. The transport equations, methods of calculating turbulent viscosity, and model constants are presented separately for each model. The features that are essentially common to both models follow, including turbulent production, generation due to buoyancy, and modeling heat transfer.

Reynolds (Ensemble) Averaging

The advanced turbulence models in Ansys Icepak are based on Reynolds averages of the governing equations. In Reynolds averaging, the solution variables in the instantaneous (exact) Navier-Stokes equations are decomposed into the mean (ensemble-averaged or time-averaged) and fluctuating components. For the velocity components:

Equation 11

$$u_i = \overline{u}_i + u_i' i$$

where $\overline{u}_i$ and u_i' are the mean and instantaneous velocity components (i = 1, 2, 3).

Likewise, for pressure and other scalar quantities:

Equation 12

$$\varphi = \bar{\varphi} + \varphi'$$

where φ denotes a scalar such as pressure or energy. Substituting expressions of this form for the flow variables into the instantaneous continuity and momentum equations and taking a time (or ensemble) average (and dropping the overbar on the mean velocity, $\overline{u}$) yields the ensemble-averaged momentum equations. They can be written in Cartesian tensor form as:

Equation 13

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 14

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) =$$

Equation 15

$$-\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right)$$

> **Note:** "Equation 13" on the previous page and "Equation 15" above are called "Reynolds-averaged" Navier-Stokes (RANS) equations. They have the same general form as the instantaneous Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values. Additional terms now appear that represent the effects of turbulence.
>
> These "Reynolds stresses", $-\rho\overline{u_i' u_j'}$, must be modeled in order to close "Equation 15" above,
>
> For variable-density flows, "Equation 13" on the previous page and "Equation 15" above can be interpreted as Favre-averaged Navier-Stokes equations [8], with the velocities representing mass-averaged values. As such, "Equation 13" on the previous page and "Equation 15" above can be applied to density-varying flows.

Boussinesq Approach

The Reynolds-averaged approach to turbulence modeling requires that the Reynolds stresses in "Equation 15" above be appropriately modeled. A common method employs the Boussinesq hypothesis [8] to relate the Reynolds stresses to the mean velocity gradients:

Equation 16

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij}$$

The Boussinesq hypothesis is used in the Spalart-Allmaras model and the $k-\varepsilon$ models. The advantage of this approach is the relatively low computational cost associated with the computation of the turbulent viscosity, μ_t. In the case of the Spalart-Allmaras model, only one additional transport equation (representing turbulent viscosity) is solved. In the case of the $k-\varepsilon$ models, two additional transport equations (for the turbulence kinetic energy, k, and the turbulence dissipation rate, ε) are solved, and μ_t is computed as a function of k and ε. The

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

disadvantage of the Boussinesq hypothesis as presented is that it assumes μ_t is an isotropic scalar quantity, which is not strictly true.

Choosing an Advanced Turbulence Model

This section provides an overview of the issues related to the advanced turbulence models provided in Icepak.

The Spalart-Allmaras Model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity for turbomachinery applications.

On a cautionary note, however, the Spalart-Allmaras model is still relatively new, and no claim is made regarding its suitability to all types of complex engineering flows. For instance, it cannot be relied on to predict the decay of homogeneous, isotropic turbulence. Furthermore, one-equation models are often criticized for their inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from a wall-bounded to a free shear flow.

The Standard $k-\varepsilon$ Model

The simplest “complete models” of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k-\varepsilon$ model in Icepak falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [18]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semiempirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

As the strengths and weaknesses of the standard $k-\varepsilon$ model have become known, improvements have been made to the model to improve its performance. One of these variants is available in Icepak: the RNG k-ε model [29].

The RNG $k-\varepsilon$ Model

The RNG $k-\varepsilon$ model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard $k-\varepsilon$ model, but includes the following refinements:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The RNG model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard $k-\varepsilon$ model uses user-specified, constant values.
- While the standard $k-\varepsilon$ model is a high-Reynolds-number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds number effects.

These features make the RNG $k-\varepsilon$ model more accurate and reliable for a wider class of flows than the standard $k-\varepsilon$ model.

The Realizable $k-\varepsilon$ Model

The realizable $k-\varepsilon$ model is a relatively recent development and differs from the standard $k-\varepsilon$ model in two important ways:

- The realizable $k-\varepsilon$ model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ε, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The term “realizable” means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG $k-\varepsilon$ model is realizable.

The Enhanced Two-Equation Models

The $k-\varepsilon$ models are primarily valid for turbulent core flows (that is, the flow in the regions somewhat far from walls). Consideration therefore needs to be given as to how to make these models suitable for wall-bounded flows.

Turbulent flows are significantly affected by the presence of walls. Obviously, the mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. However, the turbulence is also changed by the presence of the wall in non-trivial ways. Very close to the wall, viscous damping reduces the tangential velocity fluctuations, while kinematic blocking reduces the normal fluctuations. Toward the outer part of the near-wall region, however, the turbulence is rapidly augmented by the production of turbulence kinetic energy due to the large gradients in mean velocity.

The near-wall modeling significantly impacts the fidelity of numerical solutions, inasmuch as walls are the main source of mean vorticity and turbulence. It is in the near-wall region where the solution variables have large gradients and where the momentum and other scalar transports

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are the greatest. Therefore, accurate representation of the flow in the near-wall region determines successful predictions of wall-bounded turbulent flows.

Numerous experiments have shown that the near-wall region can be largely subdivided into three layers. In the innermost layer, called the viscous sublayer, the flow is almost laminar, and the (molecular) viscosity plays a dominant role in momentum and heat or mass transfer. In the outer layer, called the fully-turbulent layer, turbulence plays a major role. Finally, there is an interim region between the viscous sublayer and the fully turbulent layer where the effects of molecular viscosity and turbulence are equally important.

To more accurately resolve the flow near the wall, the *enhanced* two-equation models combine three $k-\varepsilon$ models (standard, RNG and realizable) with enhanced wall treatment.

Enhanced Wall Treatment

Enhanced wall treatment is a near-wall modeling method that combines a two-layer model with enhanced wall functions [3 9 14 15 27 28].

In the two-layer model, the viscosity-affected near-wall region is completely resolved all the way to the viscous sublayer. The two-layer approach is an integral part of the enhanced wall treatment and is used to specify both ε and the turbulent viscosity in the near-wall cells. In this approach, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region. The demarcation of the two regions is determined by a wall-distance-based, turbulent Reynolds number.

If the near-wall mesh is fine enough to be able to resolve the laminar sublayer (typically $y^+ \approx 1$), then the enhanced wall treatment will be identical to the traditional two-layer zonal model. However, the restriction that the near-wall mesh must be sufficiently fine everywhere might impose too large a computational requirement. Ideally, then, one would like to have a near-wall formulation that can be used with coarse meshes (usually referred to as wall-function meshes) as well as fine meshes (low-Reynolds-number meshes). In addition, excessive error should not be incurred for intermediate meshes that are too fine for the near-wall cell centroid to lie in the fully turbulent region, but also too coarse to properly resolve the sublayer.

To achieve the goal of having a near-wall modeling approach that will possess the accuracy of the standard two-layer approach for fine near-wall meshes and will not significantly reduce accuracy for wall-function meshes, Icepak combines the two-layer model with enhanced wall functions to result in the enhanced wall treatment.

Computational Effort: CPU Time and Solution Behavior

The standard $k-\varepsilon$ model clearly requires more computational effort than the Spalart-Allmaras model since an additional transport equation is solved. The realizable $k-\varepsilon$ model requires only slightly more computational effort than the standard $k-\varepsilon$ model. However, due to the extra terms and functions in the governing equations and a greater degree of nonlinearity,

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computations with the RNG $k-\varepsilon$ model tend to take 10-15% more CPU time than with the standard $k-\varepsilon$ model.

Aside from the time per iteration, the choice of turbulence model can affect the ability of Ansys Icepak to obtain a converged solution. For example, the standard $k-\varepsilon$ model is known to be slightly over-diffusive in certain situations, while the RNG $k-\varepsilon$ model is designed such that the turbulent viscosity is reduced in response to high rates of strain. Because diffusion has a stabilizing effect on the numerics, the RNG model is more likely to be susceptible to instability in steady-state solutions. However, this should not necessarily be seen as a disadvantage of the RNG model, since these characteristics make it more responsive to important physical instabilities such as time-dependent turbulent vortex shedding.

The Spalart-Allmaras Model

In its original form, the Spalart-Allmaras model is effectively a low-Reynolds-number model, requiring the viscous-affected region of the boundary layer to be properly resolved. In Icepak, however, the Spalart-Allmaras model has been implemented to use wall functions when the mesh resolution is not sufficiently fine. This might make it the best choice for relatively crude simulations on coarse meshes where accurate turbulent flow computations are not critical. Furthermore, the near-wall gradients of the transported variable in the model are much smaller than the gradients of the transported variables in the $k-\varepsilon$ models. This might make the model less sensitive to numerical error when non-layered meshes are used near walls.

Transport Equation for the Spalart-Allmaras Model

The transported variable in the Spalart-Allmaras model, $\tilde{v}$, is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region. The transport equation for $\tilde{v}$ is

Equation 17

$$\frac{\partial}{\partial t}(\rho\tilde{v})+\frac{\partial}{\partial x_i}(\rho\tilde{v}u_i)=G_\nu+\frac{1}{\sigma_{\tilde{v}}}\left[\frac{\partial}{\partial x_j}\left\{(\mu+\rho\tilde{v})\frac{\partial\tilde{v}}{\partial x_j}\right\}+C_{b2}\rho\left(\frac{\partial\tilde{v}}{\partial x_j}\right)^2\right]-Y_\nu+S_{\tilde{v}}$$

where G_ν is the production of turbulent viscosity and Y_ν is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. $\sigma_{\tilde{v}}$ and C_{b2} are constants and v is the molecular kinematic viscosity. $S_{\tilde{v}}$ is a user-defined source term. Note that since the turbulence kinetic energy k is not calculated in the Spalart-Allmaras model, the last term in "Equation 16" on page 20-5 is ignored when estimating the Reynolds stresses.

Modeling the Turbulent Viscosity

The turbulent viscosity, μ_t , is computed from

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 18

$$\mu_t = \rho \tilde{v} f_{v1}$$

where the viscous damping function, f_{v1} , is given by

Equation 19

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}$$

and

Equation 20

$$\chi \equiv \frac{\tilde{v}}{\nu}$$

Modeling the Turbulent Production

The production term, G_ν , is modeled as

Equation 21

$$G_\nu = C_{b1} \rho \tilde{S} \tilde{v}$$

where

Equation 22

$$\tilde{S} \equiv S + \frac{\tilde{v}}{\kappa^2 d^2} f_{v2}$$

and

Equation 23

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$

C_{b1} and κ are constants, d is the distance from the wall, and S is a scalar measure of the deformation tensor. By default in Icepak, as in the original model proposed by Spalart and Allmaras, S is based on the magnitude of the vorticity:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 24

$$S \equiv \sqrt{2\Omega_{ij}\Omega_{ij}}$$

where Ω_{ij} is the mean rate-of-rotation tensor and is defined by

Equation 25

$$\Omega_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right)$$

The justification for the default expression for S is that, for the wall-bounded flows that were of most interest when the model was formulated, turbulence is found only where vorticity is generated near walls. However, it has since been acknowledged that one should also take into account the effect of mean strain on the turbulence production, and a modification to the model has been proposed [6] and incorporated into Icepak.

This modification combines measures of both rotation and strain tensors in the definition of S:

Equation 26

$$S \equiv |\Omega_{ij}| + C_{prod}\ \min\left(0, |S_{ij}| - |\Omega_{ij}|\right)$$

where

Equation 27

$$C_{prod} = 2.0,\ \left|\Omega_{ij}\right| \equiv \sqrt{2\Omega_{ij}\Omega_{ij}},\ \left|S_{ij}\right| \equiv \sqrt{2S_{ij}S_{ij}}$$

with the mean strain rate, S_{ij} , defined as

Equation 28

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)$$

Including both the rotation and strain tensors reduces the production of eddy viscosity and consequently reduces the eddy viscosity itself in regions where the measure of vorticity exceeds that of strain rate. One such example can be found in vortical flows, that is, flow near the core of a vortex subjected to a pure rotation where turbulence is known to be suppressed. Including both the rotation and strain tensors more correctly accounts for the effects of rotation on turbulence. The default option (including the rotation tensor only) tends to overpredict the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

production of eddy viscosity and hence overpredicts the eddy viscosity itself in certain circumstances.

Modeling the Turbulent Destruction

The destruction term is modeled as

Equation 29

$$Y_\nu = C_{w1}\rho f_w \left(\frac{\tilde{v}}{d}\right)^2$$

where

Equation 30

$$f_w = g\left[\frac{1+C_{w3}^6}{g^6+C_{w3}^6}\right]^{1/6}$$

Equation 31

$$g = r + C_{w2}\left(r^6 - r\right)$$

Equation 32

$$r \equiv \frac{\tilde{v}}{\tilde{S}\kappa^2 d^2}$$

C_{w1} , C_{w2} , and C_{w3} are constants, and $\tilde{S}$ is given by "Equation 22" on page 20-10. Note that the modification described above to include the effects of mean strain on S will also affect the value of $S\tilde{}$ used to compute r.

Model Constants

The model constants $C_{b1}, C_{b2}, \sigma_{\tilde{v}}, C_{v1}, C_{w1}, C_{w2}, C_{w3}$ and κ have the following default values [26]:

Equation 33

$$C_{b1} = 0.1355, C_{b2} = 0.622, \sigma_{\tilde{v}} = \frac{2}{3}, C_{v1} = 7.1$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 34

$$C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{(1 + C_{b2})}{\sigma_{\tilde{v}}}, C_{w2} = 0.3, C_{w3} = 2.0, \kappa = 0.4187$$

Wall Boundary Conditions

At walls, the modified turbulent kinematic viscosity, $\tilde{v}$, is set to zero.

When the mesh is fine enough to resolve the laminar sublayer, the wall shear stress is obtained from the laminar stress-strain relationship:

Equation 35

$$\frac{u}{u_\tau} = \frac{\rho u_\tau y}{\mu}$$

If the mesh is too coarse to resolve the laminar sublayer, it is assumed that the centroid of the wall-adjacent cell falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

Equation 36

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho u_\tau y}{\mu} \right)$$

where u is the velocity parallel to the wall, u_τ is the shear velocity, y is the distance from the wall, κ is the von Kármán constant (0.4187), and E = 9.793.

Convective Heat and Mass Transfer Modeling

In Icepak, turbulent heat transport is modeled using the concept of Reynolds' analogy to turbulent momentum transfer. The modeled energy equation is thus given by the following:

Equation 37

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}\left[u_i(\rho E + p)\right] = \frac{\partial}{\partial x_j}\left[\left(k + \frac{c_p \mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_j} + u_i(\tau_{ij})_{eff}\right] + S_h$$

where k, in this case, is the thermal conductivity, E is the total energy, and $(\tau_{ij})_{eff}$ is the deviatoric stress tensor, defined as

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 38

$$(\tau_{ij})_{eff} = \mu_{eff}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\mu_{eff}\frac{\partial u_i}{\partial x_i}\delta_{ij}$$

The term involving $(\tau_{ij})_{eff}$ represents the viscous heating. The default value of the turbulent Prandtl number is 0.85. Turbulent mass transfer is treated similarly, with a default turbulent Schmidt number of 0.7.

Wall boundary conditions for scalar transport are handled analogously to momentum, using the appropriate law-of-the-wall.

Two-Equation (Standard $k-\varepsilon$) Turbulence Model

The two-equation turbulence model (also known as the standard $k-\varepsilon$ model) is more complex than the zero-equation model. The standard $k-\varepsilon$ model is a semi-empirical model based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the standard $k-\varepsilon$ model, it is assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard $k-\varepsilon$ model is therefore valid only for fully turbulent flows.

Transport Equations for the Standard $k-\varepsilon$ Model

The turbulent kinetic energy,k, and its rate of dissipation, ε, are obtained from the following transport equations:

Equation 39

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k + G_b - \rho\varepsilon$$

and

Equation 40

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

In these equations, G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients, calculated as described later in this section. G_b is the generation of turbulent

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kinetic energy due to buoyancy, calculated as described later in this section. $C_{1\varepsilon}$,$C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. ∂k and $\partial \varepsilon$ are the turbulent Prandtl numbers for k and ε, respectively.

Modeling the Turbulent Viscosity

The eddy or turbulent viscosity turbulent viscosity, μ_t , is computed by combining k and ε as follows:

Equation 41

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

where C_μ is a constant.

Model Constants

The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$. $C_{3\mu}$, σ_k , and σ_ε have the following default values [18] :

Equation 42

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

These default values have been determined from experiments with air and water for fundamental turbulent shear flows including homogeneous shear flows and decaying isotropic grid turbulence. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

The RNG $k-\varepsilon$ Model

The RNG-based $k-\varepsilon$ turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called renormalization group (RNG) methods. The analytical derivation results in a model with constants different from those in the standard $k-\varepsilon$ model, and additional terms and functions in the transport equations for k and ε. A more comprehensive description of RNG theory and its application to turbulence can be found in .renormalization group (RNG) theory [4].

Transport Equations for the RNG ε Model

The RNG $k-\varepsilon$ model has a similar form to the standard ε model:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 43

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_i}\right)+G_k+G_b-\rho\varepsilon$$

and

Equation 44

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_i}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_i}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon$$

In these equations, G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients, calculated as described later in this section. G_b is the generation of turbulent kinetic energy due to buoyancy, calculated as described later in this section. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively.

Modeling the Effective Viscosity

The scale elimination procedure in RNG theory results in a differential equation for turbulent viscosity:

Equation 45

$$d\left(\frac{\rho^2 k}{\sqrt{\varepsilon\mu}}\right)=1.72\frac{\hat{v}}{\sqrt{\hat{v}^3-1+C_\nu}}d\hat{v}$$

where

Equation 46

$$\begin{aligned}\hat{v} &= \mu_{eff}/\mu \\ C_\nu &\approx 100\end{aligned}$$

"Equation 45" above is integrated to obtain an accurate description of how the effective turbulent transport varies with the effective Reynolds number (or eddy scale), allowing the model to better handle low-Reynolds-number low-Reynolds-number flows and near-wall flows.

In the high-Reynolds-number limit, "Equation 45" above gives

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 47

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

with C_μ = 0.0845, derived using RNG theory. It is interesting to note that this value of C_μ is very close to the empirically-determined value of 0.09 used in the standard $k-\varepsilon$ model.

In Icepak, the effective viscosity is computed using the high-Reynolds-number form in "Equation 47" above.

Calculating the Inverse Effective Prandtl Numbers

The inverse effective Prandtl numbers α_k and α_ε are computed using the following formula derived analytically by the RNG theory:

Equation 48

$$\left|\frac{\alpha - 1.3929}{\alpha_0 - 1.3929}\right|^{0.6321} \left|\frac{\alpha + 2.3929}{\alpha_0 + 2.3929}\right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}}$$

where α_0 = 1.0. In the high-Reynolds-number limit ($\frac{\mu_{mol}}{\mu_{eff}}$ #1), $\alpha_k = \alpha_\varepsilon \approx 1.393$.

The R_ε Term in the ε Equation

The main difference between the RNG and standard $k-\varepsilon$ models lies in the additional term in the ε equation given by

Equation 49

$$R_\varepsilon = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \frac{\varepsilon^2}{k}$$

where $\eta \equiv Sk/\varepsilon$, η_0 = 4.38, β = 0.012.

The effects of this term in the RNG ε equation can be seen more clearly by rearranging "Equation 44" on the previous page. Using "Equation 49" above, the last two terms in "Equation 44" on the previous page can be merged, and the resulting ε equation can be rewritten as

Equation 50

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C^*_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $C_{2\varepsilon}^{*}$ is given by

Equation 51

$$C_{2\varepsilon}^{*} \equiv C_{2\varepsilon} + \frac{C_{\mu}\rho\eta^{3}\left(1-\eta/\eta_{0}\right)}{1+\beta\eta^{3}}$$

In regions where $\eta << \eta_0$, the R term makes a positive contribution, and $C_{2\varepsilon}^{*}$ becomes larger than $C_{2\varepsilon}$. In the logarithmic layer, for instance, it can be shown that $\eta \approx 3.0$, giving $C_{2\varepsilon}^{*} \approx 2.0$, which is close in magnitude to the value of $C_{2\varepsilon}$ in the standard $k-\varepsilon$ model (1.92). As a result, for weakly to moderately strained flows, the RNG model tends to give results largely comparable to the standard $k-\varepsilon$ model.

In regions of large strain rate ($\eta\ \eta_0$), however, the R term makes a negative contribution, making the value of $C_{2\varepsilon}^{*}$ less than $C_{2\varepsilon}$. In comparison with the standard $k-\varepsilon$ model, the smaller destruction of ε augments ε, reducing k and eventually the effective viscosity. As a result, in rapidly strained flows, the RNG model yields a lower turbulent viscosity than the standard $k-\varepsilon$ model.

Thus, the RNG model is more responsive to the effects of rapid strain and streamline curvature than the standard $k-\varepsilon$ model, which explains the superior performance of the RNG model for certain classes of flows.

Model Constants

The model constants $C_{1\varepsilon}$ and $C_{2\varepsilon}$ in "Equation 44" on page 20-16 have values derived analytically by the RNG theory. These values, used by default in Icepak, are $C_{1\varepsilon}$ = 1.42, $C_{2\varepsilon}$ = 1.68.

Modeling Turbulent Production in the $k-\varepsilon$ Models

From the exact equation for the transport of k, the term G_k , representing the production of turbulent kinetic energy, can be defined as

Equation 52

$$G_k = -\rho\overline{u_i' u_j'}\frac{\partial u_j}{\partial x_i}$$

To evaluate G_k in a manner consistent with the Boussinesq hypothesis,

Equation 53

$$G_k = \mu_t S^2$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where S is the modulus of the mean rate-of-strain tensor, defined as

Equation 54

$$S \equiv \sqrt{2S_{ij}S_{ij}}$$

with the mean strain rate S_{ij} given by

Equation 55

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)$$

Effects of Buoyancy on Turbulence in the $k-\varepsilon$ Models

When a non-zero gravity field and temperature gradient are present simultaneously, the $k-\varepsilon$ models in Icepak account for the generation of k due to buoyancy (G_b in "Equation 39" on page 20-14 and "Equation 43" on page 20-16), and the corresponding contribution to the production of ε in "Equation 40" on page 20-14 and "Equation 44" on page 20-16.

The generation of turbulence due to buoyancy is given by

Equation 56

$$G_b = \beta g_i \frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i}$$

where Pr_t is the turbulent Prandtl number for energy. For the standard $k-\varepsilon$ model, the default value of Pr_t is 0.85. In the case of the RNG $k-\varepsilon$ model, $Pr_t = 1/\alpha$ where α is given by "Equation 48" on page 20-17, but with $\alpha_0 = 1/Pr = k/\mu c_p$. The coefficient of thermal expansion, β, is defined as

Equation 57

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$$

It can be seen from the transport equation for k ("Equation 39" on page 20-14 or "Equation 43" on page 20-16) that turbulent kinetic energy tends to be augmented (G_b 0) in unstable stratification. For stable stratification, buoyancy tends to suppress the turbulence (G_b <0). In Ansys Icepak, the effects of buoyancy on the generation of k are always included when you have both a non-zero gravity field and a non-zero temperature (or density) gradient.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While the buoyancy effects on the generation of k are relatively well understood, the effect on ε is less clear. In Ansys Icepak, by default, the buoyancy effects on ε are neglected simply by setting Gb to zero in the transport equation for ε ("Equation 40" on page 20-14 or "Equation 44" on page 20-16).

The degree to which ε is affected by the buoyancy is determined by the constant $C_{3\varepsilon}$. In Icepak, $C_{3\varepsilon}$ is not specified, but is instead calculated according to the following relation [7]:

Equation 58

$$C_{3\varepsilon} = \tanh\left|\frac{v}{u}\right|$$

where v is the component of the flow velocity parallel to the gravitational vector and u is the component of the flow velocity perpendicular to the gravitational vector. In this way, $C_{3\varepsilon}$ will become 1 for buoyant shear layers for which the main flow direction is aligned with the direction of gravity. For buoyant shear layers that are perpendicular to the gravitational vector, 3ε will become zero.

Convective Heat Transfer Modeling in the $k-\varepsilon$ Models

In Icepak, turbulent heat transport is modeled using the concept of Reynolds' analogy to turbulent momentum transfer. The modeled energy equation is thus given by the following:

Equation 59

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}\left[u_i(\rho E + p)\right] = \frac{\partial}{\partial x_i}\left(k_{eff}\frac{\partial T}{\partial x_i}\right) + S_h$$

where E is the total energy and k_{eff} is the effective conductivity.

For the standard $k-\varepsilon$ model, k_{eff} is given by

Equation 60

$$k_{eff} = k + \frac{c_p \mu_t}{Pr_t}$$

with the default value of the turbulent Prandtl number set to 0.85.

For the RNG $k-\varepsilon$ model, the effective thermal conductivity is

Equation 61

$$k_{eff} = \alpha c_p \mu_{eff}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where α is calculated from "Equation 48" on page 20-17, but with $\alpha_0 = 1/Pr = k/\mu c_p$.

The fact that α varies with μ_{mol}/μ_{eff} , as in "Equation 48" on page 20-17, is an advantage of the RNG $k-\varepsilon$ model. It is consistent with experimental evidence indicating that the turbulent Prandtl number varies with the molecular Prandtl number and turbulence [17]. "Equation 48" on page 20-17 works well across a very broad range of molecular Prandtl numbers, from liquid metals ($\mathbf{Pr} \approx 10^{-}$) to paraffin oils ($\mathbf{Pr} \approx 10^{3}$), which allows heat transfer to be calculated in low-Reynolds-number regions. "Equation 48" on page 20-17 smoothly predicts the variation of effective Prandtl number from the molecular value ($\alpha = / \mathbf{Pr}$) in the viscosity-dominated region to the fully turbulent value (α = 1.393) in the fully turbulent regions of the flow.

$k-\omega$ SST Model

The $k-\omega$ SST model is based on the coupling of the SST $k-\omega$ transport equations, one for the intermittency and one for the transition onset criteria, in terms of momentum-thickness Reynolds number. An Ansys empirical correlation (Langtry and Menter) has been developed to cover standard bypass transition as well as flows in low-free-stream turbulence environments. The $k-\omega$ SST model can be used as a low Reynolds number turbulence model. It also exhibits good behavior in adverse pressure gradients and separating flow.

Transport Equations for the Transition SST Model

The transport equation for the intermittency γ is defined as:

Equation 62

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial(\rho U_j \gamma)}{\partial x_j} = P_{\gamma 1} - E_{\gamma 1} + P_{\gamma 2} - E_{\gamma 2} + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\gamma}\right)\frac{\partial \gamma}{\partial x_j}\right]$$

The transition sources are defined as follows:

Equation 63

$$\begin{aligned} P_{\gamma 1} &= C_{a1} F_{length} \rho S [\gamma F_{onset}]^{c_{\gamma 3}} \\ E_{\gamma 1} &= C_{e1} P_{\gamma 1} \gamma \end{aligned}$$

where S is the strain rate magnitude, F_{length} is an empirical correlation that controls the length of the transition region, and C_{a1} and C_{e1} hold the values of 2 and 1, respectively. The destruction/relaminarization sources are defined as follows:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 64

$$P_{\gamma 2} = C_{a2}\rho\Omega\gamma F_{turb}$$
$$E_{\gamma 2} = C_{e2}P_{\gamma 2}\gamma$$

where Ω is the vorticity magnitude. The transition onset is controlled by the following functions:

Equation 65

$$Re_V = \frac{\rho y^2 S}{\mu}$$
$$R_T = \frac{\rho k}{\mu\omega}$$

Equation 66

$$F_{onset1} = \frac{Re_v}{2.193 Re_{\theta c}}$$
$$F_{onset2} = min\left(max\left(F_{onset1}, F_{onset1}^4\right), 2.0\right)$$

Equation 67

$$F_{onset3} = max\left(1 - \left(\frac{R_T}{2.5}\right)^3, 0\right)$$
$$F_{onset} = max\left(F_{onset2} - F_{onset3}, 0\right)$$
$$F_{turb} = e^{-\left(\frac{R_T}{4}\right)^4}$$

$Re_{\theta c}$ is the critical Reynolds number where the intermittency first starts to increase in the boundary layer. This occurs upstream of the transition Reynolds number $\tilde{Re_{\theta t}}$ and the difference between the two must be obtained from an empirical correlation. Both the F_{length} and $Re_{\theta c}$ correlations are functions of $\tilde{Re_{\theta t}}$.

The constants for the intermittency equation are:

Equation 68

$$C_{a1} = 2;\ C_{e1} = 1;\ C_{a2} = 0.06;\ C_{e2} = 50;\ c_{\gamma 3} = 0.5;\ \sigma_\gamma = 1.0$$

Separation-Induced Transition Correction

The modification for separation-induced transition is:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 69

$$\begin{aligned} \gamma_{sep} &= min\left(C_{s1} max\left[\left(\frac{Re_v}{3.235 Re_{\theta c}}\right) - 1, 0\right] F_{reattch}, 2\right) F_{\theta t} \\ F_{reattch} &= e^{-\left(\frac{R_T}{20}\right)^4} \\ \gamma_{eff} &= max\left(\gamma, \gamma_{sep}\right) \end{aligned}$$

Here, C_{s1} is a constant with a value of 2.

The model constants in "Equation 69" above have been adjusted from those of Menter et al. [XmenterICE] in order to improve the predictions of separated flow transition. The main difference is that the constant that controls the relation between Re_v and $Re_{\theta c}$ was changed from 2.193, its value for a Blasius boundary layer, to 3.235, the value at a separation point where the shape factor is 3.5 [XmenterICE]. The boundary condition for γ at a wall is zero normal flux, while for an inlet, γ is equal to 1.0.

The transport equation for the transition momentum thickness Reynolds number $\tilde{Re_{\theta t}}$ is

Equation 70

$$\frac{\partial\left(\rho\widetilde{Re_{\theta t}}\right)}{\partial t} + \frac{\partial\left(\rho U_j \widetilde{Re_{\theta t}}\right)}{\partial x_j} = P_{\theta t} + \frac{\partial}{\partial x_j}\left[\sigma_{\theta t}\left(\mu + \mu_t\right)\frac{\partial\widetilde{Re_{\theta t}}}{\partial x_j}\right]$$

The source term is defined as follows:

Equation 71

$$\begin{aligned} P_{\theta t} &= c_{\theta t}\frac{\rho}{t}\left(Re_{\theta t} - \tilde{Re_{\theta t}}\right)\left(1.0 - F_{\theta t}\right) \\ t &= \frac{500\mu}{\rho U^2} \end{aligned}$$

Equation 72

$$F_{\theta t} = min\left(max\left(F_{wake} e^{\left(-\frac{y}{\delta}\right)^4}, 1.0 - \left(\frac{\gamma - 1/50}{1.0 - 1/50}\right)^2\right), 1.0\right)$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 73

$$\begin{aligned}\theta_{BL} &= \frac{\widetilde{Re_{\theta t}}\mu}{\rho U}\\ \delta_{BL} &= \frac{15}{2}\theta_{BL}\\ \delta &= \frac{50\Omega y}{U}\delta_{BL}\end{aligned}$$

Equation 74

$$\begin{aligned}Re_{\omega} &= \frac{\rho\omega y^2}{\mu}\\ F_{wake} &= e^{-\left(\frac{Re_{\omega}}{1E+5}\right)^2}\end{aligned}$$

The model constants for the $\tilde{Re_{\theta t}}$ equation are:

Equation 75

$$c_{\theta t} = 0.03\sigma_{\theta t} = 2.0$$

The boundary condition for $\tilde{Re_{\theta t}}$ at a wall is zero flux. The boundary condition for $\tilde{Re_{\theta t}}$ at an inlet should be calculated from the empirical correlation based on the inlet turbulence intensity.

The model contains three empirical correlations. $Re_{\theta t}$ is the transition onset as observed in experiments. This has been modified from Menter et al.[XmenterICE] in order to improve the predictions for natural transition. It is used in "Equation 70" on the previous page. F_{length} is the length of the transition zone and is substituted in "Equation 62" on page 20-21. $Re_{\theta c}$ is the point where the model is activated in order to match both $Re_{\theta t}$ and F_{length} , and is used in "Equation 66" on page 20-22. At present, these empirical correlations are proprietary and are not given in this User's guide.

Equation 76

$$\begin{aligned}Re_{\Theta t} &= f(Tu, \lambda)\\ F_{length} &= f\left(\widetilde{Re_{\Theta t}}\right)\\ Re_{\Theta c} &= f\left(\widetilde{Re_{\Theta t}}\right)\end{aligned}$$

The first empirical correlation is a function of the local turbulence intensity, Tu, and the Thwaites' pressure gradient coefficient λ_{θ} is defined as

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 77

$$\lambda_\theta = \left(\theta^2 / \nu \right) dU / ds$$

where dU / ds is the acceleration in the streamwise direction.

Coupling the Transition Model and SST Transport Equations

The transition model interacts with the SST turbulence model by modification of the k-equation, as follows:

Equation 78

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k^* - Y_k^* + S_k$$

Equation 79

$$G_k^* = \gamma_{eff} \widetilde{G_k}$$

Equation 80

$$Y_k^* = min\left(max\left(\gamma_{eff}, 0.1\right), 1.0\right) Y_k$$

where $\widetilde{G_{\tilde{k}}}$ and Y_k are the original production and destruction terms for the SST model. Note that the production term in the ω -equation is not modified. The rationale behind the above model formulation is given in detail in Menter et al. [XmenterICE].

In order to capture the laminar and transitional boundary layers correctly, the mesh must have a y^+ of approximately one. If the y^+ is too large (that is, > 5), then the transition onset location moves upstream with increasing y^+ . It is recommended that you use the bounded second order upwind based discretization for the mean flow, turbulence and transition equations.

Specifying Inlet Turbulence Levels

It has been observed that the turbulence intensity specified at an inlet can decay quite rapidly depending on the inlet viscosity ratio (μ_t / μ) (and hence turbulence eddy frequency). As a result, the local turbulence intensity downstream of the inlet can be much smaller than the inlet value (see "Figure 1: Exemplary Decay of Turbulence Intensity (Tu) as a Function of Streamwise Distance (x)" on page 20-27). Typically, the larger the inlet viscosity ratio, the smaller the turbulent decay rate. However, if too large a viscosity ratio is specified (that is, > 100), the skin

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

friction can deviate significantly from the laminar value. There is experimental evidence that suggests that this effect occurs physically; however, at this point it is not clear how accurately the transition model reproduces this behavior. For this reason, if possible, it is desirable to have a relatively low (that is, $\approx 1 - 10$) inlet viscosity ratio and to estimate the inlet value of turbulence intensity such that at the leading edge of the blade/airfoil, the turbulence intensity has decayed to the desired value. The decay of turbulent kinetic energy can be calculated with the following analytical solution:

Equation 81

$$k = k_{inlet}(1 + \omega_{inlet}\beta t)^{\frac{-\beta^*}{\beta}}$$

For the SST turbulence model in the freestream the constants are:

Equation 82

$$\beta = 0.09, \beta^* = 0.0828$$

The time scale can be determined as follows:

Equation 83

$$t = \frac{x}{V}$$

where x is the streamwise distance downstream of the inlet and V is the mean convective velocity. The eddy viscosity is defined as:

Equation 84

$$\mu_t = \frac{\rho k}{\omega}$$

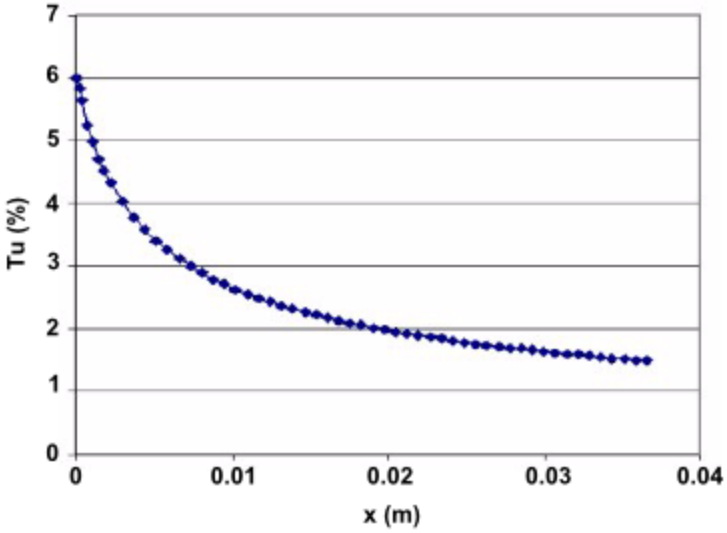
The decay of turbulent kinetic energy equation can be rewritten in terms of inlet turbulence intensity ($T_{u_{inlet}}$) and inlet eddy viscosity ratio (μ_t / μ) as follows:

Equation 85

$$T_u = \left(T_{u_{inlet}}^2 \left[1 + \frac{3\rho V x \beta T_{u_{inlet}}^2}{2\mu(\mu_t / \mu)_{inlet}} \right]^{\frac{-\beta^*}{\beta}} \right)^{0.5}$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: Exemplary Decay of Turbulence Intensity (Tu) as a Function of Streamwise Distance (x)



Buoyancy-Driven Flows and Natural Convection

The importance of buoyancy forces in a mixed convection flow can be measured by the ratio of the Grashof and Reynolds numbers:

Equation 86

$$\frac{\mathrm{Gr}}{\mathrm{Re}^2} = \frac{g\beta\Delta TL}{v^2}$$

When this number approaches or exceeds unity, you should expect strong buoyancy contributions to the flow. Conversely, if it is very small, buoyancy forces may be ignored in your simulation. In pure natural convection, the strength of the buoyancy-induced flow is measured by the Rayleigh number:

Equation 87

$$\mathrm{Ra} = \frac{g\beta\Delta TL^3\rho}{\mu\alpha}$$

where β is the thermal expansion coefficient:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 88

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$$

and α is the thermal diffusivity:

$$\alpha = \frac{k}{\rho c_p}$$

Rayleigh numbers less than 10^8 indicate a buoyancy-induced laminar flow, with transition to turbulence occurring over the range of 10^8 < Ra <10^{10}.

Icepak uses either the Boussinesq model or the ideal gas law in the calculation of natural-convection flows, as described in the following sections.

- "The Boussinesq Model" below
- "Incompressible Ideal Gas Law" on the facing page

The Boussinesq Model

> **Note:**
>
> When using the Boussinesq model, it might be necessary to create and assign a custom material for air if the ambient conditions of the simulation differ significantly from the ambient temperature (20 degrees Celsius) used to define the default air material used by Icepak.

By default, Icepak uses the Boussinesq model for natural-convection flows. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

Equation 89

$$(\rho - \rho_0)\, g \approx -\rho_0 \beta\, (T - T_0)\, g$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. "Equation 89" above is obtained by using the Boussinesq approximation $\rho = \rho_0\, (1 - \beta \Delta T)$ to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when β(T−T0) << 1.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Incompressible Ideal Gas Law

In Icepak, if you choose to define the density using the ideal gas law, Icepak will compute the density as

Equation 90

$$\rho = \frac{p_{op}}{\frac{R}{M}T}$$

where *R* is the universal gas constant and p_{op} is the **Operating Pressure** specified on the **Design Settings** dialog box **Advanced** tab (see Icepak Design Settings). In this form, the density depends only on the operating pressure and not on the local relative pressure field.

Definition of the Operating Density

When the Boussinesq approximation is not used, the operating density, ρ_0, appears in the body-force term in the momentum equations as $(\rho-\rho_0)g$.

This form of the body-force term follows from the redefinition of pressure in Icepak as

Equation 91

$$p'_s = p_s - \rho_0 g x$$

The hydrostatic balance in a fluid at rest is then

Equation 92

$$p'_s = 0$$

The definition of the operating density is thus important in all buoyancy-driven flows.

Radiation

Overview

The terms radiative heat transfer and thermal radiation are commonly used to describe heat transfer caused by electromagnetic (EM) waves. All materials continually emit and absorb EM waves, or photons. The strength and wavelength of emission depends on the temperature of the emitting material. At absolute zero K, no radiation is emitted from a surface. For heat transfer applications, wavelengths in the infrared spectrum are generally of greatest importance and are, therefore, the only ones considered in Icepak.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While both conduction and convection (the other basic modes of heat transfer) require a medium for transmission, radiation does not. Therefore, thermal radiation can traverse a long distance without interacting with a medium. Also, for most applications, conductive and convective heat transfer rates are linearly proportional to temperature differences. Radiative heat transfer rates, on the other hand, are (for the most part) proportional to differences in absolute temperature raised to the fourth power.

In electronics applications, where the medium of interest is typically air, any absorption (and hence re-emission) and scattering of radiation can be ignored. For such applications, therefore, only surface-to-surface radiation need be considered for analysis.

Gray-Diffuse Radiation

Icepak's radiation models assume the surfaces to be gray and diffuse. Emissivity and absorptivity of a gray surface are independent of the wavelength. Also, by Kirchhoff's law , the emissivity equals the absorptivity ($\varepsilon = \alpha$). For a diffuse surface, the reflectivity is independent of the outgoing (or incoming) directions.

As stated earlier, for applications of interest, the exchange of radiative energy between surfaces is virtually unaffected by the medium that separates them. Thus, according to the gray-body model, if a certain amount of radiation (E) is incident on a surface, a fraction (ρE) is reflected, a fraction (αE) is absorbed, and a fraction (τE) is transmitted. Because for most applications the surfaces in question are opaque to thermal radiation (in the infrared spectrum), the surfaces can be considered opaque. The transmissivity, therefore, can be neglected. It follows, from conservation of energy, that $\alpha + \rho = 1$, since $\alpha = \varepsilon$ (emissivity), and $\rho = 1 - \varepsilon$.

The Discrete Ordinates (DO) Radiation Model

The discrete ordinates (DO) radiation model solves the radiative transfer equation (RTE) for a finite number of discrete solid angles, each associated with a vector direction $\vec{s}$ fixed in the global Cartesian system (x, y, z). The DO model solves for as many transport equations as there are directions $\vec{s}$. The solution method is identical to that used for the fluid flow and energy equations.

The DO Model Equations

The DO model considers the radiative transfer equation (RTE) in the direction $\vec{s}$ as a field equation. Thus, Equation 8 is written as

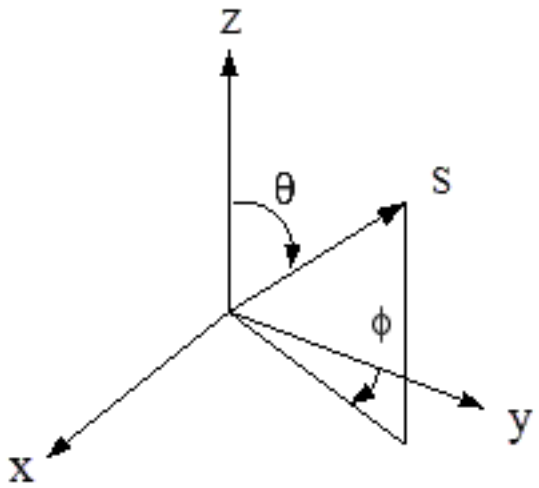
Equation 93

$$\nabla \cdot \left(I\left(\vec{r},\vec{s}\right)\vec{s}\right) + \left(a+\sigma_s\right) I\left(\vec{r},\vec{s}\right) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I\left(\vec{r},\vec{s}'\right) \Phi\left(\vec{s}\cdot\vec{s}'\right) d\Omega'$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

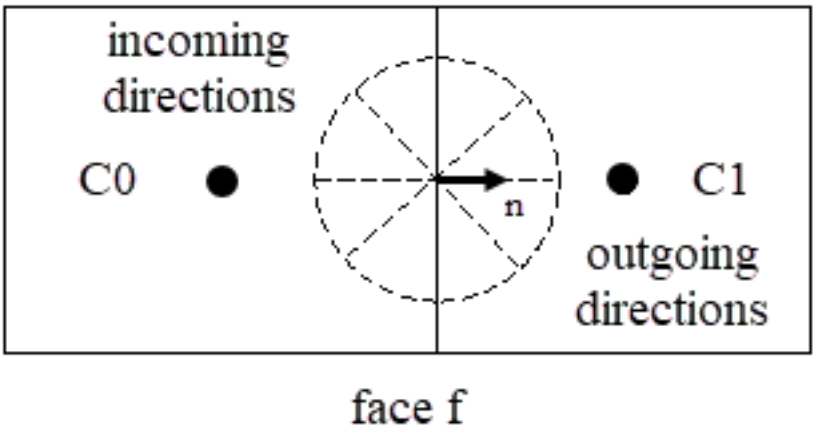
Angular Discretization and Pixelation

Each octant of the angular space 4 at any spatial location is discretized into $N \times N$ solid angles of extent, called control angles. The angles are the polar and azimuthal angles respectively, and are measured with respect to the global Cartesian system (*x*,*y*, *z*) as shown in The Angular Coordinate system, shown below. The theta and phi extents of the control angle, are constant. In two-dimensional calculations, only four octants are solved due to symmetry, making a total of directions in all. In three-dimensional calculations, a total of directions are solved. In the case of the non-gray model equations are solved for each band.

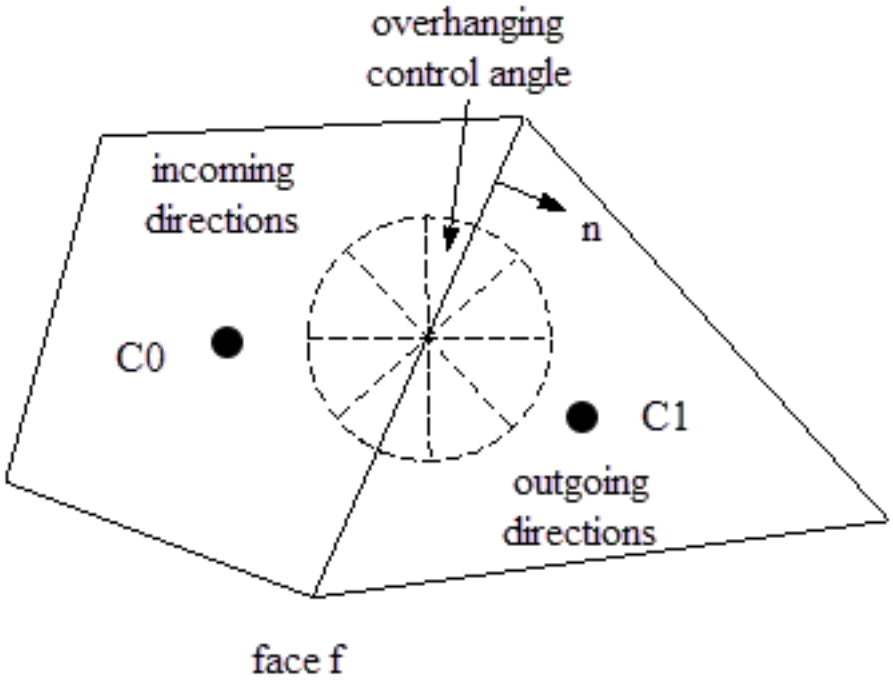


When Cartesian meshes are used, it is possible to align the global angular discretization with the control volume face, as shown in the image below.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

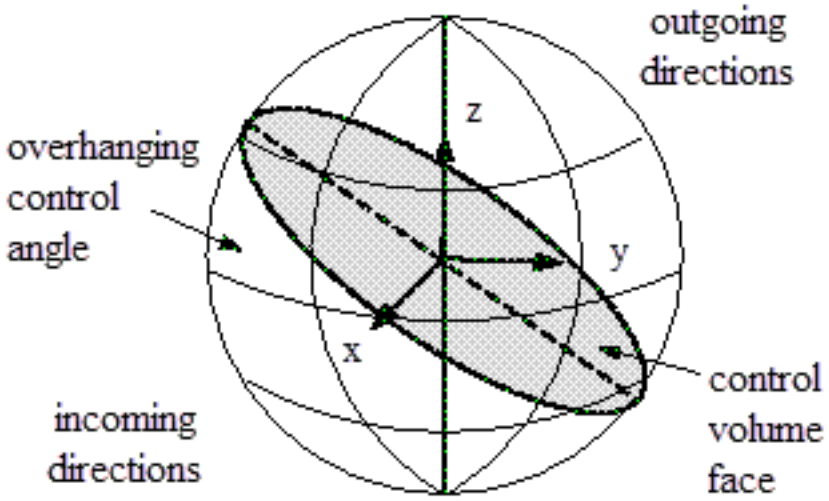


For generalized unstructured meshes however, control volume faces do not in general align with the global angular discretization, as shown in the image below leading to the problem of control angle overhang.



Essentially, control angles can straddle the control volume faces, so that they are partially outgoing to the face. The image below shows a 3D example of a face with control angle overhang.

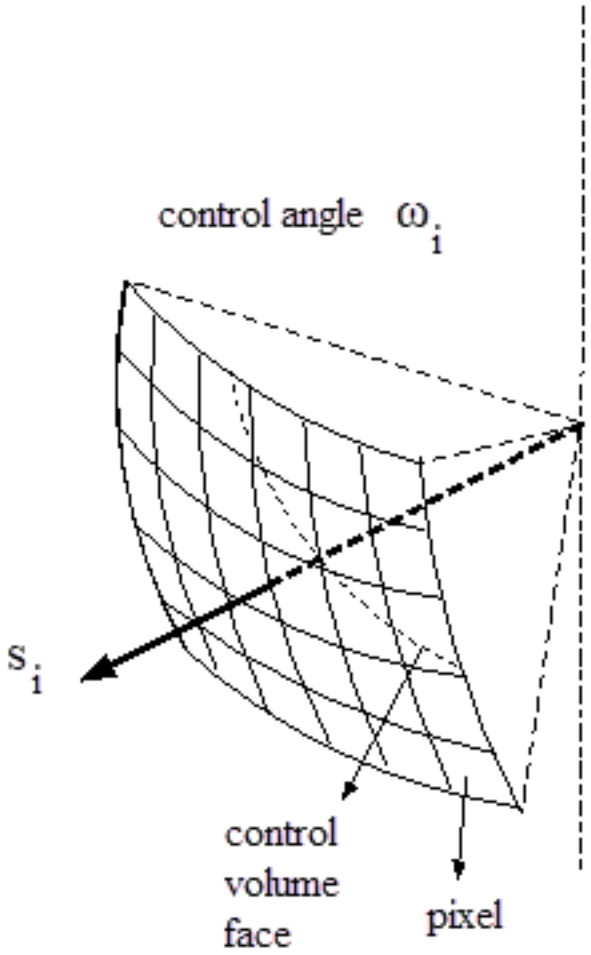
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The control volume face cuts the sphere representing the angular space at an arbitrary angle. The line of intersection is a great circle. Control angle overhang may also occur as a result of reflection and refraction. It is important in these cases to correctly account for the overhanging fraction. This is done through the use of pixelation.

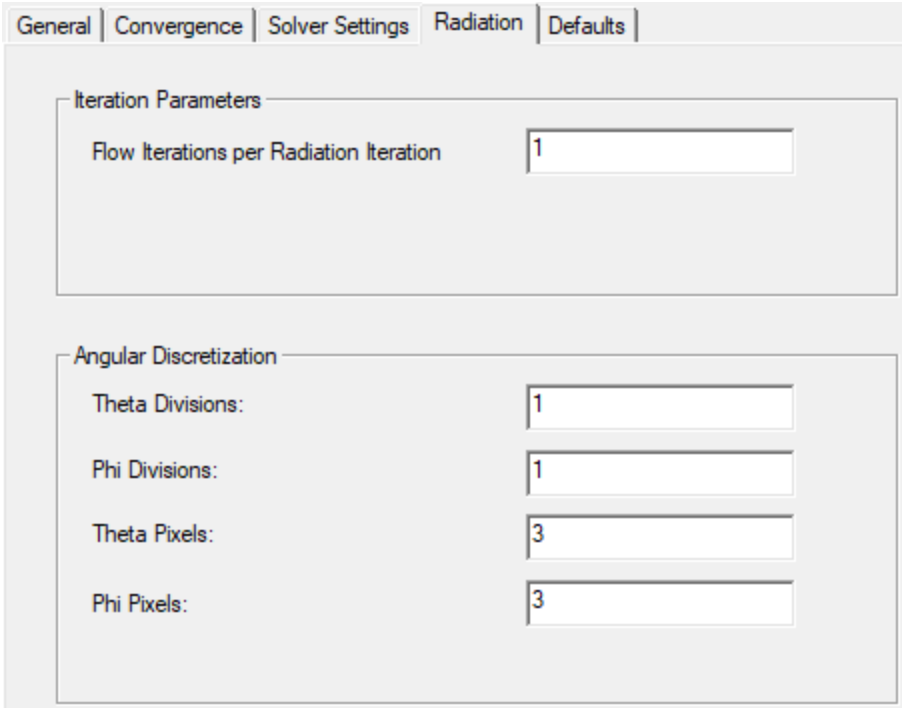
Each overhanging control angle is divided into $N_{\theta_P} \times N_{\theta_P}$ pixels, as shown in the image below.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The energy contained in each pixel is then treated as incoming or outgoing to the face. The influence of overhang can thus be accounted for within the pixel resolution. Icepak allows you to choose the pixel resolution. For problems involving gray-diffuse radiation, the default pixelation of 1 X 1 is usually sufficient. For problems involving symmetry, periodic, specular, or semi-transparent boundaries, a pixelation of 3 X 3 is recommended. For problems involving inclined or curved surfaces, a minimum pixelation of 3 X 3 is recommended. You should be aware, however that increasing the pixelation adds to the cost of computation. The pixel resolution is specified in a solution setup with **Theta Pixels** and **Phi Pixels** on the **Radiation** tab. In the following example, a 3 X 3 pixelation is specified.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Ray Tracing Radiation Model

The ray paths are calculated and stored prior to the fluid flow calculation. At each radiating face, rays are fired at discrete values of the polar and azimuthal angles. To cover the radiating hemisphere, θ is varied from 0 to $\frac{\pi}{2}$ and ϕ from 0 to 2π. Each ray is then traced to determine the control volumes it intercepts as well as its length within each control volume. This information is then stored in the radiation file (.s2s.gz), which is then read in before the fluid flow calculations begin.

Solution Procedures

Overview of Numerical Scheme

Icepak will solve the governing integral equations for mass and momentum, and (when appropriate) for energy and other scalars such as turbulence. A control-volume-based technique is used that consists of:

- Division of the domain into discrete control volumes using a computational grid.
- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables (unknowns) such as velocities, pressure, temperature, and conserved scalars.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

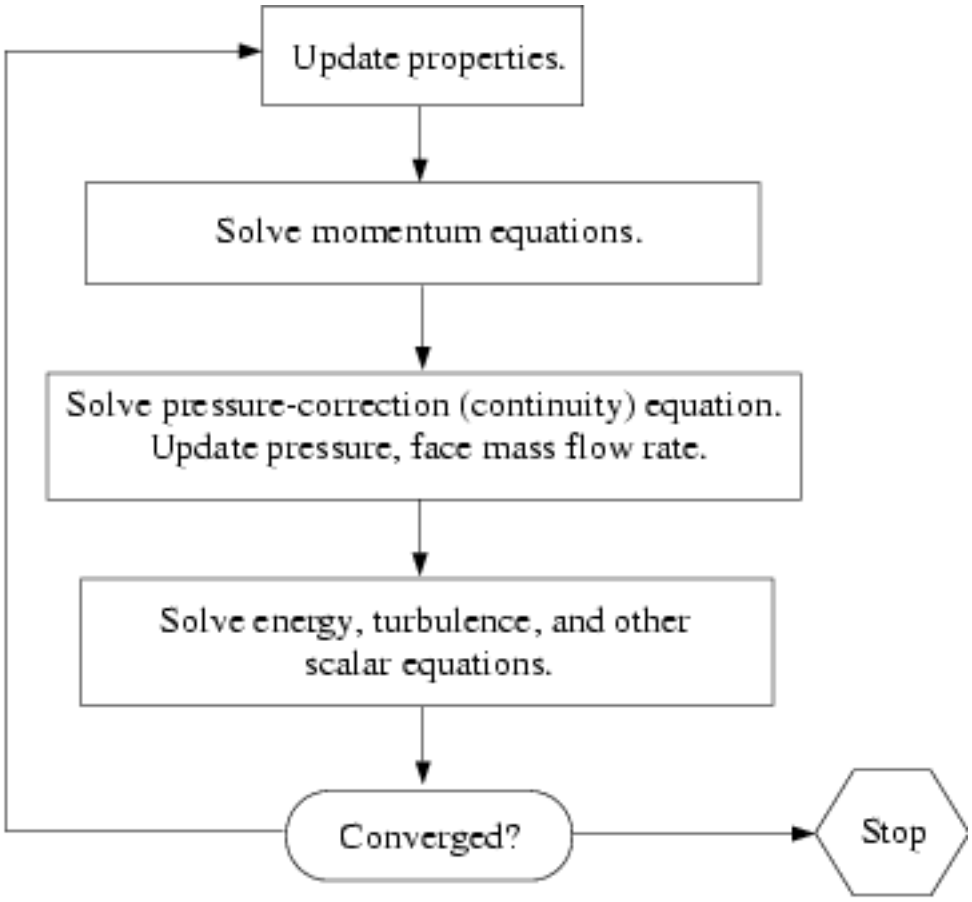
The governing equations are solved sequentially (that is, segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated in "Overview of Solution Method" on the facing page and outlined below:

1. Fluid properties are updated, based on the current solution. (If the calculation has just begun, the fluid properties will be updated based on the initialized solution.)
2. The *u*, *v* and *w* momentum equations are each solved in turn using current values for pressure and face mass fluxes, in order to update the velocity field.
3. Because the velocities obtained in Step 2 may not satisfy the continuity equation locally, a "Poisson-type" equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied.
4. Where appropriate, equations for scalars such as turbulence, energy, and radiation are solved using the previously updated values of the other variables.
5. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overview of Solution Method



Linearization

The discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized takes an "implicit" form with respect to the dependent variable (or set of variables) of interest. For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

This will result in a system of linear equations with one equation for each cell in the domain. Because there is only one equation per cell, this is sometimes called a "scalar" system of equations. A point implicit (Gauss-Seidel) linear equation solver is used in conjunction with an algebraic multigrid (AMG) method to solve the resultant scalar system of equations for the

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dependent variable in each cell. For example, the *x*-momentum equation is linearized to produce a system of equations in which *u* velocity is the unknown. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated *u*-velocity field.

In summary, Icepak solves for a single variable field (for example, *p*) by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time, and so on.

Spatial Discretization

Icepak uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis.

Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity *φ*. This is demonstrated by the following equation written in integral form for an arbitrary control volume *v* as follows:

Equation 106

$$\oint \rho\varphi\vec{v}\cdot d\vec{A} = \oint \Gamma_\varphi \nabla\varphi \cdot d\vec{A} + \int_V S_\varphi dV$$

where

ρ = density

$\vec{v}$ = velocity vector (= $u\hat{i} + v\hat{j}$ in 2D)

$\vec{A}$ = surface area vector

Γφ = diffusion coefficient for φ

∇φ = gradient of $\phi(= (\partial\phi/\partial x)\hat{i} + (\partial\phi/\partial y)\hat{j}$ in 2D)

S_φ = source of φ per unit volume

"Equation 106" above is applied to each control volume, or cell, in the computational domain. The two-dimensional, triangular cell shown in "Control Volume Used to Illustrate Discretization of a Scalar Transport Equation" on the facing page is an example of such a control volume. Discretization "Equation 106" above on a given cell yields

Equation 107

$$\sum_f^{N_{faces}} \rho\vec{v}_f\varphi_f\vec{A}_f = \sum_f^{N_{faces}} \Gamma_\varphi(\nabla\varphi)_n\vec{A}_f + S_\varphi V$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

N *faces* = number of faces enclosing cell

φ_f = value of φ convected through face f

$\vec{p v}_f \cdot \vec{A}_f$ = mass flux through the face

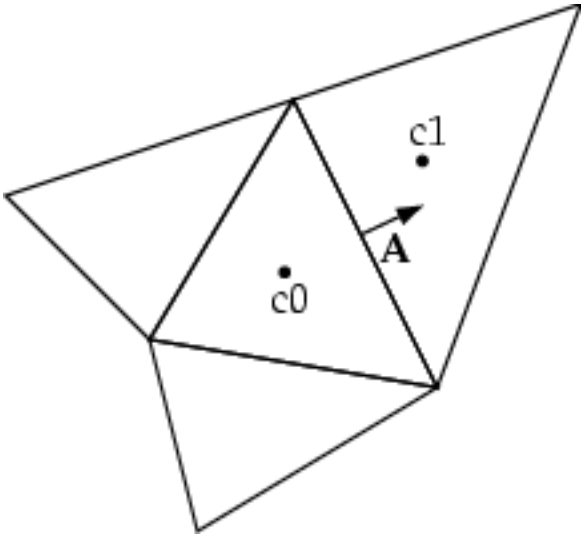
$\vec{A}_f$ = area of face f(A) (= $|A_x \hat{i} + A_x \hat{j}|$ in 2D))

$(\nabla\varphi)_n$ = ∇φ normal to face f

V = cell volume

The equations solved by Icepak take the same general form as the one given above and apply readily to multi-dimensional, unstructured meshes composed of arbitrary polyhedra.

Control Volume Used to Illustrate Discretization of a Scalar Transport Equation



Icepak stores discrete values of the scalar φ at the cell centers (c0 and c1 in "Control Volume Used to Illustrate Discretization of a Scalar Transport Equation" above). However, face values φ_f are required for the convection terms in "Equation 107" on the previous page and must be interpolated from the cell center values. This is accomplished using an upwind scheme.

Upwinding means that the face value φ_f is derived from quantities in the cell upstream, or "upwind," relative to the direction of the normal velocity v_n in "Equation 107" on the previous page. Icepak allows you to choose from two upwind schemes: first-order upwind, and second-order upwind. These schemes are described below.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The diffusion terms in "Equation 107" on page 20-38 are central-differenced and are always second-order accurate.

First-Order Upwind Scheme

When first-order accuracy is desired, quantities at cell faces are determined by assuming that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell; the face quantities are identical to the cell quantities. Thus when first-order upwinding is selected, the face value φ_f is set equal to the cell-center value of φ in the upstream cell.

Second-Order Upwind Scheme

When second-order accuracy is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach [T. J., Barth. The design and application of upwind schemes on unstructured meshes. Technical Report AIAA-89-0366. AIAA 27th Aerospace Sciences Meeting. Reno, Nevada:] . In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. Thus when second-order upwinding is selected, the face value φ_f is computed using the following expression:

Equation 108

$$\varphi_f = \varphi + \nabla\varphi \cdot \Delta\vec{s}$$

where φ and $\nabla\ \varphi$ are the cell-centered value and its gradient in the upstream cell, and Δs is the displacement vector from the upstream cell centroid to the face centroid. This formulation requires the determination of the gradient $\nabla\ \varphi$ in each cell. This gradient is computed using the divergence theorem, which in discrete form is written as

Equation 109

$$\nabla\varphi = \frac{1}{V}\sum_f^{N_{faces}} \tilde{\varphi}_f \vec{A}$$

Here the face values $\tilde{\varphi}_f$ are computed by averaging φ from the two cells adjacent to the face. Finally, the gradient $\nabla\ \varphi$ is limited so that no new maxima or minima are introduced.

Linearized Form of the Discrete Equation

The discretized scalar transport equation ("Equation 107" on page 20-38) contains the unknown scalar variable φ at the cell center as well as the unknown values in surrounding neighbor cells.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This equation will, in general, be nonlinear with respect to these variables. A linearized form of "Equation 107" on page 20-38 can be written as

Equation 110

$$a_P \varphi = \sum_{nb} a_{nb} \varphi_{nb} + b$$

where the subscript *nb* refers to neighbor cells, and a_p and a_{nb} are the linearized coefficients for φ and φ_b.

The number of neighbors for each cell depends on the grid topology, but will typically equal the number of faces enclosing the cell (boundary cells being the exception).

Similar equations can be written for each cell in the grid. This results in a set of algebraic equations with a sparse coefficient matrix. For scalar equations, Ansys Icepak solves this linear system using a point implicit (Gauss-Seidel) linear equation solver in conjunction with an algebraic multigrid (AMG) method algebraic multigrid (AMG) solvermultigrid which is described in Restriction, Prolongation, and Coarse-Level Operators.

Under-Relaxation

Because of the nonlinearity stability of the equation set being solved by Icepak, it is necessary to control the change of φ. This is typically achieved by under-relaxation, which reduces the change of φ produced during each iteration. In a simple form, the new value of the variable φ within a cell depends upon the old value, φ_{old} , the computed change in φ, $\Delta\varphi$, and the under-relaxation factor, α, as follows:

Equation 111

$$\varphi = \varphi_{old} + \alpha \Delta \varphi$$

Discretization of the Momentum and Continuity Equations

In this section, special practices related to the discretization of the momentum and continuity equations and their solution are addressed. These practices are most easily described by considering the steady-state continuity and momentum equations in integral form:

Equation 112

$$\oint \rho \vec{v} \cdot d\vec{A} = 0$$

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 113

$$\oint \rho \vec{v}\vec{v} \cdot d\vec{A} = -\oint pI \cdot d\vec{A} + \oint \bar{\bar{\tau}} \cdot d\vec{A} + \int_V \vec{F} dV$$

where I is the identity matrix, $\bar{\bar{\tau}}$ is the stress tensor, and $\vec{F}$ is the force vector.

Discretization of the Momentum Equation

The discretization scheme described earlier in this section for a scalar transport equation is also used to discretize the momentum equations. For example, the x-momentum equation can be obtained by setting φ = u:

Equation 114

$$a_P u = \sum_{nb} a_{nb} u_{nb} + \sum p_f A \cdot \hat{\imath} + S$$

If the pressure field and face mass fluxes were known, "Equation 114" above could be solved in the manner outlined earlier in this section, and a velocity field obtained. However, the pressure field and face mass fluxes are not known *a priori* and must be obtained as a part of the solution. There are important issues with respect to the storage of pressure and the discretization of the pressure gradient term; these are addressed later in this section.

Icepak uses co-located scheme, whereby pressure and velocity are both stored at cell centers. However, requires the value of the pressure at the face between cells *c0* and *c1*, shown in Control Volume Used to Illustrate Discretization of a Scalar Transport Equation. Therefore, an interpolation scheme is required to compute the face values of pressure from the cell values.

Pressure Interpolation Schemes

The default pressure interpolation scheme in Icepak is the standard scheme. This scheme interpolates the pressure values at the faces using momentum equation coefficients . This procedure works well as long as the pressure variation between cell centers is smooth. When there are jumps or large gradients in the momentum source terms between control volumes, the pressure profile has a high gradient at the cell face, and cannot be interpolated using this scheme. If this scheme is used, the discrepancy shows up in overshoots/undershoots of cell velocity.

Flows for which the standard pressure interpolation scheme will have trouble include flows with large body forces, such as in strongly swirling flows and in high-Rayleigh-number natural convection. In such cases, it is necessary to pack the mesh in regions of high gradient to resolve the pressure variation adequately.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Another source of error is that Icepak assumes that the normal pressure gradient at the wall is zero. This is valid for boundary layers, but not in the presence of body forces or curvature. Again, the failure to correctly account for the wall pressure gradient is manifested in velocity vectors pointing in/out of walls.

The other scheme available in Ansys Icepak is the body-force-weighted scheme. This scheme computes the face pressure by assuming that the normal acceleration of the fluid resulting from the pressure gradient and body forces is continuous across each face. This works well if the body forces are known a priori in the momentum equations (for example, buoyancy and axisymmetric swirl calculations). This scheme is good for high-Rayleigh-number natural convection flows.

Discretization of the Continuity Equation

"Equation 112" on page 20-41 may be integrated over the control volume in Control Volume Used to Illustrate Discretization of a Scalar Transport Equation to yield the following discrete equation

Equation 115

$$\sum_{f}^{N_{faces}} J_f A_f = 0$$

where J_f is the mass flux through face f, ρv_n.

As described in Overview of Numerical Scheme, the momentum and continuity equations are solved sequentially. In this sequential procedure, the continuity equation is used as an equation for pressure. However, pressure does not appear explicitly in "Equation 114" on the previous page for incompressible flows, since density is not directly related to pressure. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used for introducing pressure into the continuity equation. This procedure is outlined below.

To proceed further, it is necessary to relate the face values of velocity v_n to the stored values of velocity at the cell centers. Linear interpolation of cell-centered velocities to the face results in unphysical checker-boarding of pressure. Icepak uses a procedure similar to that outlined by Rhie and Chow to prevent checkerboarding. The face value of velocity v_n is not averaged linearly; instead, momentum-weighted averaging, using weighting factors based on the a_P coefficient from , is performed. Using this procedure, the face flow rate J_f may be written as

Equation 116

$$J_f = \hat{J}_f + d_f (p_{c0} - p_{c1})$$

where P_{c0} and P_{c1} are the pressures within the two cells on either side of the face, and $\hat{J}_f$ contains the influence of velocities in these cells (see Control Volume Used to Illustrate

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discretization of a Scalar Transport Equation). The term d_f is a function of $\bar{a}_P$, the average of the momentum equation a_P coefficients for the cells on either side of face *f*.

Pressure-Velocity Coupling with SIMPLE

Pressure-velocity coupling is achieved by using "Equation 114" on page 20-42 to derive an equation for pressure from the discrete continuity equation ("Equation 114" on page 20-42). Icepak uses the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) pressure-velocity coupling algorithm. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

If the momentum equation is solved with a guessed pressure field p^*, the resulting face flux $J_f{}^*$ computed from "Equation 114" on page 20-42

Equation 117

$$J_f^* = \hat{J}\ {}_f^* + d_f\left(p_{c0}^* - p_{c1}^*\right)$$

does not satisfy the continuity equation. Consequently, a correction $J_f{}'$ is added to the face flow rate $J_f{}^*$ so that the corrected face flow rate J_f

Equation 118

$$J_f = J_f^* + J_f'$$

satisfies the continuity equation. The SIMPLE algorithm postulates that $J_f{}'$ be written as

Equation 119

$$J_f' = d_f\left(p_{c0}' - p_{c1}'\right)$$

where p' is the cell pressure correction.

The SIMPLE algorithm substitutes the flux correction equations ("Equation 118" above and "Equation 119" above) into the discrete continuity equation ("Equation 114" on page 20-42) to obtain a discrete equation for the pressure correction p' in the cell:

Equation 120

$$a_P p' = \sum_{nb} a_{nb} p_{nb}' + b$$

where the source term *b* is the net flow rate into the cell:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 121

$$b = \sum_{f}^{N_{faces}} J_f^* A_f$$

The pressure-correction equation ("Equation 120" on the previous page) may be solved using the algebraic multigrid (AMG) method described in Restriction, Prolongation, and Coarse-Level Operators. Once a solution is obtained, the cell pressure and the face flow rate are corrected using

Equation 122

$$p = p^* + \alpha_p p'$$

Equation 123

$$J_f = J_f^* + d_f \left(p'_{c0} - p'_{c1}\right)$$

Here α_p is the under-relaxation factor for pressure (see "Equation 113" on page 20-42 and related description for information about under-relaxation). The corrected face flow rate J_f satisfies the discrete continuity equation identically during each iteration.

Coupled Pressure-Velocity Formulation

Selecting Coupled pressure-velocity formulation from the Solver options group box indicates that you are using the pressure-based coupled algorithm which is described in this section.

The pressure-based solver allows you to solve your flow problem in either a segregated or coupled manner. Using the coupled approach offers some advantages over the non-coupled or segregated approach. The coupled scheme obtains a robust and efficient single phase implementation for steady-state flows, with superior performance compared to the segregated SIMPLE solution scheme. For transient flows, using the coupled algorithm is necessary when the quality of the mesh is poor, or if large time steps are used.

The pressure-based segregated algorithm solves the momentum equation and pressure correction equations separately. This semi-implicit solution method results in slow convergence.

The coupled algorithm solves the momentum and pressure-based continuity equations together. The full implicit coupling is achieved through an implicit discretization of pressure gradient terms in the momentum equations, and an implicit discretization of the face mass flux, including the Rhie-Chow pressure dissipation terms.

In the momentum equations ("Equation 114" on page 20-42), the pressure gradient for component *k* is of the form

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 124

$$\sum_f p_f A_k = -\sum_j a^{u_k p} p_j$$

Where $a^{u_k p}$ is the coefficient derived from the Gauss divergence theorem and coefficients of the pressure interpolation schemes. Finally, for any i th cell, the discretized form of the momentum equation for component u_k is defined as

Equation 125

$$\sum_j a_{ij}{}^{u_k u_k} u_{kj} + \sum_j a_{ij}{}^{u_k p} p_j = b_i{}^{u_k}$$

In the continuity equation, the balance of fluxes is replaced using the flux expression in "Equation 114" on page 20-42, resulting in the discretized form

Equation 126

$$\sum_k \sum_j a_{ij}{}^{pu_k} u_{kj} + \sum_j a_{ij}{}^{pp} p_j = b_i{}^{p}$$

As a result, the overall system of equations ("Equation 125" above and "Equation 126" above), after being transformed to the δ -form, is presented as

Equation 127

$$\sum_j [A]_{ij} \vec{X}_j = \vec{B}_i$$

where the influence of a cell *i* on a cell *j* has the form

Equation 128

$$A_{ij} = \begin{bmatrix} a_{ij}^{pp} & a_{ij}^{pu} & a_{ij}^{pv} & a_{ij}^{pw} \\ a_{ij}^{up} & a_{ij}^{uu} & a_{ij}^{uv} & a_{ij}^{uw} \\ a_{ij}^{v\ p} & a_{ij}^{v\ u} & a_{ij}^{v\ v} & a_{ij}^{v\ w} \\ a_{ij}^{wp} & a_{ij}^{wu} & a_{ij}^{wv} & a_{ij}^{ww} \end{bmatrix}$$

and the unknown and residual vectors have the form

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 129

$$\vec{X}_j = \begin{bmatrix} p_i' \\ u_i' \\ v_i' \\ w_i' \end{bmatrix}$$

Equation 130

$$\vec{B}_i = \begin{bmatrix} -r_i^p \\ -r_i^u \\ -r_i^v \\ -r_i^w \end{bmatrix}$$

Note "Equation 127" on the previous page that is solved using the coupled AMG.

Pseudo Transient Under-Relaxation

The pseudo transient under-relaxation method is a form of implicit under-relaxation. Here, the under-relaxation is controlled through the pseudo time step size. The pseudo time step size, which is automatically calculated by the solver, can be the same or different for different equations solved.

Equation 131

$$\rho_p \Delta V \frac{\phi_p - \phi_p^{old}}{\Delta t} + a_p \phi_p = \sum_{nb} a_{nb} \phi_{nb} + b$$

where Δt is the pseudo time step.

Time Discretization

In Icepak the time-dependent equations must be discretized in both space and time. The spatial discretization for the time-dependent equations is identical to the steady-state case (see Spatial Discretization). Temporal discretization involves the integration of every term in the differential equations over a time step Δt. The integration of the transient terms is straightforward, as shown below.

A generic expression for the time evolution of a variable φ is given by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 132

$$\frac{\partial \varphi}{\partial t} = F(\varphi)$$

where the function *F* incorporates any spatial discretization. If the time derivative is discretized using backward differences, the first-order accurate temporal discretization is given by

Equation 133

$$\frac{\varphi^{n+1} - \varphi^{n}}{\Delta t} = F(\varphi)$$

where:

φ = a scalar quantity

n + 1 = value at the next time level, t + Δt

n = value at the current time level, t

Once the time derivative has been discretized, a choice remains for evaluating F(φ): in particular, which time level values of φ should be used in evaluating F?

One method (the method used in Icepak) is to evaluate F(φ) at the future time level:

Equation 134

$$\frac{\varphi^{n+1} - \varphi^{n}}{\Delta t} = F\left(\varphi^{n+1}\right)$$

This is referred to as implicit integration since φ^{n+1} in a given cell is related to φ^{n+1} in neighboring cells through $F(\varphi^{n+1})$:

Equation 135

$$\varphi^{n+1} = \varphi^{n} + \Delta t F\left(\varphi^{n+1}\right)$$

until φ^{i} stops changing (that is, converges). At that point, φ^{n+1} is set to φ^{i} .

The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multigrid Method

Approach

Icepak uses a multigrid scheme to accelerate the convergence of the solver by computing corrections on a series of coarse grid levels. The use of this multigrid scheme can greatly reduce the number of iterations and the CPU time required to obtain a converged solution, particularly when your model contains a large number of control volumes.

The Need for Multigrid

Implicit solution of the linearized equations on unstructured meshes is complicated by the fact that there is no equivalent of the line-iterative methods that are commonly used on structured grids. Because direct matrix inversion is out of the question for realistic problems and whole-field solvers that rely on conjugate-gradient (CG) methods have robustness problems associated with them, the methods of choice are point implicit solvers like Gauss-Seidel method. Although the Gauss-Seidel scheme rapidly removes local (high-frequency) errors in the solution, global (low-frequency) errors are reduced at a rate inversely related to the grid size. Thus, for a large number of nodes, the solver stalls and the residual reduction rate becomes prohibitively low.

Multigrid techniques allow global error to be addressed by using a sequence of successively coarser meshes. This method is based upon the principle that global (low-frequency) error existing on a fine mesh can be represented on a coarse mesh where it again becomes accessible as local (high-frequency) error: because there are fewer coarse cells overall, the global corrections can be communicated more quickly between adjacent cells. Because computations can be performed at exponentially decaying expense in both CPU time and memory storage on coarser meshes, there is the potential for very efficient elimination of global error. The fine-grid relaxation scheme or smoother, in this case either the point-implicit Gauss-Seidel or the explicit multi-stage scheme, is not required to be particularly effective at reducing global error and can be tuned for efficient reduction of local error.

The Basic Concept in Multigrid

Consider the set of discretized linear (or linearized) equations given by

Equation 136

$$A\varphi_e + b = 0$$

where φ_e is the exact solution. Before the solution has converged there will be a defect d associated with the approximate solution φ:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 137

$$A\varphi + b = d$$

We seek a correction ψ to φ such that the exact solution is given by

Equation 138

$$\varphi_e = \varphi + \psi$$

Substituting "Equation 138" above into "Equation 136" on the previous page gives

Equation 139

$$\begin{aligned} A(\varphi + \psi) + b &= 0 \\ A\psi + (A\varphi + b) &= 0 \end{aligned}$$

Now using "Equation 137" above into "Equation 139" above we obtain

Equation 140

$$A\psi + d = 0$$

which is an equation for the correction in terms of the original fine level operator A and the defect d. Assuming the local (high-frequency) errors have been sufficiently damped by the relaxation scheme on the fine level, the correction ψ will be smooth and therefore more effectively solved on the next coarser level.

Restriction and Prolongation

Solving for corrections on the coarse level requires transferring the defect down from the fine level (restriction), computing corrections, and then transferring the corrections back up from the coarse level (prolongation). We can write the equations for coarse level corrections ψ^H as

Equation 141

$$A^H \psi^H + Rd = 0$$

where A^H is the coarse level operator and R the restriction operator responsible for transferring the fine level defect down to the coarse level. Solution of "Equation 141" above is followed by an update of the fine level solution given by

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 142

$$\varphi^{new} = \varphi + P\psi^{H}$$

where *P* is the prolongation operator used to transfer the coarse level corrections up to the fine level.

Unstructured Multigrid

The primary difficulty with using multigrid on unstructured grids is the creation and use of the coarse grid hierarchy. On a structured grid, the coarse grids can be formed simply by removing every other grid line from the fine grids and the prolongation and restriction operators are simple to formulate (for example, injection and bilinear interpolation).

Multigrid Cycles

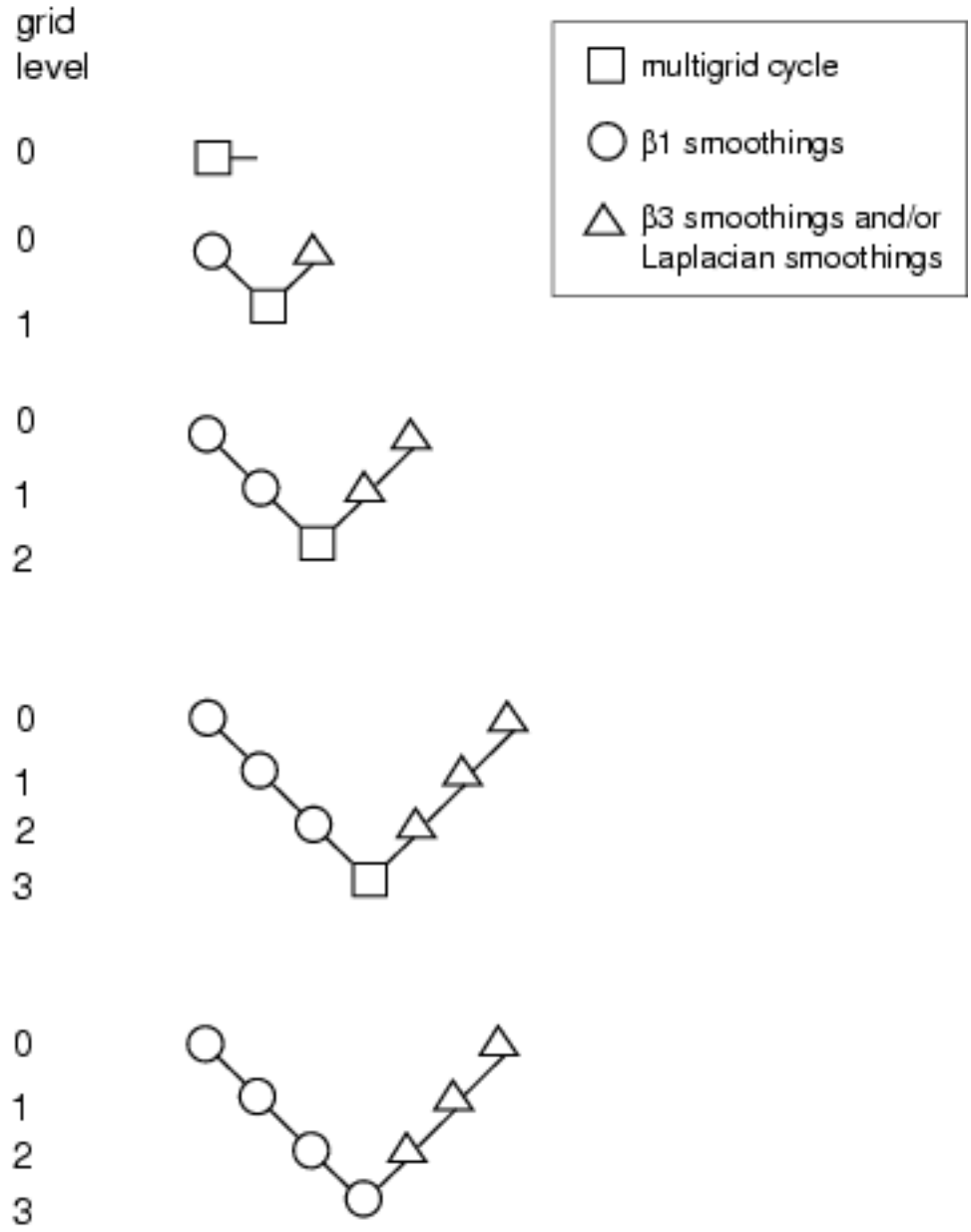
A multigrid cycle can be defined as a recursive procedure that is applied at each grid level as it moves through the grid hierarchy. Four types of multigrid cycles are available in Icepak: the V, W, F, and flexible (flex) cycles.

The V and W Cycles

V-Cycle Multigrid and W-Cycle Multigrid show the V and W multigrid cycles (defined below). In each figure, the multigrid cycle is represented by a square, and then expanded to show the individual steps that are performed within the cycle. You may want to follow along in the figures as you read the steps below.

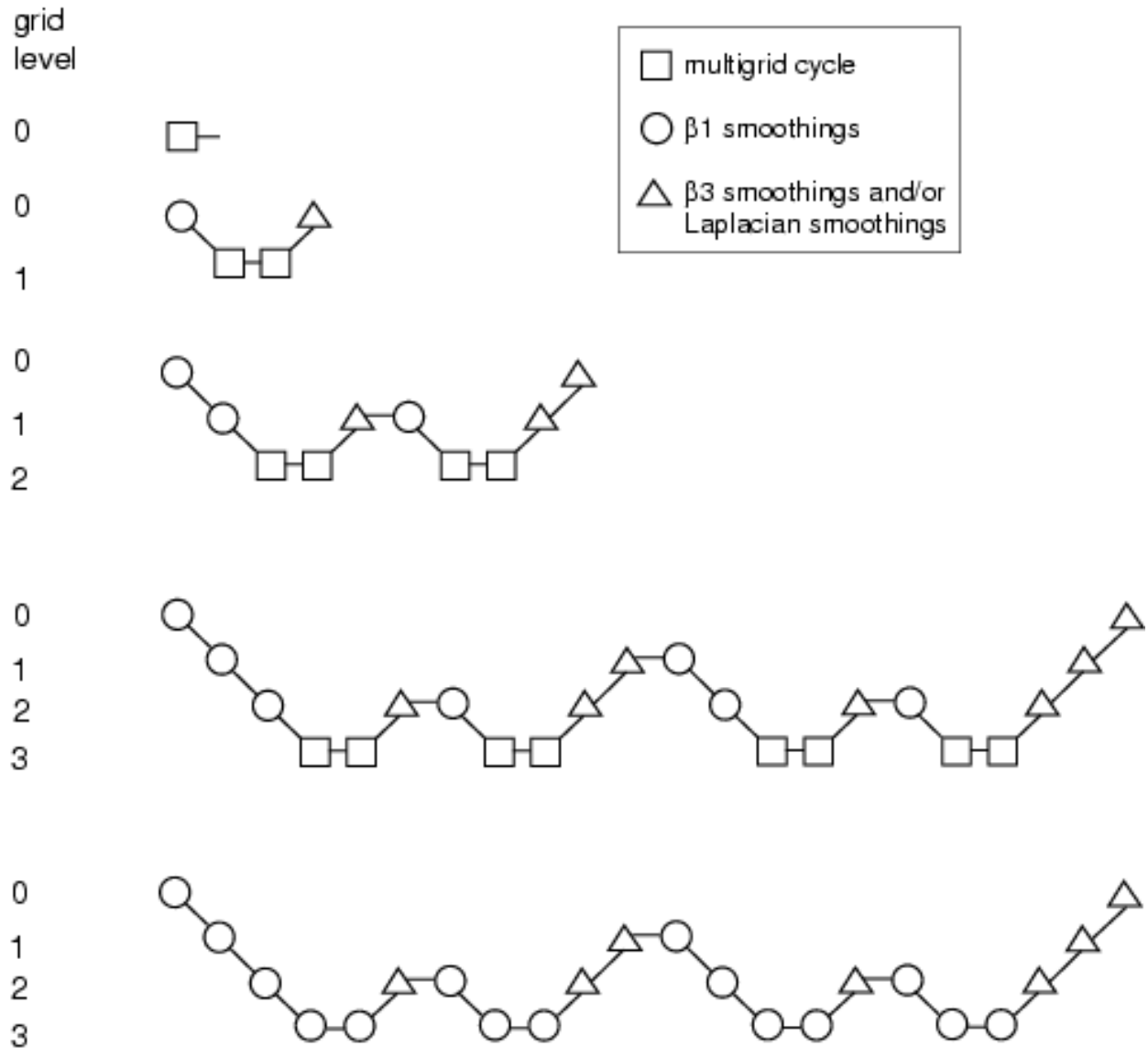
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V-Cycle Multigrid



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

W-Cycle Multigrid



For the V and W cycles, the traversal of the hierarchy is governed by three parameters, β_1 , β_2 , and β_3 , as follows:

1. β_1 " smoothings," (sometimes called pre-relaxation sweeps), are performed at the current grid level to reduce the high-frequency components of the error (local error).

   In V-Cycle Multigrid and W-Cycle Multigrid, this step is represented by a circle and marks the start of a multigrid cycle. The high-wave-number components of error should be reduced until the remaining error is expressible on the next coarser mesh without

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

significant aliasing.

If this is the coarsest grid level, then the multigrid cycle on this level is complete. (In V-Cycle Multigrid and W-Cycle Multigrid there are 3 coarse grid levels, so the square representing the multigrid cycle on level 3 is equivalent to a circle, as shown in the final diagram in each figure.)

> **Note:**
>
> In Icepak, β1 is zero (that is, pre-relaxation is not performed).

2. Next, the problem is "restricted" to the next coarser grid level using the appropriate restriction operator.

   In V-Cycle Multigrid and W-Cycle Multigrid, the restriction from a finer grid level to a coarser grid level is designated by a downward-sloping line.
3. The error on the coarse grid is reduced by performing β_2 multigrid cycles (represented in V-Cycle Multigrid and W-Cycle Multigrid and as squares). Commonly, for fixed multigrid strategies β_2 is either 1 or 2, corresponding to V-cycle and W-cycle multigrid, respectively.
4. Next, the cumulative correction computed on the coarse grid is "interpolated" back to the fine grid using the appropriate prolongation operator and added to the fine grid solution.

   In V-Cycle Multigrid and W-Cycle Multigrid, the prolongation is represented by an upward-sloping line.

   The high-frequency error now present at the fine grid level is due to the prolongation procedure used to transfer the correction.
5. In the final step, β_3 smoothings (post-relaxations) are performed to remove the high-frequency error introduced on the coarse grid by the β_2 multigrid cycles.

   In V-Cycle Multigrid and W-Cycle Multigrid, this relaxation relaxation scheme procedure is represented by a single triangle.

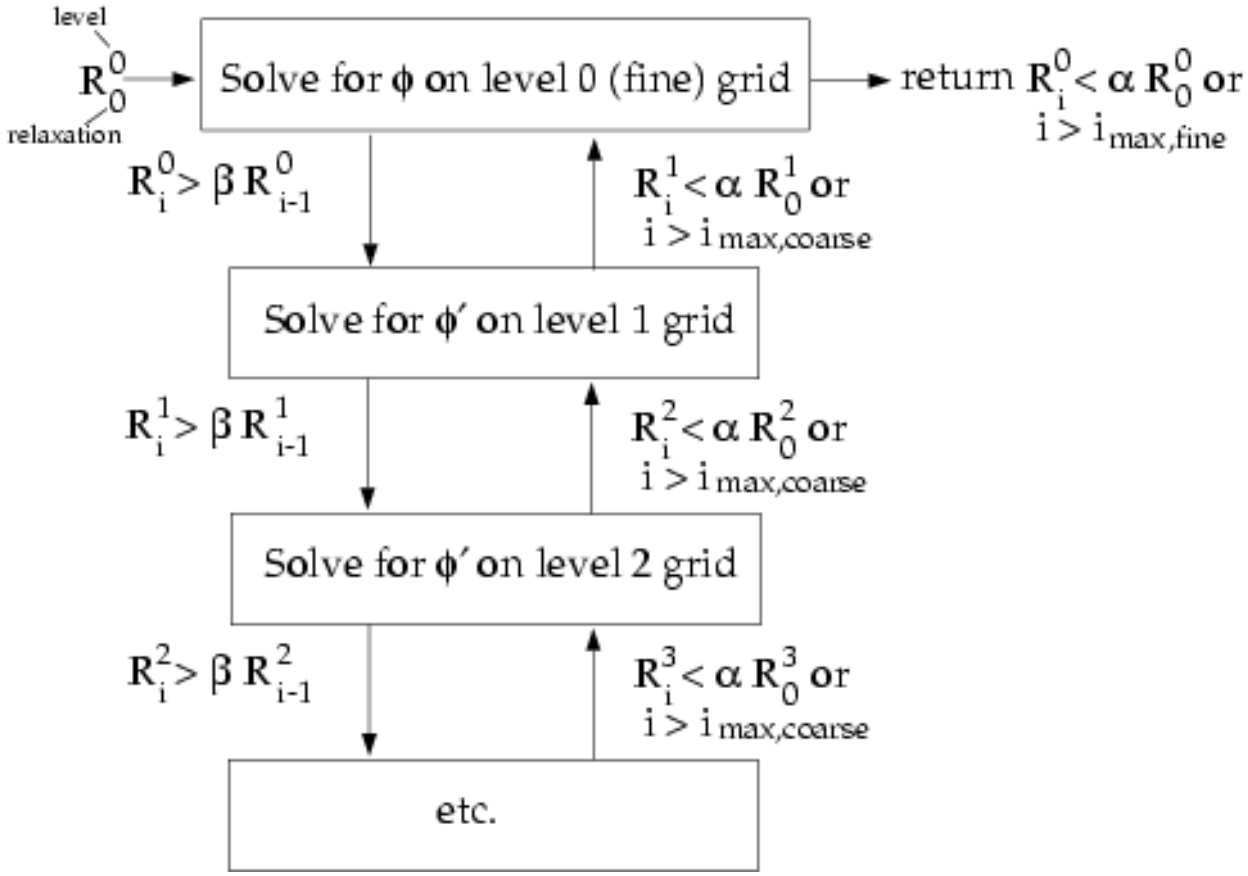
The Flexible Cycle

For the flexible cycle, the calculation and use of coarse grid corrections is controlled in the multigrid procedure by the logic illustrated in the following image. This logic ensures that coarser grid calculations are invoked when the rate of residual reduction multigrid solverresidual reduction rate on the current grid level is too slow. In addition, the multigrid controls dictate when the iterative solution of the correction on the current coarse grid level is sufficiently converged and should thus be applied to the solution on the next finer grid. These two decisions are controlled by the parameters α and β shown in the following image, as described in detail below. Note that the logic of the multigrid procedure is such that grid levels may be visited repeatedly

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

during a single global iteration on an equation. For a set of 4 multigrid levels, referred to as 0, 1, 2, and 3, the flex-cycle multigrid procedure for solving a given transport equation might consist of visiting grid levels as 0-1-2-3-2-3-2-1-0-1-2-1-0, for example.

Logic Controlling the Flex Multigrid Cycle



The main difference between the flexible cycle and the V and W cycles is that the satisfaction of the residual reduction tolerance and termination criterion determine when and how often each level is visited in the flexible cycle, whereas in the V and W cycles the traversal pattern is explicitly defined.

The F Cycles

The multigrid F cycle is essentially a combination of the V and W cycles described in "Multigrid Cycles" on page 20-51.

Recall that the multigrid cycle is a recursive procedure. The procedure is expanded to the next coarsest grid level by performing a single multigrid cycle on the current level. Referring to V-Cycle Multigrid and W-Cycle Multigrid, this means replacing the square on the current level

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(representing a single cycle) with the procedure shown for the 0–1 level cycle (the second diagram in each figure). We see that a V cycle consists of:

$$\textbf{pre sweep} \rightarrow \textbf{restrict} \rightarrow \textbf{V cycle} \rightarrow \textbf{prolongate} \rightarrow \textbf{post sweep}$$

and a W cycle:

$$\textbf{pre sweep} \rightarrow \textbf{restrict} \rightarrow \textbf{W cycle} \rightarrow \textbf{W cycle} \rightarrow \textbf{prolongate} \rightarrow \textbf{post sweep}$$

An F cycle is formed by a W cycle followed by a V cycle:

$$\textbf{pre sweep} \rightarrow \textbf{restrict} \rightarrow \textbf{W cycle} \rightarrow \textbf{V cycle} \rightarrow \textbf{prolongate} \rightarrow \textbf{post sweep}$$

As expected, the F cycle requires more computation than the V cycle, but less than the W cycle. However, its convergence properties turn out to be better than the V cycle and roughly equivalent to the W cycle. The F cycle is the default AMG cycle for the coupled equation set and for the scalar energy equation.

The Residual Reduction Rate Criteria

The multigrid procedure invokes calculations on the next coarser grid level when the error reduction rate on the current level is insufficient, as defined by

Equation 143

$$R_i > \beta R_{i-1}$$

Here R_i is the absolute sum of residuals (defect) computed on the current grid level after the relaxation on this level. The above equation states that if the residual present in the iterative solution after i relaxations is greater than some fraction, β (between 0 and 1), of the residual present after the $(i - 1)$th relaxation, the next coarser grid level should be visited. Thus β is referred to as the residual reduction tolerance, and determines when to give up on the iterative solution at the current grid level and move to solving the correction equations on the next coarser grid. The value of β controls the frequency with which coarser grid levels are visited. The default value is 0.1. A larger value will result in less frequent visits, and a smaller value will result in more frequent visits.

The Termination Criteria

Provided that the residual reduction rate is sufficiently rapid, the correction equations will be converged on the current grid level and the result applied to the solution field on the next finer grid level.

The correction equations on the current grid level are considered sufficiently converged when the error in the correction solution is reduced to some fraction, α (between 0 and 1), of the original error on this grid level:

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 144

$$R_i < \alpha R_0$$

Here, R_i is the residual on the current grid level after the ith iteration on this level, and R_0 is the residual that was initially obtained on this grid level at the current global iteration. The parameter α , referred to as the termination criterion, has a default value of 0.1. Note that the above equation is also used to terminate calculations on the lowest (finest) grid level during the multigrid procedure. Thus, relaxations are continued on each grid level (including the finest grid level) until the criterion of this equation is obeyed (or until a maximum number of relaxations has been completed, in the case that the specified criterion is never achieved).

Restriction, Prolongation, and Coarse-Level Operators

The multigrid algorithm in Ansys Icepak is referred to as an "algebraic" multigrid (AMG) scheme because, as we shall see, the coarse level equations are generated without the use of any geometry or re-discretization on the coarse levels; a feature that makes AMG particularly attractive for use on unstructured meshes. The advantage is that no coarse grids have to be constructed or stored, and no fluxes or source terms need be evaluated on the coarse levels. This approach is in contrast with FAS (sometimes called "geometric") multigrid in which a hierarchy of meshes is required and the discretized equations are evaluated on every level. In theory, the advantage of FAS over AMG is that the former should perform better for non-linear problems since non-linearities in the system are carried down to the coarse levels through the re-discretization; when using AMG, once the system is linearized, non-linearities are not felt by the solver until the fine level operator is next updated.

AMG Restriction and Prolongation Operators

The restriction and prolongation operators used here are based on the additive correction (AC) strategy described for structured grids by Hutchinson and Raithby . Inter-level transfer is accomplished by piecewise constant interpolation and prolongation. The defect in any coarse level cell is given by the sum of those from the fine level cells it contains, while fine level corrections are obtained by injection of coarse level values. In this manner the prolongation operator is given by the transpose of the restriction operator

Equation 145

$$P = R^T$$

The restriction operator is defined by a coarsening or "grouping" of fine level cells into coarse level ones. In this process each fine level cell is grouped with one or more of its "strongest" neighbors, with a preference given to currently ungrouped neighbors. The algorithm attempts to collect cells into groups of fixed size, typically two or four, but any number can be specified. In

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the context of grouping, strongest refers to the neighbor *j* of the current cell *i* for which the coefficient A_{ij} is largest. For sets of coupled equations A_{ij} is a block matrix and the measure of its magnitude is simply taken to be the magnitude of its first element. In addition, the set of coupled equations for a given cell are treated together and not divided amongst different coarse cells. This results in the same coarsening for each equation in the system.

AMG Coarse Level Operator

he coarse level operator A^H is constructed using a Galerkin approach. Here we require that the defect associated with the corrected fine level solution must vanish when transferred back to the coarse level. Therefore we may write

Equation 146

$$Rd^{new} = 0$$

Upon substituting "Equation 137" on page 20-50 and "Equation 142" on page 20-51 for d^{new} and φ^{new} we have

Equation 147

$$\begin{aligned} R\left[A\varphi^{new} + b\right] &= 0 \\ R\left[A\left(\varphi + P\psi^H\right) + b\right] &= 0 \end{aligned}$$

Now rearranging and using "Equation 137" on page 20-50 once again gives

Equation 148

$$\begin{aligned} RAP\psi^H + R\left(A\varphi + b\right) &= 0 \\ RAP\psi^H + Rd &= 0 \end{aligned}$$

Comparison of "Equation 148" above with "Equation 137" on page 20-50 leads to the following expression for the coarse level operator:

Equation 149

$$A^H = RAP$$

The construction of coarse level operators thus reduces to a summation of diagonal and corresponding off-diagonal blocks for all fine level cells within a group to form the diagonal block of that group's coarse cell.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Residuals

During the solution process you can monitor the convergence dynamically by checking residuals. At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. This history is also saved in the data file. The residual sum is defined below.

On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value ("round-off") and then stop changing ("level out"). For single precision computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round-off. Double precision residuals can drop up to twelve orders of magnitude.

After discretization, the conservation equation for a general variable φ at a cell P can be written as

Equation 150

$$a_P \varphi_P = \sum_{nb} a_{nb} \varphi_{nb} + b$$

Here a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, and b is the contribution of the constant part of the source term Sc in $S = S_c + S_P\varphi$ and of the boundary conditions. In "Equation 150" above,

Equation 151

$$a_P = \sum_{nb} a_{nb} - S_P$$

The residual R^{φ} computed by Icepak is the imbalance in "Equation 150" above summed over all the computational cells P. This is referred to as the "unscaled" residual. It may be written as

Equation 152

$$R^{\phi} = \sum_{cells P} \left| \sum_{nb} a_{nb} \phi_{nb} + b - aP\phi P \right|$$

In general, it is difficult to judge convergence by examining the residuals defined by "Equation 152" above since no scaling of is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of φ with which to compare the residual. Icepak scales the residual using a scaling factor representative of the flow rate of φ through the domain. This "scaled" residual is defined as

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equation 153

$$R^{\varphi} = \frac{\sum_{cells P} \left| \sum_{nb} a_{nb} \varphi_{nb} + b - a_P \varphi_P \right|}{\sum_{cells P} \left| a_P \varphi_P \right|}$$

For the momentum equations the denominator term $a_P \varphi_P$ is replaced by a_P , where v_P is the magnitude of the velocity at cell P.

The scaled residual is a more appropriate indicator of convergence, and is the residual displayed by Icepak.

For the continuity equation, the unscaled residual is defined as

Equation 154

$$R^{c} = \sum_{\text{cells } P} \left| \text{rate of mass creation in cell } P \right|$$

R^{c} = \sum _{{\rm cells}\ P}{|{\rm rate\ of\ mass\ creation\ in\ cell\ P}|} \label {def-resid-eq5}\mylabel {def-resid-eq5}

The scaled residual for the continuity equation is defined as

Equation 155

$$\frac{R^{c}_{iterationN}}{R^{c}_{iteration5}}$$

The denominator is the largest absolute value of the continuity residual in the first five iterations.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21 - Icepak Troubleshooting

Icepak Troubleshooting provides a list of error messages, additional context, and guidance for issues in the following areas:

- Meshing
- Boundary Conditions

Icepak Troubleshooting - Meshing

Error Message	Cause	Guidance
Unknown planes.	A 2D sheet primitive object has a plane other than xy, xz, or yz.	Contact Ansys support for assistance.
Incorrect polygon vert count.	A polygon object has less than 3 vertices, or the specified number of vertices does not match the actual number.	Check the polygon shape definition for repeated vertices. If no repeated vertices are defined, contact Ansys support for assistance.
Domain is not present, error in input file.	The air region is not correctly identified by the geometry engine.	Contact Ansys support for assistance.
External mesh "file name" cannot be reused. Please check if it is a valid mesh file.	The reuse mesh file named "file name" cannot be located or opened.	Check if the correct mesh file has been selected in the reuse mesh operation.
Polyhedron is not supported in Icepak. Reuse mesh cell type must be hex, prism, pyramid or tet.	The reuse mesh contains polyhedral cells, which are not supported by Icepak.	When meshing in Fluent, avoid using polyhedral cells.
External mesh's bounding box must match the object's box. The local coordinate system of the object might be wrong.	The reused mesh's bounding box does not match that of the geometry.	Check if the correct mesh file has been selected in the reuse mesh operation. Also, check if the reuse mesh conforms well with the geometry.
Failed to triangulate object "object name". Polygon vertices may be collapsed due to min gap settings.	The polygon shape "object name" cannot be converted into a triangulated surface for meshing.	Check the vertices of the polygon to see overlapped vertices or self-intersection. If so, correct the geometry.
CART3D/CART2D	3D/2D multi-level meshing is not	Use non-isotropic transition

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HangingNode FAILED for assembly (mesh region) "assembly name".	successful for the mesh region "assembly name".	of 3D multi-level meshing.

Icepak Troubleshooting - Boundary Conditions

Error Message	Cause	Guidance
Interior Opening "OpeningName" cannot have a Velocity specification.	Velocity is specified on an internal opening.	Switch the opening inlet type to Pressure.
Heat Transfer Coefficient specification for Stationary wall "WallName" will be ignored as it is internal to the domain.	A stationary wall with heat transfer coefficient specified is on the interior of the computational domain.	Switch the external condition to Heat Flux.
This design has power specified but does not have any wall or flow boundaries. The solution might not converge.	A design has a power specification but does not have a wall or flow boundary condition assigned.	Create an external wall boundary or appropriate flow boundaries.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22 - Scripting

Information about creating, editing, and using IronPython 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 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23 - Glossary

This section defines the terminology used in the Ansys desktop help topics. Terms are listed in alphabetical order.

A **B** **C**
D **E** **F**
G **H** **I**
J **K** **L**
M **N** **O**
P **Q** **R**
S **T** **U**
V **W** **X**
Y **Z**

Glossary: A

A/D

See Analog-to-Digital.

ACPR

Adjacent channel power ratio.

Active Substrate

A hybrid or multichip module substrate formed from a semiconductor. Termed active because components such as transistors can be fabricated directly into the substrate.

Active Trimming

The process of trimming components such as resistors while the circuit is under power. Such components are fabricated directly onto the substrate of a hybrid or multichip module, and the trimming is usually performed using a laser beam.

Active-High

A signal whose active state is considered to be a logic 1.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active-Low

A signal whose active state is considered to be a logic 0.

Actuator

A transducer that converts an electronic signal into a physical equivalent. For example, a loudspeaker is an actuator which converts electronic signals into corresponding sounds.

Adaptive Hardware

Refers to devices which allow new design variations to be "compiles" in real-time, which may be thought of as dynamically creating subroutines in hardware (see also Virtual Hardware and Cache Logic).

Additive Process

A process in which conducting material is added to specific areas of a substrate. Groups of tracks, individual tracks, or portions of tracks can be built up to precise thicknesses by iterating the process multiple times with selective masking.

Address Bus

A unidirectional set of signals used by a computer to point to memory locations in which it is interested.

Analog

A continuous value that most closely resembles the real world and can be as precise as the measuring technique allows.

Analog Circuit

A collection of components used to generate or process analog signals.

Analog-to-Digital (A/D)

The process of converting an analog value into its digital equivalent.

Anisotropic Adhesive

Special adhesives which contain minute particles of conductive material. These adhesives find particular application with the flipped-chip techniques used to mount bare die on the substrates of hybrids, multichip modules, or circuit boards. The conducting particles are only brought in contact with each other at the sites where the raised pads on the die are pressed down over their corresponding pads on the substrate, thereby forming good electrical connections between the pads.

Anti-Fuse Technology

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A programmable logic device technology in which conducting paths (anti-fuses) are grown by applying signals of relatively high voltage and current to the device's inputs.

Anti-Pad

The area of copper etched away around a via or a plated through-hole on a power or negative signal plane, thereby preventing an electrical connection being made to that plane.

Application-Specific Integrated Circuit (ASIC)

A device whose function is determined for a particular application or group of applications.

Application-Specific Standard Part (ASSP)

Refers to an integrated circuit created by a device manufacturer using ASIC technologies, and for these components to be sold as standard parts to anybody who wants to buy them.

ASIC

See Application-Specific Integrated Circuit.

ASIC Cell

A logic function in the cell library defined by the manufacturer of an application-specific integrated circuit.

Assertion-Level Logic

Special symbols which are used to more precisely indicate the function of gates with active-low inputs.

ASSP

See Application-Specific Standard Part.

Asynchronous

A signal whose data is acknowledged or acted upon immediately, irrespective of any clock signal.

Atto

Unit qualifier (symbol = a) representing one millionth of one millionth of one millionth, or 10-18. For example, 3aS stands for 3 x 10-18 seconds.

Attenuator

A passive device used to reduce signal strength while maintaining proper input and output impedance.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: B

Backplane

The medium used to interconnect a number of circuit boards. Typically refers to a special, heavy-duty printed or discrete wired circuit board.

Ball Grid Array (BGA)

A packaging technology similar to a pad grid array, in which a device's external connections are arranged as an array of conducting pads on the base of the package. However, in the case of a ball grid array, small balls of solder are attached to the conducting pads.

Bandpass

The frequency limits between half-power points of a signal or filter.

Bare Die

An unpackaged integrated circuit.

Base

Refers to the number of digits in a numbering system. For example, the decimal numbering system is said to be base-10. May also be referred to as the "radix".

Basic Cell

A predefined group of unconnected components that is replicated across the surface of a gate array.

BER

Bit error rate.

BGA

See Ball Grid Array.

BiCMOS

A technology in which the function of each logic gate is implemented using low-power CMOS, while the output stage is implemented using high-drive bipolar transistors

Binary Encoding

A form of state assignment for state machines that requires the minimum number of state variables.

BiNMOS

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A relatively new low-voltage integrated circuit technology in which complex combinations of bipolar and NMOS transistors are used to form sophisticated output stages providing both high speed and low static power dissipation.

Bi-quinary

A system which utilizes two bases, base-2 and base-5, to represent decimal numbers. Each decimal digit is represented by the sum of two parts, one of which has the value of decimal zero or five, and the other the values of zero through four. The abacus is one practical example of the use of a bi-quinary system.

Bit

Abbreviation of binary digit.

Bipolar Junction Transistor (BJT)

A family of transistors.

Blackbox

Same as N-Port.

Blind Via

A via that is only visible from one side of the substrate.

Bobble

A small circle used on the inputs to a logic gate symbol to indicate an active-low input or control, or on the outputs to indicate a negation (inversion) or a complementary signal. Some engineers prefer to use the term bubble.

Bounce Pad

A special pattern etched onto the power or negative signal plane of a microwire circuit board to be used in conjunction with a laser beam which is employed to create blind vias. The laser beam evaporates the epoxy forming the outer layers of the board and continues down to the bounce pad which reflects, or bounces, it back up, thereby terminating the via.

Braze

To unite or fuse two pieces of metal by heating, or with a hard solder with a high melting point.

Bulk Storage

Refers to some form of media, typically magnetic, such as tape or a disk drive which can be used to store large quantities of information relatively inexpensively.

Buried Via

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A via used to link conducting layers internal to a substrate. Such a via is not visible from either side of the substrate.

Bundle

A set of signals related in some way that makes it appropriate to group them together for ease of representation or manipulation. May contain both scalar and vector elements; for example, {a,b,c,d[5:0]}.

Bus

A set of signals performing a common function and carrying similar data. Typically represented using vector notation; for example, address[7:0].

Byte

A group of eight binary digits, or bits.

Glossary: C

Capacitance

A measure of the ability of two adjacent conductors separated by an insulator to hold a charge when a voltage differential is applied between them. Capacitance is measured in units of Farads.

CDMA

Code division multiple access.

Cell Library

The collective name for the set of logic functions defined by the manufacturer of an application-specific integrated circuit. The designer decides which types of cells should be realized and connected together to make the device perform its desired function.

Ceramic

An inorganic, nonmetallic material, such as alumina, beryllia, steatite, or forsterite, which is fired at a high temperature and is often used in electronics as a substrate or to create component packages.

CGA

See Column Grid Array.

Channel

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1)The area between two arrays of basic cells in a channeled gate array.

(2)The gap between the source and drain regions in a MOS transistor.

Channeled Gate Array

Application-specific integrated circuit organized as arrays of basic cells. The areas between the arrays are known as channels.

Channel-Less Gate Array

Application-specific integrated circuit organized as a single large array of basic cells. May also be referred to as a "sea of cells" or a "sea of gates" device.

Checksum

The final cyclic-redundancy-check value stored in a linear feedback shift register (or software equivalent). Also known as a "signature" in the guided-probe variant of functional test.

Chip

Popular name for an integrated circuit.

Chip-On-Board (COB)

A process in which unpackaged integrated circuits are physically and electrically attached to a circuit board, and are then encapsulated with a "glob" of protective material such as epoxy.

Chip-On-Chip (COC)

A process in which unpackaged integrated circuits are mounted on top of each other. Each die is very thin and it is possible to have over a hundred dies forming a 3D cube.

Chip-On-Flex (COF)

Similar to chip-on-board (COC), except that the unpackaged integrated circuits are attached to a flexible printed circuit.

Circuit Board

The generic name for a wide variety of interconnection techniques, which include rigid, flexible, and rigid-flex boards in single-sided, double-sided, multilayer, and discrete wired configurations.

CMOS

Logic gates constructed using both NMOS and PMOS transistors connected in a complementary manner.

Coaxial Cable

A conductor in the form of a central wire surrounded first by a dielectric (insulating) layer, and then by a conducting tube which serves to shield the central wire from external interference.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coefficient of Thermal Expansion

Defines the amount a material expands and contracts due to changes in temperature. If materials with different coefficients of thermal expansion are bonded together, changes in temperature will cause shear forces at the interface between them.

Cofired Ceramic

A substrate formed from multiple layers of "green" ceramic that are bonded together and fired at the same time.

Column Grid Array (CGA)

A packaging technology similar to a pad grid array, in which a device's external connections are arranged as an array of conducting pads on the base of the package. However, in the case of a column grid array, small columns of solder are attached to the conducting pads.

Combinatorial

A digital function whose output value is directly related to the current combination of values on its inputs. Also known as combinational.

Compiled Cell Technology

A technique used to create portions of a standard cell application-specific integrated circuit. The masks used to create components and interconnections are directly generated from Boolean representations using a silicon compiler. May also be used to create data-path functions and memory functions.

Complementary Output

Refers to a function with two outputs carrying complementary logical values. One output is referred to as the true output and the other as the complementary output.

Complex Programmable Logic Device (CPLD)

A device that contains a number of PLA or PAL functions sharing a common programmable interconnection matrix.

Component

Components are items placed on schematics and layouts to represent electrical elements and sub circuits. The component information defines a signal-processing function or source, or secondarily a data source, data channel, or similar entity. Components have pins for connections, bitmaps in the project tree, and properties for simulation. The information that constitutes a component includes its component chooser bitmap, schematic symbol, layout footprint, pin properties, parameters and parameter values, and netlist string definition. A component can be associated with more than one simulation if it can be analyzed in more than one simulator.

Computer-Generated Hologram (CGH)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to a slice of quartz or similar material into which three-dimensional patterns are cut using a laser. The angles of the patterns cut into the quartz are precisely calculated for use in the optical communication strategy known as holographic interconnect. All of these calculations are performed by a computer, and the laser used to cut the three-dimensional patterns into the quartz is also controlled by a computer. Thus, the slice of quartz is referred to a computer-generated hologram.

Conductive Ink Technology

A technique in which tracks are screen printed directly onto the surface of a circuit board using a conductive ink.

Configurable Hardware

A product whose function may be customized once or a very few times (see also Reconfigurable Hardware, Remotely Reconfigurable Hardware, Dynamically Reconfigurable Hardware, and Virtual Hardware).

Conjunction

Propositions combined with an AND operator; for example, "You have a parrot on your head AND you have a fish in your ear." The result of a conjunction is true if all the propositions comprising that conjunction are true.

Conversion Loss

The ratio in dB of the IF output of a mixer to the RF input power. All conversion loss measurements and specification are normally based on the mixer being terminated on all ports and a stated LO signal power level being applied.

Cost Function

In an optimization setup, a cost function is based on goal values specified for at least one solution quantity. Optimetrics changes the design parameter values to fulfill the cost function. In Icepak, the cost function can be based on any solution quantity that Icepak can compute, such as residuals or monitor data.

CPLD

See Complex Programmable Logic Device.

CRC

See Cyclic Redundancy Check.

CSIC

See Customer-Specific Integrated Circuit.

Customer-Specific Integrated Circuit (CSIC)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An alternative and possibly more accurate name for an ASIC, but this term is rarely used in the industry and shows little indication of finding favor with the masses.

Chemical Vapor Deposition (CVD)

A process for growing thin films on a substrate, in which a gas containing the required molecules is converted into a plasma by heating it to extremely high temperatures using microwaves. The plasma carries atoms to the surface of the substrate where they are attracted to the crystalline structure of the substrate. This underlying structure acts as a template. The new atoms continue to develop the structure to build up a layer on the substrate's surface.

Chemical Vapor Infiltration (CVI)

A process similar to chemical vapor deposition (CVD) but, in this case, the process commences by placing a crystalline powder of the required substance in a mold. Additionally, thin posts, or columns, can be pre-formed in the mold, and the powder can be deposited around them. When exposed to the same plasma as used in the CVD technique, the powder coalesces into a polycrystalline mass. After the CVI process has been performed, the posts can be dissolved leaving holes through the crystal for use in creating vias. CVI processes can produce layers twice the thickness of those obtained using CVD techniques at a fraction of the cost.

CW

Continuous wave; refers to an unmodulated sine-wave signal.

Cyclic Redundancy Check (CRC)

A calculation used to detect errors in data communications, typically performed using a linear feedback shift register. Similar calculations may be used for a variety of other purposes such as data compression.

Glossary: D

D/A

See Digital-to-Analog.

Data Bus

A bi-directional set of signals used by a computer to convey information from a memory location to the central processing unit and vice versa. More generally, a set of signals used to convey data between digital functions.

Data-Path Function

A well-defined function such as an adder, counter, or multiplier used to process digital data.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dBm

Power level relative to 1mV rms.

DC Balance

Stream of data encoded to ensure an equal balance of 1 or 0 . 8b10b encoding has been developed to ensure DC balancing.

DC Coupling

A method of coupling two different circuits together, allowing them to share both the static DC and varying AC characteristics of a signal.

Decoder (digital)

A logic function that uses a binary value, or address, to select between a number of outputs and to assert the selected output by placing it in its active state.

Deep Sub-Micron

Typically taken to refer to integrated circuits containing structures which are smaller than 0.5 microns.

DeMorgan Transformation

The transformation of a Boolean expression into an alternate, and often more convenient, form.

Design

Designs are the building blocks of projects, and can be Circuit designs or 3D models. Designs consist of schematics or geometrical models, model data, solution setup information, output graphs and tables, and other pieces of information that go into describing simulation of electrical circuits. A design is the largest single simulatable entity in a project.

Design Variation

A single combination of variable values that is solved during a parametric or optimization setup.

Device

A discrete, separate electrical entity such as a diode, a capacitor or a packaged transistor.

Die

(1)An unpackaged integrated circuit. In this case, the plural of die is also die (in much the same way that "a shoal of herring" is the plural of "herring"). (2)A piece of metal with a design engraved or embossed on it for stamping onto another material, upon which the design appears in relief.

Die Separation

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The process of separating individual die from the wafer by marking the wafer with a diamond scribe and fracturing it along the scribed lines.

Die Stacking

A technique used in specialist applications in which several bare die are stacked on top of each other to form a sandwich. The die are connected together and then packaged as a single entity.

Dielectric Layer

(1) An insulating layer used to separate two signal layers.

(2) An insulating layer used to modify the electrical characteristics of an MCM-D substrate.

Diffusion Layer

The surface layer of a piece of semiconductor into which impurities are diffused to form P-type and N-type material. In addition to forming components, the diffusion layer may also be used to create embedded traces.

Digital

A value represented as being in one of a finite number of discrete states called quanta. The accuracy of a digital value is dependent on the number of quanta used to represent it.

Digital Circuit

A collection of logic gates used to process or generate digital signals.

Digital Signal Processor (DSP)

A primarily digital component used to process either digital or analog signals. In the case of the latter, the signal may first be conditioned, then converted into a digital equivalent using an analog-to-digital (A/D) converter function. The signal conditioning and A/D functions may either be external to the DSP or resident in the device. A typical DSP application might be the compression/decompression of video data.

Digital-to-Analog (D/A)

The process of converting a digital value into its analog equivalent.

Diode

A two-terminal device that only conducts electricity in one direction; in the other direction it behaves like an open switch. The term diode is typically taken to refer to a semiconductor device, although alternative implementations such as vacuum tubes are available.

Diode-Transistor Logic (DTL)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Logic gates implemented using particular configurations of diodes and bipolar junction transistors. For the majority of today's designers, diode-transistor logic is of historical interest only.

Discrete Device

Typically taken to refer to an electronic component such as a resistor, capacitor, diode, or transistor that is presented in an individual package. More rarely, the term may be used in connection with a simple integrated circuit containing a small number of primitive gates.

Discrete Wire Board (DWB)

A form of circuit board in which a special computer-controlled wiring machine ultrasonically bonds extremely fine insulated wires into the surface layer of the board. This discipline has enjoyed only limited recognition, but may be poised to emerge as the technology-of-choice for high-speed designers.

Discrete Wire Technology

The technology used to fabricate discrete wire boards.

Doping

The process of inserting selected impurities into a semiconductor to create P-type or N-type material.

Double-Sided

A printed circuit board with tracks on both sides

DPSK

Differential phase-shift keying.

DQPSK

Differential quadrature phase-shift keying.

DRAM

See Dynamic RAM.

DSP

See Digital Signal Processor.

DTL

See Diode-Transistor Logic.

DUT

Device under test.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Flex

A type of flexible printed circuit which is used in applications that are required to undergo constant flexing such as ribbon cables in printers.

Dynamic RAM (DRAM)

A memory device in which each cell is formed from a transistor-capacitor pair. Called dynamic because the capacitor loses its charge over time, and each cell must be periodically recharged if it is to retain its data.

Glossary: E

E2PROM

See electrically Erasable Programmable Read-Only Memory.

EBE

See Electron Beam Epitaxy.

ECL

See Emitter-Coupled Logic.

Edge port

A place in a layout or footprint geometry through which excitation signals enter and leave the structure.

Edge-Sensitive

An input that only affects a function when it transitions from one logic value to another.

EEPROM

See electrically Erasable Programmable Read-Only Memory.

electrically Erasable Programmable Read-Only Memory (EEPROM or E2PROM)

A memory device whose contents can be electrically programmed by the designer. Additionally, the contents can be electrically erased allowing the device to be reprogrammed. Also known as EEPROM and E2PROM.

Electromigration

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1)A process in which structures on an integrated circuit's substrate are eroded by the flow of electrons in much the same way as land is eroded by a river (also known as subatomic erosion). (2)The process of forming transistor-like regions in a semiconductor using an intense magnetic field.

Electron Beam Epitaxy (EBE)

A technique for creating thin films on substrates in precise patterns, in which the substrate is first coated with a layer of dopant material before being placed in a high vacuum. A guided beam of electrons is fired at the substrate causing the dopant to be driven into it, effectively allowing molecular-thin layers to be "painted" onto the substrate where required.

Electron Beam Lithography

An integrated circuit fabrication process in which fine beams of electrons are used to draw extremely high-resolution patterns directly into the resist without the use of a mask.

Electro-Static Discharge (ESD)

The process of moving around can generate static electricity. The term electro-static discharge refers to a charged person, or object, discharging static electricity. Although the current associated with such a static charge is low, the electric potential can be in the millions of volts and can severely damage electronic components. CMOS devices are particularly prone to damage from static electricity.

EM

Electromagnetic.

Emitter-Coupled Logic (ECL)

Logic gates implemented using particular configurations of bipolar junction transistors.

Enzyme

One of numerous complex proteins which are produced by living cells and catalyze biochemical reactions at body temperatures.

EPROM

See Erasable Programmable Read-Only Memory.

Equivalent Gate

A concept in which each type of logic function is assigned an equivalent gate value for the purposes of comparing functions and devices. However, the definition of an equivalent gate varies depending on who you're talking to.

Equivalent Integrated Circuit

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A concept used to compare the component density supported by diverse interconnection technologies such as circuit boards, hybrids, and multichip modules

Erasable Programmable Read-Only Memory (EPROM)

A memory device whose contents can be electrically programmed by the designer. Additionally, the contents can be erased by exposing the die to ultraviolet light through a quartz window mounted in the top of the component's package.

ESD

See Electro-Static Discharge.

Etching

The process of selectively removing any material not protected by a resist using an appropriate solvent or acid. In some cases the unwanted material is removed using an electrolytic process.

Eutectic Bond

A bond formed when two pieces of metal, or metal-coated materials, are pressed together and vibrated at ultrasonic frequencies.

Euler Angles

Euler angles are used in Ansoft software to carry out a coordinate transformation from one coordinate system to another. The Swiss mathematician and physicist Leonhard Euler first developed the classical rotation theorem to describe rotations in 3D space. The angles used are Euler angles and can be used to describe any 3D rotation. These angles, given by (ö, è, ø) represent a series of sequential rotations about two axis of the coordinate system. The first rotation (ö) represents a rotation about the Z-axis of the source coordinate system (X, Y, Z) which results in an intermediate coordinate system denoted by (X'', Y'', Z''). The second rotation (è) represents a rotation of the intermediate coordinate system about the X''-axis, again resulting in an intermediate coordinate system denoted by (X', Y', Z'). The third and final rotation (ø) represents a rotation about the Z'-axis of the intermediate coordinate system. The final rotation completes the rotation and results in the "target" coordinate system denoted (X, Y, Z).

For further information see, Eric W. Weisstein, "Euler Angles." From *MathWorld* - A Wolfram Web Resource.

http://mathworld.wolfram.com/EulerAngles.html .

Eye Diagram

Eye diagrams are commonly used to analyze signal integrity issues with communications channels. The bits are superimposed at unit intervals representing the duration of each bit.

Eye Mask

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The size of the eye opening in the center of an eye diagram indicates the amount of voltage and timing margin available to sample this signal. Thus, for a particular electrical interface, a fixed reticule or window could be placed over the eye diagram showing how the actual signal compares to minimum criteria window, know as the eye mask. If a margin rectangle with width equal to the required timing margin and height equal to the required voltage margin fits into the opening, then the signal has adequate margins. Voltage margin can often be traded off for timing margin.

Glossary: F

Fan-Out Via

In the case of surface mount devices attached to double-sided or multilayer boards, each component pad is usually connected by a short length of track to a via which forms a link to other conducting layers, and this via is known as a fan-out via. The term fan-out via is generally also taken to include any vias that fall inside the device's footprint (under the body of the device). Some designers attempt to differentiate these vias from those that fall outside the device's footprint by referring to them as fan-in vias, but this is not an industry-standard term.

Falling-Edge

A transition from a logic 1 to a logic 0. Also known as a negative edge.

Falltime

The time it takes for a waveform to transition from the high logic state to the low logic state. Falltime is usually measured from 90% of the total signal swing to 10% of the signal swing.

Fan-In and Fan-Out Vias

In the case of surface mount devices attached to double-sided or multilayer boards, each component pad is usually connected by a short length of track to a via which forms a link to other conducting layers, and this via is known as a fan-out via. The term fan-out via is generally also taken to include any vias that fall inside the device’s footprint (under the body of the device). Some designers attempt to differentiate these vias from those that fall outside the device’s footprint by referring to them as fan-in vias, but this is not an industry-standard term.

Femto

Unit qualifier (symbol = f) representing one thousandth of one millionth of one millionth, or 10-15. For example, 3fS stands for 3 x 10-15 seconds.

FET

See Field-Effect Transistor.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field-Effect Transistor (FET)

A transistor whose control, or gate, signal creates an electromagnetic field which turns the transistor ON or OFF.

Field-Programmable Gate Array (FPGA)

A programmable logic device which is more versatile than traditional programmable devices such as PALs and PLAs, but less versatile than an application-specific integrated circuit. Some field-programmable gate arrays use fuses such as those found in programmable logic devices, but others are based on SRAM equivalents.

Field-Programmable Interconnect Chip (FPIC)

An alternative, proprietary name for a field-programmable interconnect device (FPID).

Field-Programmable Interconnect Device (FPID)

A device which is used to connect logic devices together, and which can be dynamically reconfigured in the same way as standard SRAM-based FPGAs. Because each FPID may have around 1,000 pins, only a few such devices are typically required on a circuit board.

Filter

Filters are used to block out undesired frequencies. There are two types of filters: band pass and rejection. A band pass filter permits only the desired range to pass through, while the rejection filter attenuates an undesired range of frequencies.

Firmware

Refers to programs, or sequences of instructions, that are hard-coded into non-volatile memory devices.

First-In First Out (FIFO)

A memory device in which data is read out in the same order that it was written in.

Flash (e.g., Gold Flash)

An extremely thin layer of gold with a thickness measured on the molecular level which is either electroplated or chemically plated onto a surface.

FLASH Memory

An evolutionary technology that combines the best features of the EPROM and E2PROM technologies. The name FLASH is derived from the technology's fast reprogramming time compared to EPROM.

Flex

See Flexible Printed Circuit.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flexible Printed Circuit (FPC or Flex)

A specialist circuit board technology, often abbreviated to "flex", in which tracks are printed onto flexible materials. There are a number of flavors of flex, including static flex, dynamic flex, and rigid flex.

Flipped Chip

A generic name for processes in which unpackaged integrated circuits are mounted directly onto a substrate with their component-sides facing the substrate.

Flipped TAB

A combination of flipped chip and tape automated bonding.

Footprint

The area occupied by a device mounted on a substrate.

Fourier Analysis

A mathematical procedure used to determine the collection of sine waves (differing in frequency and amplitude) that is necessary to make up the square-wave pattern under consideration.

FPC

See Flexible Printed Circuit.

FPGA

See Field-Programmable Gate Array.

FPIC

See Field-Programmable Interconnect Chip.

FPID

See Field-Programmable Interconnect Device.

FR4

The most commonly used insulating base material for circuit boards. FR4 is made from woven glass fibers which are bonded together with an epoxy. The board is cured using a combination of temperature and pressure which causes the glass fibers to melt and bond together, thereby giving the board strength and rigidity. The first two characters stand for "Flame Retardant". FR4 is technically a form of fiberglass, and some people do refer to these composites as fiberglass boards or fiberglass substrates, but not often.

Free-Space Optical Interconnect

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of optical interconnect in which laser-diode transmitters communicate directly with photo-transistor receivers without employing optical fibers or optical waveguides.

Full Custom

An application-specific integrated circuit in which the designer has complete control over every mask layer used to fabricate the device. The manufacturer does not provide a cell library or pre-fabricate any components on the substrate.

Functional Latency

Refers to the fact that, at any given time, only a portion of the logic functions in a device or system are typically active (doing anything useful).

Functional Test

A test strategy in which signals are applied to a circuit's inputs, and the resulting signals which are observed on the circuit's outputs are compared to known good values.

Fuse

See Fusible-Link Technology.

Fusible-Link Technology (Fuse)

A programmable logic device technology which employs links called fuses. Individual fuses can be removed by applying pulses of relatively high voltage and current to the device's inputs.

Fuzz-Button

A small ball of fibrous gold used in one technique for attaching components such as multichip modules to circuit boards. Fuzz-buttons are inserted between the pads on the base of the package and their corresponding pads on the board. When the package is forced against the board, the fuzz-buttons compress to form good electrical connections. Even when the pressure is removed, the fuzz-buttons act in a similar manner to Velcro and continue to hold the component in place. One of the main advantages of the fuzz-button approach is that it allows broken devices to be quickly removed and replaced. Even though fuzz-button technology would appear to be inherently unreliable, it is used in such devices as missiles, so one can only assume that it is fairly robust.

Glossary: G

Gain

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The ratio of the power output to the power input of the amplifier in dB. The gain is specified in the linear operating range of the amplifier where a 1 dB increase in input power gives rise to a 1 dB increase in output power. Gain = 20*log(S21)

Gate Array

An application-specific integrated circuit in which the manufacturer pre-fabricates devices containing arrays of unconnected components organized in groups called basic cells. The designer specifies the function of the device in terms of cells from the cell library and the connections between them, and the manufacturer then generates the masks used to create the metallization layers.

Glue Logic

Simple logic gates used to interface more complex functions together.

Gold Flash

An extremely thin layer of gold with a thickness measured on the molecular level which is either electroplated or chemically plated onto a surface.

Goal

In an optimization setup, a goal is the value of a solution quantity that you want to be achieved during the optimization. A goal is represented as one row in the cost function table. Each cost function defined in an optimization setup must include at least one goal.

Gray Code

A sequence of binary values in which each pair of adjacent values differs by only a single bit; for example, 00, 01, 11, 10.

Green Ceramic

Unfired, malleable ceramic.

Ground Bounce

Momentary noise on the device negative signal plane causing a 0 signal to erroneously be seen as a 1. Ground bounce is caused by simultaneously switching outputs (SSO).

Guard Condition

A Boolean expression associated with a state transition in a state diagram or state table. The expression must be satisfied for that state transition to be executed.

Guided Probe

A form of functional test in which the operator is guided in the probing of a circuit to isolate a faulty component or track.

Guided-Wave

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of optical interconnect, in which optical waveguides are fabricated directly on the substrate of a multichip module. These waveguides can be created using variations on standard opto-lithographic thin-film processes.

Glossary: H

Hard Macro (Macro Cell)

A logic function defined by the manufacturer of an application-specific integrated circuit. The function is described in terms of the simple functions provided in the cell library and the connections between them. The manufacturer also defines how the cells forming the macro will be assigned to basic cells and the routing of tracks between the basic cells.

Hardware

Generally understood to refer to any of the physical portions constituting an electronic system, including components, circuit boards, power supplies, cabinets, and monitors.

Harmonic

Integer multiples of the fundamental frequency of interest commonly produced by a non-linear amplifier.

Harmonic Balance

A frequency domain analysis technique for simulating nonlinear circuits and systems. This method assumes the input stimulus consists of a relatively few steady state sinusoids. Therefore the solution can be expressed as a sum of steady state sinusoids that includes the input frequencies in addition to any significant harmonics or mixing terms. A circuit with a single input source will require a single tone HB simulation. The harmonic balance simulation is ideal for situations where transient simulation methods are problematic, such as:

- Components modeled in frequency domain, for instance (dispersive) transmission lines
- Circuit time constants large compared to period of simulation frequency
- Circuits with lots of reactive components

Harmonic balance methods, therefore, are the best choice for most microwave circuits excited with sinusoidal signals (e.g., mixers, power amplifiers).

Harmonic Tuning

Impedance-matching at the harmonic frequencies for enhanced performance and efficiency.

Hertz (Hz)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit of frequency. One Hertz equals one cycle, or one oscillation, per second.

Heterojunction

The interface between two regions of dissimilar semiconductor materials. The interface of a hetrojunction has naturally occurring electric fields which can be used to accelerate electrons, and transistors created using hetrojunctions can switch much faster than their counterparts of the same size.

Hexadecimal

Base-16 numbering system. Each hexadecimal digit can be directly mapped onto four binary digits, or bits.

High Impedance State

The state on a signal that is not being driven by any value. A high-impedance state is indicated by the character Z.

Holographic Interconnect

A form of optical interconnect based on a thin slice of quartz, into which three-dimensional images are cut using a laser beam. Thus, the quartz is referred to as a computer-generated hologram, and this interconnection strategy is referred to as holographic.

Homojunction

An interface between two regions of semiconductor having the same basic composition but opposing types of doping. Homojunctions dominate current processes because they are easier to fabricate than heterojunctions.

Hybrid

An electronic sub-system in which a number of integrated circuits (packaged and/or unpackaged) and discrete components are attached directly to a common substrate. Connections between the components are formed on the surface of the substrate, and some components such as resistors and inductors may be fabricated directly onto the substrate.

Hydrogen Bond

The electrons in a water molecule are not distributed equally, because the oxygen atom is a bigger, more robust fellow which grabs more than its fair share. The end result is that the oxygen atom has an overall negative charge, while the two hydrogen atoms are left feeling somewhat on the positive side. This unequal distribution of charge means that the hydrogen atoms are attracted to anything with a negative bias; for example, the oxygen atom of another water molecule. The resulting bond is known as a hydrogen bond.

Hz

See Hertz.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: I

IBIS

IBIS (I/O Buffer Information Specification) is a standard for electronic behavioral specifications of integrated circuit input/output analog characteristics. The core of an IBIS model is a table of current versus voltage and I/O switching timing information. Xilinx IBIS models contain tables for typical, slow/MIN (weak transistors, high temperature, low voltage) and fast/MAX (strong transistors, low temperature, high voltage) process corners. IBIS models are derived from SPICE simulation results and/or lab measurements. The benefit for IBIS model user is fast and accurate simulation while preserving IC vendors intellectual property (information about circuit and process details).

IC

See Integrated Circuit.

ICR

See In-Circuit Reconfigurable.

Impedance

The resistance to the flow of current caused by resistive, capacitive, or inductive devices (or undesired elements) in a circuit.

Impedance Matching

Function of ensuring that the impedance of the transmitter, the receiver, and the transmission line are identical. Mismatched impedances could result in signal reflections, ringing, overshoot, undershoot, and stairstep waveforms.

Incident Voltage

The user specified voltage at an input.

In-Circuit Reconfigurable (ICR)

An SRAM-based, or similar component which can be dynamically reprogrammed on-the-fly while remaining resident in the system.

Inductance

A property of a conductor that allows is to store energy in a magnetic field which is induced by a current flowing through it. Inductance is measured in units of Henries (the base unit is a Henry).

Insertion Loss

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Insertion Loss (dB) is defined as the drop in power as a signal enters an RF component. This value not only includes the reflected incoming signal, but also the attenuation of the component.

In-System Programmable (ISP)

An E2-based, FLASH-based, or similar component which can be reprogrammed while remaining resident on the circuit board.

Integrated Circuit (IC)

A device in which components such as resistors, capacitors, diodes, and transistors are formed on the surface of a single piece of semiconductor.

Ion

A particle formed when an electron is added to, or subtracted from, a neutral atom or group of atoms.

Ion Implantation

A process in which beams of ions are directed at a semiconductor to alter its type and conductivity in certain regions.

Isolation

The ratio (expressed in dB) of the power level at one port compared to the resulting power level of the output port.

ISP

See In-System Programmable.

Glossary: J

JEDEC

See Joint Electronic Device Engineering Council.

Jitter

The jitter of a periodic signal is the delay between the expected transition of the signal and the actual transition. Jitter is a zero mean random variable. When worst case analysis is undertaken the maximum value of this random variable is used.

Jitter Tolerance

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jitter tolerance is defined as the peak-to-peak amplitude of sinusoidal jitter applied on the input that causes a predefined, acceptable loss at the output. For example jitter applied to the input of an OC-N equipment interface that causes an equivalent 1dB optical power penalty.

Jitter Transfer

Jitter transfer is defined as the ratio of jitter on the output of a device to the jitter applied on the input of the device, versus frequency. Jitter transfer is important in applications where the system is utilized in a loop-timed mode, where the recovered clock is used as the source of the transmit clock.

Joint Electronic Device Engineering Council (JEDEC)

A council which creates, approves, arbitrates, and oversees industry standards for electronic devices. In programmable logic, the term JEDEC refers to a textual file containing information used to program a device. The file format is a JEDEC approved standard and is commonly referred to as a JEDEC file.

Jumper

A small piece of wire used to link two tracks on a circuit board.

Glossary: K

Karnaugh Map

A graphical technique for representing a logical function. Karnaugh maps are often useful for the purposes of minimization.

Kelvin Scale of Temperature

A scale of temperature which was invented by the British mathematician and physicist William Thomas, first Baron of Kelvin. Under the Kelvin, or absolute, scale of temperature, 0 K (corresponding to -273oC) is the coldest possible temperature and is known as absolute zero.

Kilo

Unit qualifier (symbol = K) representing one thousand, or 103. For example, 3KHz stands for 3 x 103 Hertz.

Kirchhoff's current law

The sum of all currents entering a node is equal to the sum of all currents leaving the node.

Kirchhoff's voltage law

The directed sum of the electrical potential differences around a circuit must be zero.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: L

Laminate

A material constructed from thin layers or sheets. Often used in the substrate of circuit boards.

Large-Scale Integration (LSI)

Refers to the number of logic gates in a device. By one convention, large-scale integration represents a device containing 100 to 999 gates.

Laser Diode

A special semiconductor diode which emits a beam of coherent light.

Last-In First-Out (LIFO)

A memory device in which data is read out in the reverse order to which it was written in.

Latch-Up Condition

A condition in which a circuit draws uncontrolled amounts of current, and certain voltages are forced, or "latched-up", to some level. Particularly relevant in the case of CMOS devices which can latch-up if their operating conditions are violated.

Lateral Thermal Conductivity

Good lateral thermal conductivity means that the heat generated by components mounted on a substrate can be conducted horizontally across the substrate and out through its leads.

Layers (and Stackup)

Layers are used in the layout editor to organize and isolate sets of geometry or other visual indicators. Signal, Negative Signal, and Dielectric are common physical layers, while Symbol (to show component symbols in layout), Error, and Ratsnest (to show connectivity) are non-physical layers. The stackup contains additional properties of the physical layers, such as material, thickness, and elevation. Geometrical information on these layers is used to generate masks for manufacturing.

LDMOS

Laterally diffused metal oxide semiconductor.

Lead

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1) A metallic element (chemical symbol Pb).

(2) A metal conductor used to provide a connection from the inside of a device package to the outside world for soldering or other mounting techniques. Leads are also commonly called pins.

Lead Frame

A metallic frame containing leads and a base to which an unpackaged integrated circuit is attached. After encapsulation, the outer part of the frame is cut away and the leads are bent into the required shapes.

Lead Through-Hole (LTH)

A technique for populating circuit boards in which component leads are inserted into plated through-holes. Often abbreviated to "through-hole" or "thru-hole". When all of the components have been inserted, they are soldered to the board, usually using a wave soldering technique.

Level-Sensitive

An input whose effect on a function depends only on its current logic value or level, and is not directly related to it transitioning from one logic value to another.

LFSR

See Linear Feedback Shift Register.

Library

A library is a collection of one or more components or component dependencies (materials, symbols, footprints, or padstacks) stored in a container file. A library must be configured to a circuit before use, either by the user (manually) or by loading technology files (automatically). User libraries and Personal libraries are used to add foundry support, user defined models, and any custom set of components or simulation models. See the Library Overview topic for more information.

LIFO

See Last-In First-Out.

Limiting Level

The input power level when the output power is goes into compression and no longer becomes linear.

Line

Used to refer to the width of a track; for example, "This track has a line-width of 0.12mm."

Linear Feedback Shift Register (LFSR)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A shift register whose data input is generated as an XOR or XNOR of two or more elements in the register chain.

Linearity

Describes how closely an output signal is to a perfectly scaled multiple of a corresponding input signal.

Load Pull

Automated measurement of RF performance of a device under test (at a constant frequency) by varying the source and load impedance presented to the device

Logic Function

A mathematical function that performs a digital operation on digital data and returns a digital value.

Logic Gate

The physical implementation of a logic function.

Logic Synthesis

A process in which a program is used to optimize the logic used to implement a design.

Low-Fired Cofired

Similar in principle to standard cofired ceramic substrate techniques. However, low-fired cofired uses modern ceramic materials with compositions that allow them to be fired at temperatures as low as 650oC to 750oC. Firing at these temperatures in an inert atmosphere such as nitrogen allows non-refractory metals such as copper to be used to create tracks.

LSI

See Large-Scale Integration.

LTH

See Lead Through-Hole.

Glossary: M

Mask Programmable

A device such as a read-only memory which is programmed during its construction using a unique set of masks.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximal Displacement

A linear feedback shift register whose taps are selected such that changing a single bit in the input data stream will cause the maximum possible disruption to the register's contents.

Maximal Length

A linear feedback shift register that sequences through (2n - 1) states before returning to its original value.

Maxterm

The logical OR of the inverted variables associated with an input combination to a logical function.

MBE

See Molecular Beam Epitaxy.

MCM

See Multichip Module.

Medium-Scale Integration (MSI)

Refers to the number of logic gates in a device. By one convention, medium-scale integration represents a device containing 13 to 99 gates.

Meg

Unit qualifier (symbol = M) representing one million, or 106. For example, 3MHz stands for 3 x 106 Hertz.

Metallization Layer

A layer of conducting material on an integrated circuit that is selectively deposited or etched to form connections between logic gates. There may be several metallization layers separated by dielectric (insulating) layers.

Metal-Oxide Semiconductor (MOS)

A family of transistors where the controlling terminal is connected to a plate that is separated from the semiconduxtor by an insulating layer. This plate was originally made out metal (we now use polysilicon, or poly) and the insulator is an oxide -- hence the "metal-oxide" appellation.

Meta-Stable

A condition where the outputs of a logic function are oscillating uncontrollably between undefined values.

Micro

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit qualifier (symbol = u) representing one millionth, or 10-6. For example, 3uS stands for 3 x 10-6 Seconds.

Microwave

The range in the electromagnetic spectrum from 300 MHz to 30 GHz (with corresponding wavelengths from 100 cm to 1 cm).

Microwire

A trade name for one incarnation of discrete wire technology. Microwire augments the main attributes of multiwire with laser-drilled blind vias, allowing these boards to support the maximum number of tracks and components.

Millman's method

The voltage on the ends of branches in parallel is equal to the sum of the currents flowing in every branch divided by the total equivalent conductance.

Minterm

The logical AND of the variables associated with an input combination to a logical function.

MMIC

Monolithic Microwave Integrated Circuit

Mod or Modulus

Refers to the number of states that a function such as a counter will pass through before returning to its original value. For example, a function that counts from 00002 to 11112 has a modulus of 16 and would be called a modulo-16 or mod-16 counter.

Molecular Beam Epitaxy (MBE)

A technique for creating thin films on substrates in precise patterns, in which the substrate is placed in a high vacuum, and a guided beam of ionized molecules is fired at it, effectively allowing molecular-thin layers to be "painted" onto the substrate where required.

MOS

See Metal-Oxide Semiconductor.

MOSFET

Metal-oxide semiconductor field-effect transistor.

MSI

See Medium-Scale Integration.

Multichip Module (MCM)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A generic name for a group of advanced interconnection and packaging technologies featuring unpackaged integrated circuits mounted directly onto a common substrate.

Multilayer

A printed circuit board constructed from a number of very thin single-sided and/or double-sided boards which are bonded together using a combination of temperature and pressure.

Multiplexer (digital)

A logic function that uses a binary value, or address, to select between a number of inputs and conveys the data from the selected input to the output.

Multiwire

A trade name for one incarnation of discrete wire technology.

Multizone

A stackup that contains zones or areas, each of which contains a subset of the layers in the stackup.

Mutual Capacitance

The capacitance between two conductors (one considered aggressor, the other victim) when all other conductors are connected together and then regarded as an ignored ground. It describes the amount of coupling due to the electric field. The mutual capacitance will inject an often undesired current into the victim line proportional to the rate of change of voltage on the aggressor line. Mutual Capacitance it a cause of crosstalk.

Mutual Inductance

The inductance between two conductors (one considered aggressor, the other victim) placed close enough that the magnetic field induced by a current flowing into the aggressor line encompasses the victim. The mutual inductance will inject an often undesired voltage noise onto the victim proportional to the rate of change of the current on the aggressor line.

Glossary: N

Nano

Unit qualifier (symbol = n) representing one thousandth of one millionth, or 10^{-9}. For example, 3nS stands for 3 x 10^{-9} Seconds.

Nanobot

A molecular-sized robot (see Nanotechnology below)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nanophase Materials

A form of matter which was only recently discovered, in which small clusters of atoms form the building blocks of a larger structure. These structures differ from those of naturally occurring crystals, in which individual atoms arrange themselves into a lattice.

Nanotechnology

Nanotechnology is an elusive term that is used by different research-and-development teams to refer to whatever it is that they're working on at the time. However, irrespective of their particular area of interest, nanotechnology always refers to something extremely small. One of the more exciting branches of nanotechnology that has been suggested as having potential in the future is that of micro-miniature electronic products that assemble themselves.

N-channel MOS (NMOS)

Refers to the order in which the semiconductor is doped in a MOS device. That is, which structures are constructed as N-type versus P-type material.

Negative-Edge

A transition from a logic 1 to a logic 0. Also known as a falling edge.

Negative Ion

An atom or group of atoms with an extra electron.

Negative Logic

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more negative potential is considered to represent TRUE and the more positive potential is considered to represent FALSE. Also known as negative true logic.

Negative Resist

A process where ultraviolet radiation passing through the transparent areas of a mask causes the resist to be cured. The uncured areas are then removed using an appropriate solvent.

Negative Signal Plane

A conducting layer in, or on, a substrate providing a grounding, or reference, point for components. There may be several negative signal planes separated by insulating layers.

Negative-True

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more negative potential is considered to represent TRUE and the more positive potential is considered to represent FALSE. Also known as negative logic.

Nibble

See Nybble.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NMOS

See N-channel MOS.

Noise Figure / Noise Factor

The Noise Factor of a transducer at a specified input frequency is the ratio of (a/b) where “a and b” are:

(a) the available Signal to Noise Ratio (SNR) at the signal generator terminals per unit bandwidth when the temperature of the input termination (generator or source) is 290 K and the bandwidth is limited by the transducer, to

(b) the available SNR per unit bandwidth at the output terminals of the transducer.

Traditionally:

Noise Figure NF = 10 log(noise factor F)

Noise Temperature (T_e) = T_o(F - 1)

Where:

- T_e is the noise temperature
- T_o is standard temperature 290 K
- F is noise factor

Noise Floor

This is defined as the lowest possible input to a chain or a component, that will produce a detectable output.

Noise Temperature

This is the amount of thermal noise in a chain or a component. Noise Factor and Noise Temperature (T_e) are related as follows:

Noise Temperature (T_e) = (F - 1)T_o

Where:

- T_e is the noise temperature
- T_o is standard temperature 290 K
- F is noise factor

For example, a noise figure of 2.0 dB is equivalent to a Noise Temperature of 170 K.

Nominal Design

The original model on which Optimetrics analyses are based.

Non-Volatile

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A memory device which does not lose its data when power is removed from the system.

Non-Volatile RAM

A device which is generally formed from an SRAM die mounted in a package with a very small battery, or as a mixture of SRAM and EEPROM cells fabricated on the same die.

Norton's theorem

Any two-terminal collection of voltage sources and resistors is electrically equivalent to an ideal current source in parallel with a single resistor.

NPN (N-type - P-type - N-type)

Refers to the order in which the semiconductor is doped in a bipolar junction transistor.

N-Port

An N-port component is typically characterized by network parameter data contained in the N-port itself in spreadsheet form or (more usually) by network parameter data contained in an external file.

N-type

A piece of semiconductor doped with impurities that make it amenable to donating electrons.

Nybble

A group of four binary digits, or bits (also called a nibble).

Glossary: O

Octal

Base-8 numbering system. Each octal digit can be directly mapped onto three binary digits, or bits.

Ohm

Unit of resistance. The Greek letter omega, *, is often used to represent ohms; for example, 1M* indicates one million ohms.

Ohm's law

The voltage across a resistor is the product of its resistance and the current flowing through it.

One-Hot Encoding

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of state assignment for state machines in which each state is represented by an individual state variable.

One-Time Programmable

A device such as a PAL, PLA, or PROM that can only be programmed a single time and whose contents cannot be subsequently erased.

Optical Interconnect

The generic name for interconnection strategies based on opto-electronic systems, including fiber-optics, free-space, guided-wave, and holographic techniques.

Optical Lithography

A process in which radiation at optical wavelengths (usually in the ultraviolet range) is passed through a mask, and the resulting patterns are projected onto a layer of resist coating the substrate material.

Optical Mask

A sheet of material carrying patterns that are either transparent or opaque to the wavelengths used in an optical-lithographic process. Such a mask can carry hundreds of thousands of fine lines and geometric shapes.

Opto-Electronic

Refers to a system which combines optical and electronic components.

Organic Resist

A material which is used to coat a substrate and is then selectively cured to form an impervious layer. These materials are called organic because they are based on carbon compounds as are living creatures.

Organic Solvent

A solvent for organic materials such as those used to form organic resists.

Organic Substrate

Substrate materials such as FR4, in which woven glass fibers are bonded together with an epoxy. These materials are called organic because epoxies are based on carbon compounds as are living creatures.

Overglassing

One of the final stages in the integrated circuit fabrication process in which the entire surface of the wafer is coated with a layer of silicon dioxide or silicon nitride. This layer may also be referred to as the barrier layer or the passivation layer. An additional lithographic step is required to pattern holes in this layer to allow connections to be made to the pads.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: P

Package

Leaded assembly (inside of which one or more dies are mounted and connected) for use in larger circuits.

Pad

An area of metallization on a substrate used for probing or to connect to a via, plated through-hole, or an external interconnect.

Pad Grid Array (PGA)

A packaging technology in which a device's external connections are arranged as an array of conducting pads on the base of the package.

Padstack

Refers to any pads, anti-pads, and thermal relief pads associated with a via or a plated through-hole as it passes through the layers forming the substrate.

Padcap

A special flavor of circuit board used for high-reliability military applications. Distinguished by the fact that the outer surfaces of the board have pads but no tracks. Signal layers are only created on the inner planes, and tracks are connected to the surface pads by vias.

Parallel-In Serial-Out (PISO)

Refers to a shift register in which the data is loaded in parallel and read out serially.

Parasitic Effects

The effects caused by undesired resistance, capacitance, or inductance inherent in the material or topology of a track or component.

Passive Trimming

A process in which a laser beam is used to trim components such as thick-film and thin-film resistors on an otherwise unpopulated and unpowered hybrid or multichip module substrate. Probes are placed at each end of a component to monitor its value while the laser evaporates some of the material forming the component.

Pass-Transistor Logic

A technique for connecting MOS transistors such that data signals pass between their source and drain terminals. Pass-transistor logic minimizes the number of transistors required to

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

implement a function, and is typically employed by designers of cell libraries or full-custom integrated circuits.

PGA

See either Pad Grid Array or Pin Grid Array.

Photo-Transistor

A special transistor which converts an optical input in the form of light into an equivalent electronic signal in the form of a voltage or current.

Pico

Unit qualifier (symbol = p) representing one millionth of one millionth, or 10^{-12}. For example, 3pS stands for 3 x 10^{-12} Seconds.

PIN Diode

A diode where a thin layer exists between the N and P regions. Rectification with pin diodes is limited. They actually behave more like a variable resistor that changes based upon the DC bias.

Pin Grid Array (PGA)

A packaging technology in which a device's external connections are arranged as an array of conducting leads, or pins, on the base of the package.

PISO

See Parallel-In Serial-Out.

Place-Value

Refers to a numbering system in which the value of a particular digit depends both on the digit itself and its position in the number.

PlasmA gaseous state in which the atoms or molecules are dissociated to form ions.

Plated Through-Hole (PTH)

(1) A hole in a double-sided or multilayer board that is used to accommodate a through-hole component lead and is plated with copper.

(2) An alternative name for the lead through-hole technique for populating circuit boards in which component leads are inserted into plated through-holes.

PMOS (P-channel MOS)

Refers to the order in which the semiconductor is doped in a MOS device. That is, which structures are constructed as P-type versus N-type material.

PNP (P-type - N-type - P-type)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the order in which the semiconductor is doped in a bipolar junction transistor.

Polysilicon Layer

An internal layer in an integrated circuit used to create the gate electrodes of MOS transistors. In addition to forming gate electrodes, the polysilicon layer can also be used to interconnect components. There may be several polysilicon layers separated by dielectric (insulating) layers.

Populating

The act of attaching components to a substrate.

Positive Logic

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more positive potential is considered to represent TRUE and the more negative potential is considered to represent FALSE. Also known as positive true logic.

Positive Resist

A process where radiation passing through the transparent areas of a mask causes previously cured resist to be degraded. The degraded areas are then removed using an appropriate solvent.

Positive-Edge

A transition from a logic 0 to a logic 1. Also known as a rising edge.

Positive-True

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more positive potential is considered to represent TRUE and the more negative potential is considered to represent FALSE. Also known as positive logic.

Power Amplifier

A class of amplifier with the primary purpose of delivering high output power (usually accompanied by significant dissipated power).

Power Plane

A conducting layer in or on the substrate providing power to the components. There may be several power planes separated by insulating layers.

Prepreg

Non-conducting semi-cured layers of FR4 used to separate conducting layers in a multilayer circuit board.

Primitives

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simple logic functions such as BUF, NOT, AND, NAND, OR, NOR, XOR, and XNOR may be referred to as primitive logic gates or primitives.

Product Term

A set of literals linked by an AND operator.

Programmable Array Logic (PAL)

A programmable logic device in which the AND array is programmable but the OR array is pre-defined.

Programmable Logic Array (PLA)

The most user-configurable of the traditional programmable logic devices, because both the AND and OR arrays are programmable.

Programmable Logic Device (PLD)

The generic name for a device constructed in such a way that the designer can configure, or "program" it to perform a specific function.

Programmable Read-Only Memory (PROM)

A programmable logic device in which the OR array is programmable but the AND array is predefined. Usually considered to be a memory device whose contents can be electrically programmed (once) by the designer.

Project

A container that groups designs and their associated settings, including report definitions, in a file with a .adsn extension. To the fullest extent possible, projects are portable in that they include, rather than merely refer to, the library elements (graphical symbols, materials, footprints, and so on) of the components and models they contain. Multiple projects can be open simultaneously.

PROM

See Programmable Read-Only Memory.

Pseudo-Random

An artificial sequence of values that give the appearance of being random.

PTH

See Plated Through-Hole.

P-type

A piece of semiconductor doped with impurities that make it amenable to accepting electrons.

Pulling

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The difference between the maximum frequency of a VCO when the phase angle of the load impedance reflection coefficient varies through 360 degrees.

Pulsed Radar

A transmit and receive system used for ranging and detection that transmits a train of short bursts of high microwave signals and receives return signals reflected from a target.

Pushing

The change in frequency when the supply voltage changes, expressed in MHz/V.

Glossary: Q

Q

Quality factor; a measure of stored energy/dissipated energy. Also a measure of bandwidth.

QAM

Quadrature amplitude modulation.

QFP

See Quad Flat Pack.

QPSK

Quadrature phase-shift keying.

Quad Flat Pack (QFP)

The most commonly used package in surface mount technology to achieve a high lead count in a small area. Leads are presented on all four sides of a thin square package.

Quinary

Base-5 numbering system.

Glossary: R

Radio Frequency (RF)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The range in the electromagnetic spectrum loosely defined from 30 MHz to 3 GHz (with corresponding wavelengths from 1000 cm to 10 cm).

Radix

Refers to the number of digits in a numbering system. For example, the decimal numbering system is said to be radix-10. May also be referred to as the "base".

Rats Layer

A non-physical, default layer in the stackup that displays a drawing of logical connections between different components, circuit elements, and net connections. A single connection is called a rat and all of the connections on a rat layers is a rat's nest.

Reed-Müller Logic

Logic functions implemented using only XOR and XNOR gates.

Refractory Metal

Metals such as tungsten, titanium, and molybdenum which are capable of withstanding extremely high, or refractory, temperatures.

Remotely Reconfigurable Hardware

A product whose function may be customized remotely, by telephone or radio, while remaining resident in the system (see also Configurable Hardware, Reconfigurable Hardware, Dynamically Reconfigurable Hardware, and Virtual Hardware).

Reflection

The appearance of a previously transmitted signal on the transmission line causing interference with the current signal. Reflections are caused by a poorly terminated or discontinuous transmission line, where the signal energy is not fully absorbed within the receiver and is therefore transmitted back toward the transmitter.

Resist

A material which is used to coat the substrate and is then selectively cured to form an impervious layer.

Resistor-Transistor Logic (RTL)

Logic gates implemented using particular configurations of resistors and bipolar junction transistors. For the majority of today's designers, resistor-transistor logic is of historical interest only.

Return Loss

Return Loss (dB) is defined as a ratio of the incoming signal to the same reflected signal as it enters a component.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Loss (dB) = 10 * LOG10(Reflected Power/Incident Power)

RF

See Radio Frequency.

RF Power

A class of engineering. Circuits and signals primarily concerned with power levels ranging from a few watts to tens of thousands of watts in the RF spectrum.

RF Power Transmitter

A discrete packaged transistor used in the amplification of RF power.

Rigid Flex

Hybrid constructions which combine standard rigid circuit boards with flexible printed circuits, thereby reducing the component count, weight, and susceptibility to vibration of the circuit, and greatly increasing its reliability.

Ringing

Common name for the waveform that is seen when a transmission line ends at a high impedance discontinuity. The signal first overshoots, then dips down below the target value, and continues this with decreasing amplitude until it converges on the target voltage.

Risetime

The time it takes for a signal to rise from 10% of its total logic swing to 90% of its total logic swing.

Rising-Edge

A transition from a logic 0 to a logic 1. Also known as a positive edge.

RTL

See Resistor-Transistor Logic.

Glossary: S

Sample Rate

Time increment of analysis. Sometimes referred to as sampling rate.

Sampling

The process of converting an analog signal into a series of digital values.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Notation

A notation in which each signal is assigned a unique name; for example, a3, a2, a1, and a0.

Scaling

A technique for making transistors switch faster by reducing their size. This strategy is known as scaling, because all of the transistors features are typically reduced by the same proportion.

Schematic

Common name for a circuit diagram.

Scrubbing

The process of vibrating two pieces of metal, or metal coated materials, at ultrasonic frequencies to create a friction weld.

Seed Value

An initial value loaded into a linear feedback shift register or random number generator.

Sensor

A transducer that detects a physical quantity and converts it into a form suitable for processing. For example, a microphone is a sensor which detects sound and converts it into a corresponding voltage or current.

Sequential

A function whose output value depends not only on its current input values, but also on previous input values. That is, the output value depends on a sequence of input values.

Side-Emitting Laser Diode

A laser diode constricted at the edge of an integrated circuit's substrate such that, when power is applied, the resulting laser beam is emitted horizontally; that is, parallel to the surface of the substrate.

Sign Bit

The most significant binary digit, or bit, of a signed binary number. If set to a logic 1, this bit represents a negative quantity.

Signal Conditioning

Amplifying, filtering, or otherwise processing a signal.

Signal Layer

A layer carrying tracks in a circuit board, hybrid, or multichip module. See also wiring layer.

Signature

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the checksum value from a cyclic-redundancy-check when used in the guided-probe form of functional test.

Signature Analysis

A guided-probe functional-test technique based on signatures.

Signed Binary Number

A binary number in which the most-significant bit is used to represent a negative quantity. Thus, a signed binary number can be used to represent both positive and negative values.

Sign-Magnitude

Negative numbers in standard arithmetic are typically represented in sign-magnitude form by prefixing the value with a minus sign; for example, -27. For reasons of efficiency, computers rarely employ the sign-magnitude form. Instead, they use signed binary numbers to represent negative values.

Silicon Bumping

The process of depositing additional metallization on a die's pads to raise them fractionally above the level of the Barrier Layer.

Silicon Chip

Although a variety of semiconductor materials are available, the most commonly used is silicon and integrated circuits are popularly known as silicon chips, or simply chips.

Silicon Compiler

The program used in compiled cell technology to generate the masks used to create components and interconnections. May also be used to create data-path functions and memory functions.

Single-Sided

A printed circuit board with tracks on only one side.

Sintering

A process in which ultra-fine metal powders weld together at temperatures much lower than those required for larger pieces of the same materials.

SIPO (Serial-In Parallel-Out)

Refers to a shift register in which the data is loaded in serially and read out in parallel.

SISO (Serial-In Serial-Out)

Refers to a shift register in which the data is both loaded in and read out serially.

Skew

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time delay between different bits transmitted at the same time, measured at the receiver.

Skin Effect

In the case of high frequency signals, electrons are only conducted on the outer surface, or skin, of a conductor. This phenomenon is known as the skin effect.

Small-Scale Integration (SSI)

Refers to the number of logic gates in a device. By one convention, small-scale integration represents a device containing 1 to 12 gates.

SMD

See Surface Mount Device.

SMOBC

See Solder Mask Over Bare Copper.

SMT

See Surface Mount Technology.

SNR

Signal-to-noise ratio.

Soft Macro (Macro Function)

A logic function defined by the manufacturer of an application-specific integrated circuit. The function is described in terms of the simple functions provided in the cell library and connections between them. The assignment of cells to basic cells and the routing of the tracks is determined at the same time, and using the same tools, as for the other cells specified by the designer.

Solder Bumping

A flipped chip technique in which spheres of solder are formed on the die's pads. The die is flipped and the solder bumps are brought into contact with corresponding pads on the substrate. When all the chips have been mounted on the substrate, the solder bumps are melted using reflow soldering or vapor-phase soldering.

Solder Mask

A layer applied to the surface of the substrate that prevents solder from sticking to any metallization except where holes are patterned into the mask.

Solder Mask Over Bare Copper (SMOBC)

A technique in which the solder mask is applied in advance of the tin-lead plating. This results in lighter circuit boards because the tin-lead alloy is only used to plate the pads.

Solution

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A solution is the successful result of an analysis, or imported results available for plotting.

Space

Used to refer to the width of the gap between adjacent tracks.

SPICE

Simulated Program for Integrated Circuit Emulation

SRAM

See Static RAM.

SS-CDMA

Spread-spectrum code-division multiple access.

SSI

See Small-Scale Integration.

Stackup

An arrangement of physical signal and dielectric layers that is used in the design of circuit boards. In Circuit, the stackup editor lists conceptual non-stackup layers with the physical stackup layers.

Standard Cell

An application-specific integrated circuit which, unlike a gate array, does not use the concept of a basic cell and does not have any pre-fabricated components. The manufacturer creates custom masks for every stage of the device's fabrication allowing each logic function to be created using the minimum number of transistors.

State Assignment

The process by which the states in a state machine are assigned to the binary patterns that are to be stored in the state variables.

State Diagram

A graphical representation of the operation of a state machine.

State Machine

The actual implementation (in hardware or software) of a function that can be considered to consist of a set of states through which it sequences.

State Table

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A tabular representation of the operation of a state machine. Similar to a truth table, but also includes the current state as an input and the next state as an output.

State Transition

An arc connecting two states in a state diagram.

State Variable

One of a set of registers whose values represent the current state occupied by a state Static Flex.

Statement

A sentence that asserts or denies an attribute about an object or group of objects.

Static Flex

A type of flexible printed circuit which can be manipulated into permanent three-dimensional shapes for applications such as calculators and high-tech cameras which require efficient use of volume and not just area.

Static RAM (SRAM)

A memory device in which each cell is formed from four or six transistors configured as a latch or a flip-flop. The term static is used because, once a value has been loaded into an SRAM cell, it will remain unchanged until it is explicitly altered or until power is removed from the device.

Steady State

A condition in which nothing is changing or happening.

Subatomic Erosion

A process in which structures on an integrated circuit's substrate are eroded by the flow of electrons in much the same way as land is eroded by a river (also known as electromigration)

Substrate

Generic name for the base layer of an integrated circuit, hybrid, multichip module, or circuit board. Substrates may be formed from a wide variety of materials, including semiconductors, ceramics, FR4 (fiberglass), glass, sapphire, or diamond depending on the application. Note that the term substrate has traditionally not been widely used in the circuit board world, at least not by the people who manufacture the boards. However, there is an increasing tendency to refer to a circuit board as a substrate by the people who populate the boards. The main reason for this is that circuit boards are often used as substrates in hybrids and multichip modules, and there is a trend toward a standard terminology across all forms of interconnection technology.

Subtractive Process

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A process in which a substrate is first covered with conducting material, then any unwanted material is subsequently removed, or subtracted.

Superconductor

A material with zero resistance to the flow of electric current.

Surface Mount Device (SMD)

A component whose packaging is designed for use with surface mount technology.

Surface Mount Technology (SMT)

A technique for populating hybrids, multichip modules, and circuit boards, in which packaged components are mounted directly onto the surface of the substrate. A layer of solder paste is screen printed onto the pads and the components are attached by pushing their leads into the paste. When all of the components have been attached, the solder paste is melted using either reflow soldering or vapor-phase soldering.

Surface-Emitting Laser Diode

A laser diode constricted on an integrated circuit's substrate such that, when power is applied, the resulting laser beam is emitted directly away from the surface of the substrate.

Sweep Definition

Also called *variable sweep definition*. A set of variable values within a range that Optimetrics drives the Electronics Desktop to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

Synchronous

(1)A signal whose data is not acknowledged or acted upon until the next active edge of a clock signal. (2)A system whose operation is synchronized by a clock signal.

System Gain

The net loss of a system as a measure of reliability with respect to system parameters. It measures the difference between the output power and minimum input power required for satisfactory performance. It represents a system net loss, and is represented as a negative dB value that is larger than or equal to the summed gains and losses of a signal propagating in a system from transmitter to receiver.

Glossary: T

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tap

A register output which is used to generate the next data input to a linear feedback shift register.

Tape Automated Bonding (TAB)

A process in which transparent flexible tape has tracks created on its surface. The pads on unpackaged integrated circuits are attached to corresponding pads on the tape which is then stored in a reel. Silver-loaded epoxy is screen printed on the substrate at the site where the device is to be located and onto the pads to which the device's leads are to be connected. The reel of TAB tape is fed through an automatic machine which pushes the device and the TAB leads into the epoxy. When the silver-loaded epoxy is cured using reflow soldering or vapor-phase soldering, it forms electrical connections between the TAB leads and the pads on the substrate.

TDM

Time-division multiplexing.

TDMA

Time-division multiple access.

Technology File

A technology file is a collection of information, specifiable by name at design-creation time using the Choose Layout Technology dialog box, that specifies material stackup properties for layout, substrates for simulation, and component libraries for design creation. A technology (.asty) file may be saved from the current project via the File menu. A technology file initializes a design with a set of data to avoid repeated entry of commonly used data. This data can consist of layers and stackup information for layout, configured libraries of components, and substrate definition (s) for circuit analysis. Users and foundries can customize Technology files for their own manufacturing process and simulation models.

Tertiary Logic

An experimental technology in which logic gates are based on three distinct voltage levels. The three voltages are used to represent the tertiary digits 0, 1, and 2, and their logical equivalents FALSE, TRUE, and MAYBE.

Thermal Impedance

Relates the temperature rise for a given dissipated power (which employs an analogy to the voltage-current relationship of impedance).

Thermal Relief Pad

A special pattern etched around a via or a plated through-hole to connect it into a power or ground plane. A thermal relief pad is necessary to prevent too much heat being absorbed into the power or ground plane when the board is being soldered.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Tracking

Typically used to refer to the problems associated with optical interconnection systems whose alignment may be disturbed by changes in temperature.

Thevenin's theorem

Any two-terminal combination of voltage sources and resistors is electrically equivalent to a single voltage source in series with a single resistor.

Thick-Film Process

A process used in the manufacture of hybrids and, to a lesser extent, multichip modules in which signal and dielectric (insulating) layers are screen-printed onto the substrate.

Thin-Film Process

A process used in the manufacture of hybrids and multichip modules in which signal layers and dielectric (insulating) layers are created using opto-lithographic techniques.

Time-Of-Flight

The time taken for a signal to propagate from one logic gate or opto-electronic component to another.

Tin-Lead Plating

An electroless plating process in which exposed areas of copper on a circuit board are coated with a layer of tin-lead alloy. The alloy is used to prevent the copper from oxidizing and provides protection against contamination.

Tinning

An abbreviation of tin-lead plating, which is an electroless plating process in which exposed areas of copper on a circuit board are coated with a layer of tin-lead alloy. The alloy is used to prevent the copper from oxidizing and provides protection against contamination.

Toggle

Refers to the contents or outputs of a logic function switching to the inverse of their previous logic values.

TOI

Third-order intercept point.

Total Voltage

Total voltage = Incident voltage + reflected voltage, that is $V^T = V^a + V^b$.

Trace

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A conducting connection between electronic components. May also be called a track or a signal. In the case of integrated circuits, such interconnections are often referred to collectively as metallization.

Tracks

A conducting connection between electronic components. May also be called a trace or a signal. In the case of integrated circuits, such interconnections are often referred to collectively as metallization.

Transducer

A device that converts input energy of one form into output energy of another.

Transistor

A three-terminal semiconductor device that, in the digital world, can be considered to operate like a switch.

Tri-State Function

A function whose output can adopt three states: 0, 1, and Z (high-impedance) The function does not drive any value in the Z state and, in many respects, may be considered to be disconnected from the rest of the circuit.

Truth Table

A convenient way to represent the operation of a digital circuit as columns of input values and their corresponding output responses.

TTL (Transistor-Transistor Logic)

Logic gates implemented using particular configurations of bipolar junction transistors.

Glossary: U

ULA

See Uncommitted Logic Array.

Ultra-Large-Scale Integration (ULSI)

Refers to the number of logic gates in a device. By one convention, ultra-large-scale integration represents a device containing a million or more gates.

ULSI

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Ultra-Large-Scale Integration.

Uncommitted Logic Array (ULA)

One of the original names used to refer to gate array devices. This term has largely fallen into disuse.

Unsigned Binary Number

A binary number in which all the bits are used to represent positive quantities. Thus, an unsigned binary number can only be used to represent positive values.

Undershoot

The percentage a waveform falls below its lowest determined value before setting at the correct value.

Glossary: V

Vapor-Phase Soldering

A surface mount process in which a substrate carrying components attached by solder paste is lowered into the vapor-cloud of a tank containing boiling hydrocarbons. This melts the solder paste thereby forming good electrical connections. However, vapor-phase soldering is becoming increasingly less popular due to environmental concerns.

Vaporware

Refers to either hardware or software that exist only in the minds of the people who are trying to sell them to you.

Variable Sweep Definition

Also called *sweep definition*. A set of variable values within a range that Optimetrics drives HFSS or Q3D to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

Vector Notation

A notation used in logic simulation and synthesis in which a single name is used to reference a group of signals, and individual signals within the group are referenced by means of an index; for example, a[3:0] = a[3], a[2], a[1], and a[0].

Very-Large-Scale Integration (VLSI)

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the number of logic gates in a device. By one convention, very-large-scale integration represents a device containing 1,000 to 999,999 gates.

Via

A hole filled or lined with a conducting material which is used to link two or more conducting layers in a substrate.

Virtual Hardware or Virtual Logic

An extension of dynamically configurable hardware based on a new generation of FPGAs which were introduced around the beginning of 1994. In addition to supporting the dynamic reconfiguration of selected portions of the internal logic, these devices also feature: no disruption to the device's inputs and outputs; no disruption to the system-level clocking; the continued operation of any portions of the device that are not undergoing reconfiguration; and no disruption to the contents of internal registers during reconfiguration, even in the area being reconfigured (see also Configurable Hardware, Reconfigurable Hardware, Remotely Reconfigurable Hardware, and Dynamically Reconfigurable Hardware).

Virtual Memory

A trick used by a computer's operating system to pretend that it has access to more memory than is actually available. For example, a program running on the computer may require ten mega-bytes to store its data, but the computer may have only five mega-bytes of memory available. To get around this problem, whenever the program attempts to access a memory location that does not physically exist, the operating system performs a slight-of-hand and exchanges some of the contents in the memory with data on the hard disk.

VLSI

See Very-Large-Scale Integration.

Volatile

Refers to a memory device which loses any data it contains when power is removed from the system; for example, random-access memory in the form of SRAM or DRAM

Glossary: W

Wafer Probing

The process of testing individual integrated circuits while they still form part of a wafer. An automated tester places probes on the device's pads, applies power to the power pads, injects a

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

series of signals into the input pads, and monitors the corresponding signals returned from the output pads.

Wave Soldering

A process used to solder circuit boards populated with through-hole components. A wave generating mechanism maintains a wave of hot, liquid solder traveling back and forth across the surface of a tank. The populated circuit boards are passed over the wave soldering machine on a conveyor belt. The velocity of the convener belt is carefully controlled and synchronized such that the solder wave brushes across the bottom of the board only once.

Waveguide

A transparent path bounded by non-transparent, reflective areas, which is fabricated directly onto the surface of a substrate. Used in the optical interconnection strategy known as guided-wave.

W-CDMA

Wideband code-division multiple access. Typically defined with 5 MHz channels and 3.84 MHz carrier signals.

Wire Bonding

The process of connecting the pads on an unpackaged integrated circuit to corresponding pads on a substrate using wires that are finer than a human hair. Wire bonding may also be used to connect the pads on an unpackaged integrated circuit, hybrid, or multichip module to the leads of the component package.

Wiring Layer

A layer carrying wires in a discrete wired board. See also signal layer.

Word

A group of signals or logic functions performing a common task and carrying or storing similar data; for example, a value on the data bus could be referred to as a data word.

Glossary: X

X-Ray Lithography

Similar in principle to optical lithography, but capable of constructing much finer features due to the shorter wavelengths involved. However, X-ray lithography requires an intense source of X-rays, is more difficult to use, and is considerably more expensive than optical lithography.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: Y

Yield

The number of devices that work as planned, specified as a percentage of the total number actually fabricated.

Glossary: Z

Zepto (z)

The symbol used to represent the high-impedance state in tri-state logic.

Zone

A spatial area on a printed circuit board that may contain a subset or all of the layers in the board’s stackup.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

#

1D objects 5-110

3D modeler window 2-90

- history tree 2-92
  - commands 2-104
  - controlling the view 2-96
  - organizing 2-95
  - selecting objects 2-101
  - upgrading modeling kernel 2-105

3D models

- changing background 6-2
- clip planes 6-52
- curved surface 6-51
- fitting 6-37
- hiding objects 6-39
- lighting 6-56
- modifying view 6-1
- orientation 6-18
- panning 6-8
- rendering solids 6-47
- rendering wireframes 6-47
- rotating 6-25
- showing objects 6-41
- spinning 6-25
- user interface options 6-2
- zooming 6-9

A

aborting an analysis 8-193

ACT extensions 2-41, 2-156

Active S parameter 17-115

Active S Parameters

- technical details 17-115

Active VWSR parameter 17-115

Active Y Parameter 17-115

Active Z parameter 17-115

admittance matrix, plotting parameters 17-115

analysis

- aborting 8-193
- distributed 8-15
- monitoring solution process 16-10
- parametric (large scale DSO) 8-28
- remote 8-6

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

analysis configurations
in the registry 3-88
analyze all 16-4
animations 17-8
anisotropic tensors 4-19
anisotropy tensors
defining conductivity 4-17, 4-18, 4-20
Ansys Discovery
integration with 5-151
link options 3-37
Ansys Electronics Desktop
batchoptions 2-118, 2-136, 2-144, 2-152
files 3-1
getting started with 2-1
launching 2-5
menu bar 2-45
overview 2-11
ribbon interface 2-12, 2-16, 2-156, 3-15
running from a command line 2-118, 2-136, 2-144, 2-152
running from Windows remote terminal 2-155
status bar 2-106
system requirements 2-5
user interface 2-11
window displays 2-19
Ansys Product Improvement Program 2-8
Ansys SpaceClaim
integration with 5-150
Ansys technical support 1-19
Ansys Workbench
integration with 19-1
material data transfer 19-32
archive
restoring 3-235
saving project as 3-232
auto-complete
variables and properties 2-113
auto-complete, variables and properties 2-113
auto-save 3-134

B

background color 6-2
batchoptions 2-118, 2-136, 2-144, 2-152
command line examples 3-84
desktop settings 2-132
directories and library paths 2-131
examples 2-129
file format 3-84
in the registry 3-88
override registry entry 2-135
temp directory 2-132

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

beta options 3-7

bondwires

drawing 5-74

C

calculated expressions, plotting 17-144

capacitance 17-115

category, traces 17-115

characteristic impedance, plotting 17-115

chip model analyzer (CMA) 17-333

clean stop 8-193, 16-21

cleaning up solutions 17-6

clip plane 6-52

clip planes 6-55

clipboard

importing model from 5-255

options 3-52

closed objects 5-110

closing

Property Window 2-37

color scheme

changing 3-11

command properties

modifying 2-34

command-line syntax 2-118, 2-136, 2-144, 2-152

commands

align faces 5-138

chamfer 5-312

copy 5-254

cut 5-258

fillet 5-310

heal 5-133

hide objects 6-2

pan 6-8

paste 5-254

project sheet 5-292

purge history 5-322

redo 3-241

remove edges 5-141

remove faces 5-140

rotate view 6-25

scale 5-269

separate bodies 5-309

show objects 6-2

simplify 5-125

spin view 6-25

stitch sheets 5-138

thicken sheet 5-283

undo 3-241

wrap sheet 5-284

zoom 6-9

component libraries 5-164

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

components
editing definitions 5-207
editing properties 5-203
inserting 5-193
updating 3-240
updating definitions 5-211
viewing properties of 5-198
conductivity, defining anisotropy tensors 4-17, 4-18, 4-20
configuration files
example uses 3-74
user options 3-77
using to set options 3-66
using UpdateRegistry command 3-69
constants
list of 3-256
coordinate systems
default planes 5-379
deleting 5-387
face 5-377
modifying 5-385
moving to end 5-384
object 5-380
overview 5-368
relative 5-370
working 5-369
coordinates
assigning to objects 5-352
expressing 5-386
setting coordinate systems 5-368
copying and pasting objects 5-254
CPU time
viewing for solution tasks 17-1, 17-5, 17-5
vs. real time 17-1, 17-5, 17-5
cross-sections
creating 5-291
curves
equation-based 5-24
settings 3-46
tooltip 3-52

D

dataset expressions 3-310
datasets
adding 3-275
adding 3D 3-279
cloning 3-288
editing 3-286
exporting 3-290
importing 3-284
overview 3-275
plot properties 3-291

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

removing 3-290
using SheetScan 3-293
debug logging 2-158
Delete, Solution setup 15-24
deleting
field overlay plots 17-195
delta temperature units 3-311
derived field quantities, plotting 17-144
design area 2-90
design of experiments 7-136
designs
copying and pasting 3-231
deleting 3-239
design notes 3-242
inserting in project 3-3
nominal 7-1
read-only 3-239
setting up 3-2
updating components 3-240
viewing details in the project tree 2-28
DesignXplorer 7-194
desktop configuration 3-7
targeted 3-7
desktop-engine connection 3-21
directories
options 3-17
distributed analysis
command line options 8-194
optimal settings 8-24
overview 8-15
two-level distribution 8-24
distributed machine configurations, editing and creating 8-16
documentation file
inserting 3-241
drawing objects
overview 5-110
polylines 5-20
Driven Modal solutions, setting 3-3
Driven Terminal solutions, setting 3-3
DSO
large scale 8-28
command line syntax 8-37
configuration 8-46
monitoring 8-45
outputs 8-41
post-processing 8-43
prerequisites 8-30
troubleshooting 8-57
user interface 8-31

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

edges
- creating objects from 5-304
- detaching 5-290
- flattening 5-312
- moving 5-296
- removing 5-141
- rounding 5-310
- selecting 5-342

Eigenmode solutions, setting 3-3

encryption
- 3D components 5-179, 5-223

errors
- disk space warning 3-18

errors,logging 2-158

exporting
- 2D model files 3-192
- 3D model files 3-192
- datasets 3-290
- field overlay plots 17-194
- graphics files 3-226
- materials 4-37
- options files 3-228

expressions
- dataset 3-310
- delta temperature units 3-311
- intrinsic functions 3-305
- operators 3-304
- overview 3-304
- piecewise linear functions 3-307

F

faces
- aligning 5-138
- covering 5-289
- creating objects from 5-303
- detaching 5-290
- moving 5-296
- removing 5-140
- selecting 5-335
- uncovering 5-290

far field region, definition for post processing 17-126

field overlay plots 17-126, 17-144
- modifying phase 17-126
- on lines 17-190
- scalar plots 17-148
- vector plots 17-153

field overlays
- plotting 17-126

field quantities
- phase angle 17-219
- plotting basic 17-126

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plotting derived 17-144
fields calculator
complex vector electric field 17-220
general commands 17-217
input commands 17-206
output commands 17-230
overview 17-196
registers 17-203
scalar commands 17-220
stack commands 17-201
vector commands 17-225
file formats
.dsp 17-194
files
about 3-1
exporting 3-191
importing 3-138, 3-168, 3-168, 3-176, 3-178, 3-178
fitting objects 6-37
functions
defining 3-314
intrinsic 3-305
list of 3-314
piecewise linear 3-307

G

GDSII
importing 3-140
layer mapping files 3-143, 3-143
tech file colors 3-144
GPU 8-25
grids
settings 3-47
group delay, plotting 17-115
grouping
options 3-36
groups
selecting 5-347

H

hide selection 6-39
hiding
windows 2-20
high performance computing
configuring Ansys Electronics for 8-68
diagnostics 8-218
integration 8-65
load sharing facility (LSF) 8-107
Microsoft Windows HPC 8-71
overview 8-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

portable batch system (PBS) 8-117

remote simulation management (RSM) 8-196

schedulers 8-67

terminology 8-5, 8-66

third party schedulers 8-125

High Performance Computing 8-1

highlighting selections 3-32

history tree 2-92

commands 2-104

controlling the view 2-96

options 3-35

organizing 2-95

selecting objects 2-101

Upgrade Version 2-105

upgrading modeling kernel 2-105

HPC

setting options 3-54

I

Icepak 8-1

getting started 3-2, 3-2

introduction 3-2

impedance matrix, plotting parameters 17-115

importing

datasets 3-284

files 3-139

idf files 3-168

idx files 3-178

inserting designs 3-3

interactive scheduler jobs

design types 8-65

DSO configuration 8-61

options 8-59

overview 8-59

intersecting objects 5-303

introduction to Icepak 3-2

J

jobs

aborting 8-193

advanced submission options 8-163

exporting 8-168

importing 8-168

monitoring 8-183

multi-step submission 8-168

scripting 8-181

submitting 8-158, 16-4

submitting using command line 8-136, 8-167

Windows to Linux 8-174

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

K

keyboard shortcuts
- 3D modeler 2-108
- custom 2-111
- desktop 2-107
- general 2-106
- report window 2-109

L

large scale DSO 8-28
- command line syntax 8-37
- configuration 8-46
- monitoring 8-45
- outputs 8-41
- post-processing 8-43
- prerequisites 8-30
- troubleshooting 8-57
- user interface 8-31

legends
- settings 3-49

libraries
- personal 4-39
- system 4-39
- user 4-39

lighting 6-56

linear constraints 7-185

lines
- field plots on 17-190

Linux remote spawn 3-61, 3-63

load sharing facility (LSF) 8-107

log of solution tasks 17-1, 17-5, 17-5

logging errors 2-158

loop inductance 17-115

loop resistance 17-115

M

markers
- settings 3-49
- table settings 3-50
- X-Y settings 3-50

material browser, accessing 4-6, 4-22

materials
- adding new 4-12
- anisotropic tensors 4-19
- assigning 4-1, 4-32
- assigning properties 4-16
- assigning to objects 4-6, 4-22
- copying 4-36
- deleting 4-36
- exporting 4-37
- ferrite 4-12
- filtering 4-37

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

functional properties 4-32
libraries 4-39
linear 4-12
modifying attributes 4-33
properties as expressions 4-31
removing 4-36
searching 4-6
sorting 4-37
thermal modifiers 4-24
validating 4-36
variable properties 4-24
viewing attributes 4-33
viewing definitions 2-27
MATLAB optimization 3-10
measuring modes 5-364
memory, used during solution 17-1, 17-5, 17-5
menu bar 2-45
menus
customizing 2-80
design-specific 2-60
draw menu (layout editor) 2-61
edit menu 2-49
edit menu (3D Layout) 2-49
edit menu (report window) 2-52
file menu 2-47
help menu 2-73
project menu 2-59
shortcut menus 2-76
tools menu 2-62
adding tools 2-65, 2-69
view menu 2-52
view menu (3D Layout) 2-54
view menu (basic) 2-53
window menu 2-71
message manager 2-29
action messages 2-31
automatically expanding 2-30
clearing messages 2-31
hiding 2-31
showing new messages 2-30
Microsoft Windows HPC
commands 2-155
integration with 8-71
modal solutions, selecting 3-3
modeling kernel
upgrading version 2-105
models
analyzing 5-113
cleaning up history of 5-389
drawing 5-1
setting drawing plane 5-387
setting units of measure 5-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

monitor job 8-183

monitoring solutions 16-10

mouse sensitivity 3-30

movement mode 5-356

MPI Vendor, selecting 8-69

multi-step job submission 8-168

N

named expressions, plotting 17-144

near field region, definition for post processing 17-126

number of processors, setting 15-31

O

objects

- assigning color to 5-251
- assigning coordinates to 5-352
- assigning transparency to 5-253
- bondwires 5-74
- boolean operations on 5-251
- boxes 5-53
- center-point arc 5-12
- circles 5-29
- cones 5-59
- connecting 5-291
- creating a list of 5-332
- creating cross-section 5-291
- creating from edges 5-304
- creating from faces 5-303
- creating from intersections 5-302
- cylinders 5-48
- deleting 5-256
- drawing 5-3
- duplicating 5-264
- ellipses 5-32
- equation-based curves 5-24
- equation-based surface 5-41
- fitting 6-37
- generate history of 5-322
- helices 5-66
- hiding 6-39
- imprinting 5-316
- mirroring 5-262
- modifying 5-250
- modifying attributes 2-33
- moving 5-258
- non-model objects 5-79
- offsetting 5-264
- orientation of 5-262
- planes 5-76
- points 5-75
- polygons 5-38
- polyhedra 5-56
- polylines 5-20
- polylnes 5-19

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

purging history of 5-322
rectangles 5-35
rotating 5-260
scaling 5-269
segmented 5-77
selecting 5-324
selecting multiple types 5-346
separating bodies 5-309
showing 6-41
showing and hiding 6-2
snapping 5-91, 5-94
solid 6-47
spheres 5-45
spirals 5-71
splines 5-16
splitting 5-305
straight lines 5-5
subtracting 5-300
sweeping 5-271
three-point arcs 5-10
tori 5-63
types of 5-110
uniting 5-300
wireframe 6-47
opening, field overlay plots 17-195
operators
list of valid 3-304
optimetrics
design of experiments 7-136
linear constraints 7-185
link to DesignXplorer 7-194
optimization analysis 7-14
overview 7-1
parametric analysis 7-2
sensitivity analysis 7-113
statistical analysis 7-126
viewing results 7-188
optimization
choosing an optimizer 7-14
overview 7-14
setting up analysis 7-45
options
3D modeler 3-26
advanced 3-43
CAD 3-37
display 3-31
drawing 3-27
group 3-36
operations 3-28
snapping 3-30
3D user interface 6-2
animation 3-18, 3-45
auto-save 3-134

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis 3-47
beta 3-7
clipboard 3-52
clone 3-28
component 3-23
curve 3-46
digital 3-52
exporting as file 3-228
faceting 3-43
general 3-5
  component libraries 3-23
  default units 3-21
  desktop configuration 3-7
  desktop performance 3-18
  directories 3-17
  miscellaneous 3-10
  project 3-9
  remote analysis 3-21
  user interface 3-11
grid 3-47
group 3-36
history tree 3-35
HPC and analysis 3-54
legend 3-49
marker table 3-50
model edit 3-28
model save 3-28
mouse sensitivity 3-30
notes 3-48
object appearance 3-34
object outline 3-34
object transparency 3-34
operation history 3-32
polyline 3-27
pre/post-processing 3-18
relative coordinate system 3-27
rendering 3-34
report headers 3-48
report markers 3-49, 3-50
report table 3-53
report update 3-18
Report2D 3-46
reporter 3-44
  fields reporter 3-45
  report setup 3-44
reports 3-52
RSM 3-21
selection 3-32
setting via configuration files 3-66, 3-69
significant digits 3-44
snap 3-30
stacked 3-51
UDM/UDP 3-43

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

output variables
adding 17-123
deleting 17-125

P

password manager 2-117
personal library 4-39
phase
modifying for field overlays 17-126
PinToPin utility 17-333
planes 5-76
plots
2D polar 17-74
2D rectangular 17-60
2D rectangular stacked 17-62
3D rectangular 17-64
data tables 17-75
deleting field overlays 17-195
limit lines in 17-100
named expressions 17-144
opening field overlays 17-195
rectangular contour 17-71
saving field overlays 17-194
y markers 17-85
plotting
basic quantities 17-126
derived quantities 17-144
points 5-75
port Zo 17-115
portable batch system (PBS) 8-117
post processing
overview of options 17-1
viewing profile data 17-1, 17-5, 17-5
preview images, choosing 3-136
printing 3-243
pro, premium, enterprise licensing 3-7
profile information, viewing 17-1, 17-5, 17-5
progress window 2-40
Progress window, monitoring solutions 16-10
project manager window 2-24
project tree
automatically expanding 2-26
field overlays 17-126
field plot folders 17-126
overview 2-25
viewing design details 2-28
visualization options 3-11
projection view 6-58
projects
archiving 3-232
copying and pasting 3-231
creating new 3-128

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default names 3-1
deleting 3-239
design notes 3-242
example projects 1-11, 3-244
exporting 3-191
managing 3-1
opening 3-107
renaming 3-232
saving 3-132
validating 3-228
propagation constant, plotting 17-115
properties
auto-completing 2-113
properties dialog box 2-37
opening 2-37
properties window 2-32
automatically opening 2-37
modifying command properties 2-34
modifying object attributes 2-33
showing and hiding 2-33
properties, Icepak object 9-1
Property Window for Schematic and Layout Editors 2-34
Property window, closing 2-37
purge history 5-322

Q

quantities
plotting S-parameter 17-115
quantity categories 17-115

R

RAM, used during solution 17-1, 17-5, 17-5
range functions 17-117
read-only designs 3-239
redo command 3-241
regions
drawing 5-80, 5-83, 5-88
registry entry
overriding 2-135
release
obtaining information about 2-75
remote analysis
overview 8-6
troubleshooting 8-10
remote simulation management (RSM) 8-196
remote spawn (Linux) 3-61, 3-63
report window 2-91
reports
context 17-36
creating 17-30

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

creating a quick report 17-31
custom templates 17-51
data tables 17-75
file formats 17-56
modifying 17-40
modifying properties 17-45
types of 17-57
user defined documents (UDDs) 17-299
user defined outputs (UDOs) 17-256
zooming and fitting 17-45
results window 2-91
run commands 2-118, 2-136, 2-144, 2-152

S

S parameter
Active S Parameter 17-115
save options 3-7
saving, field overlay plots 17-194
scalar field plots
creating 17-148
schedulers
Ansys RSM 8-196
integration using IronPython 8-148
LSF 8-107
Microsoft Windows HPC 8-71
overview 8-67
PBS 8-117
proxy interfaces 8-137
selecting 8-156
terminology 8-5, 8-66
testing custom integration 8-145
testing integration 8-142
third party 8-125
selecting
edges 5-342
faces 5-335
groups 5-347
multi select 5-346
objects 5-324
objects behind objects 5-349
showing selected objects 6-41
submodels 5-347
vertices 5-344
sensitivity analysis 7-113
choosing variables 3-272
setting up designs 3-2
setting up projects 3-2
setups
solution 15-1, 15-1, 18-1, 18-17
SheetScan 3-293
checking monotonicity 3-301

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defining characteristic curve 3-299

exporting data 3-303

importing characteristic data 3-301

loading datasheets 3-296

shortcut menus 2-76

3D modeler 2-77

project manager 2-78

significant digits

setting 3-44

simulations

aborting 2-41

changing solution priority 16-19

monitoring 16-10

monitoring queued 16-9

re-solving after modifying design 16-22

running 16-1

running multiple 16-4

stopping 2-41, 16-21

solid objects 5-110

solution data

deleting 17-7

viewing 17-1

solution process

tetrahedra used during 17-1, 17-5, 17-5

viewing memory used during 17-1, 17-5, 17-5

viewing profile data 17-1, 17-5, 17-5

viewing tasks performed 17-1, 17-5, 17-5

solution quantities 17-115

solution settings

number of processors 15-31

specifying 15-1, 15-1, 18-17

transient 18-1

solution setup

deleting 15-24

solution setups

adding 15-1

solution type, setting 3-3

solutions

aborting 16-21

changing priority 16-19

monitoring 16-10

re-solving after modifying design 16-22

solving 16-1

Solutions window, displaying 16-10

S-parameters, plotting quantities 17-115

spinning the view 6-31

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

statistical analysis 7-126
choosing variables 3-273
status bar 2-106
streamline
plot selection 17-126
submit job 8-158, 16-4
surfaces
curved 6-51
equation-based 5-41
sweeping
along a path 5-275
along a vector 5-273
around an axis 5-271
draft types 5-280
faces along normal 5-280
objects 5-271
sweeps
orphaned 3-268
variable 3-268
system library 4-39
system resources
changing solution priority 8-223
requirements 2-5

T

TDR Impedance
plotting 17-115
technical support 1-19
temporary directory
configuration file format 3-82
setting 3-80
using UpdateRegistry command 3-83
terminal solutions, setting 3-3
Test Machines, Distributed Analysis Machines dialog 8-16
tetrahedra
used during solution 17-1, 17-5, 17-5
time
real vs. CPU 17-1, 17-5, 17-5
viewing for solution tasks 17-1, 17-5, 17-5
traces
adding characteristics to 17-92
adding data markers 17-83
categories 17-115
editing properties 17-78
overview 17-77
removing 17-98
y markers 17-85
tuning 3-271, 7-174
two-level distribution
guidelines 8-24

U

undo command 3-241

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

units of measure
setting default 3-21
user defined documents (UDDs) 17-299
user defined outputs (UDOs) 17-256
user library 4-39
user-defined models (UDMs) 5-143
user-defined primitives (UDPs) 5-141
UV isolines 3-32

V

validation check 3-228
variables
and object parameters 5-110
and properties, auto-complete 2-113
auto-completing 2-113
constants 3-256
copying and pasting list 3-274
design-level 3-257
arrays 3-263
defining 3-258
editing 3-258
overriding 3-258
fixed vs. non-fixed 3-268
in sensitivity analysis 3-272
in statistical analysis 3-273
intrinsic 3-255
modifying value of a fixed 7-185
optimizing 3-269
output 17-123
overview 3-244
project-level
adding 3-246
arrays 3-250
deleting 3-246
editing 3-246
sweeping 17-107
tuning 3-271, 7-174
vector field plots
creating 17-155
vertices 5-344
view options
spin 6-31
view navigation options 2-110
view options
rotate 6-25
voltage standing wave ratio, plotting 17-115

W

window displays
3D modeler 2-90
ACT extensions 2-41

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

message manager 2-29
progress window 2-40
project manager 2-24
properties window 2-32
report window 2-91
results window 2-91
status bar 2-106
windows
auto-hiding 2-19
docking 2-19
moving and resizing 2-19
showing and hiding 2-19
write solver files 16-22

Y

Y-parameters, plotting 17-115

Z

zooming 6-9
Z-parameters, plotting 17-115

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.